Downloaded From : www.EasyEngineering.net

FOR MORE EXCLUSIVE

(Civil, Mechanical, EEE, ECE)

ENGINEERING & GENERAL STUDIES

(Competitive Exams)

TEXT BOOKS, IES GATE PSU's TANCET & GOVT EXAMS
NOTES & ANNA UNIVERSITY STUDY MATERIALS

VISIT

www.EasyEngineering.net

AN EXCLUSIVE WEBSITE FOR ENGINEERING STUDENTS & GRADUATES

EASY ENGINEERING
Apprise Education, Reprise Innovations

**Note : Other Websites/Blogs Owners we requested you, Please do not Copy (or) Republish this Material.

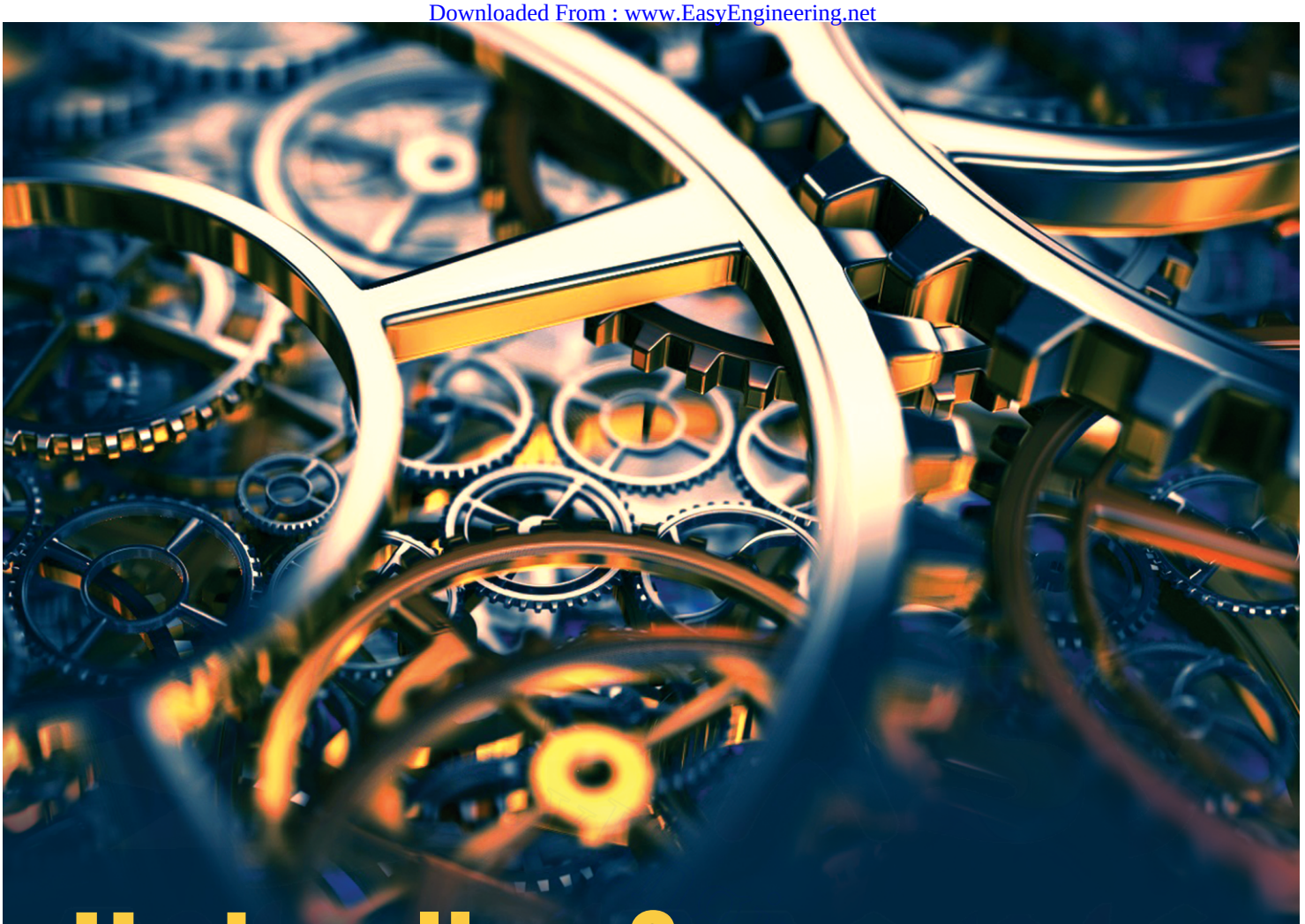
This copy is NOT FOR SALE.

**Disclimers : EasyEngineering does not own this book/materials, neither created nor scanned. we provide the links which is already available on the internet. For any quarries, Disclaimer are requested to kindly contact us. We assured you we will do our best. We DO NOT SUPPORT PIRACY, this copy was provided for students who are financially troubled but deserving to learn.

Thank You and God Bless!

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net



Hydraulics & Fluid Mechanics including Hydraulics Machines

Dr. P.N Modi Dr. S.M Seth

Rajsons Publications Pvt. Ltd.
STANDARD BOOK HOUSE (SINCE 1960)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

HYDRAULICS AND FLUID MECHANICS

Including

HYDRAULIC MACHINES

(In SI Units)

By **Dr. P.N. Modi**

B.E., M.E., Ph.D
Former Professor of Civil Engineering,
M.R. Engineering College, (Now M.N.I.T), Jaipur
Formerly Principal, Kautilya Institute of Technology and Engineering, Jaipur

and

Dr. S.M. Seth

B.E., M.E., M.I.E., Ph.D (Manchester)
Former Director, National Institute of Hydrology, Roorkee
Presently Principal, Kautilya Institute of Technology and Engineering, Jaipur

STANDARD BOOK HOUSE

Unit of : **RAJSONS PUBLICATIONS PVT. LTD.**

1705-A, Nai Sarak, PB. No. 1074, Delhi-110006 Ph.: +91-(011)-23265506
Show Room: 4262/3, First Lane, G-Floor, Gali Punjabian, Ansari Road, Darya Ganj,
New Delhi-110002 Ph.: +91-(011)43551085 Tele Fax : +91-(011)43551185,
Fax: +91-(011)-23250212

E-mail: sbh10@hotmail.com www.standardbookhouse.com

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Published by:

RAJINDER KUMAR JAIN
Standard Book House
Unit of: Rajsons Publications Pvt. Ltd.
1705-A, Nai Sarak, Delhi - 110006
Post Box: 1074
Ph.: +91-(011)-23265506 Fax: +91-(011)-23250212

Showroom:
4262/3. First Lane, G-Floor, Gali Punjabian,
Ansari Road, Darya Ganj,
New Delhi-110002
Ph.: +91-(011)-43551085, +91-(011)-43551185
E-mail: sbhl0@ hotmail.com
Web: www.standardbookhouse.com

Twenty First Edition : 2017
(*Revised and Enlarged*)

© Publishers

All rights are reserved with the Publishers. This book or any part thereof, may not be reproduced, represented, photocopy in any manner without the prior written permission of the publishers.

Price: **Rs. 780.00**

ISBN 978-81-89401-26-9

Typeset by:
C.S.M.S. Computers, Delhi.

Printed by:
R.K. Print Media Company, New Delhi

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Preface to the Twentieth Edition

In this edition the book has been thoroughly revised and enlarged. The Summary of Main Points of the subject matter of the chapter given at the end of each chapter has been very much appreciated by the student community at large. The answers of all Illustrative Examples as well as Problems have been checked.

Authors thank all the learned Professors as well as the students of the various Universities for their appreciation of the book. The authors will appreciate to receive useful suggestions for the improvement of the book and the same will be acknowledged and incorporated in the forthcoming editions of the book.

The authors thank their Publishers Shri Rajinder Kumar Jain and Shri Sandeep Jain for bringing out this edition of the book with very nice get up.

2014 **P.N. MODI**
S.M. SETH

Preface to the First Edition

There is hardly any one book which covers all the topics of Fluids Mechanics, Hydraulics and Hydraulic Machines as required for the examinations of the various Universities and AMIE, IRSE and other competitive examinations. Moreover the encouraging response received by the authors' previous book entitled *Hydraulics and Hydraulic Machines* from the students as well as teachers, has given an impetus to undertake the publication of this text book. In this book an attempt has been made to cover all the topics in the field of Fluid Mechanics as well as Hydraulics and Hydraulic Machines. Every basic principle, method, equation or theory has been presented in a simple and lucid manner which can be understood by the students without any difficulty. The book contains a large number of illustrative examples and equally large number of problems with their answer. In the selection of the solved as well as unsolved examples special care has been taken to include those examples which have appeared in AMIE, IRSE and other competitive examinations.

The text has been developed in two sections. The first portion deals with the Fundamental of Fluid Mechanics and Hydraulics. The advanced topics of Fluid Mechanics and Hydraulics have been dealt with at the end of the chapters after discussing the fundamentals. The second portion deals with the Hydraulic Machines. A complete chapter on *Flow of Compressible Fluids* has been added at the end of the book, because it is normally required to be studied by the students of Mechanical Engineering. Further a chapter introducing the students with SI units has also been added. The book has been made self-contained and therefore it will be useful for the students appearing in the examination of various universities as well as AMIE, IRSE and other competitive examinations. It is hoped that only THIS BOOK will cover the need of the engineering students of all the branches throughout their study of this subject at the undergraduate level.

The metric system has been used throughout this book. However, in order to facilitate the conversion from one system of units to another an Appendix has been added which provides the conversion factors for all the useful quantities in English (FPS), metric (MKS) and SI units. A separate chapter introducing some special flow measuring techniques and the various laboratory expreriments has also been given. An appendix giving a comparative study of incompressible and compressible fluid flows has been given to provide a better understanding of these two different types of flows. In addition some of the various advanced concepts regarding cavitation, laminar flow (sheet flow) in channels, have also been given in various appendices.

The authors do not claim originality of ideas in any part of the book. The main object in writing this text is to present the subject-matter in a simplified form. Suggestions from the readers for the improvement of the text will be appreciated.

The authors express their gratitude to Prof. R.M. Advani, Prof. M.M. Dandekar, Dr. R.D. Verma for their constant encouragement and valuable guidance. Thanks are due to Shri B.C. Punmia and Shri Jagdish Chandra for giving valuable suggestions. Thanks are also due to Shri Vishwa Nath for preparing excellent diagrams. The full cooperation and understanding of our Publishers Shri Rajinder Kumar Jain is greatly appreciated.

March, 1973 **P.N. MODI S.M. SETH**

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Preface to the First Edition

There is hardly any one book which covers all the topics of Fluids Mechanics, Hydraulics and Hydraulic Machines as required for the examinations of the various Universities and AMIE, IRSE and other competitive examinations. Moreover the encouraging response received by the authors' previous book entitled *Hydraulics and Hydraulic Machines* from the students as well as teachers, has given an impetus to undertake the publication of this text book. In this book an attempt has been made to cover all the topics in the field of Fluid Mechanics as well as Hydraulics and Hydraulic Machines. Every basic principle, method, equation or theory has been presented in a simple and lucid manner which can be understood by the students without any difficulty. The book contains a large number of illustrative examples and equally large number of problems with their answer. In the selection of the solved as well as unsolved examples special care has been taken to include those examples which have appeared in AMIE, IRSE and other competitive examinations.

The text has been developed in two sections. The first portion deals with the Fundamental of Fluid Mechanics and Hydraulics. The advanced topics of Fluid Mechanics and Hydraulics have been dealt with that the end of the chapters after discussing the fundamentals. The second portion deals with the Hydraulic Machines. A complete chapter on *Flow of Compressible Fluids* has been added at the end of the book, because it is normally required to be studied by the students of Mechanical Engineering. Further a chapter introducing the students with SI units has also been added. The book has been made self-contained and therefore it will be useful for the students appearing in the examination of various universities as well as AMIE, IRSE and other competitive examinations. It is hoped that only THIS BOOK will cover the need of the engineering students of all the branches throughout their study of this subject at the under-graduate level.

The metric system has been used throughout this book. However, in order to facilitate the conversion from one system of units to another an Appendix has been added which provides the conversion factors for all the useful quantities in English (fps), metric (MKS) and SI units. A separate chapter introducing some special flow measuring techniques and the various laboratory expreriments has also been given. An appendix giving a comparative study of incompressible and compressible fluid flows has been given to provide a better understanding of these two different types of flows. In addition some of the various advanced concepts regarding cavitation, laminar flow (sheet flow) in channels, have also been given in various appendices.

The authors do not originality of ideas in any part of the book. The main object in writing this text is to present the subject-matter in a simplified form. Suggestions from the readers for the improvement of the text will be appreciated.

The authors express their gratitude to Prof. R.M. Advani, Prof. M.M. Dandekar, Dr. R.D. Verma for their constant encouragement and valuable guidance. Thanks are due to Shri B.C. Punmia and Shri Jagdish Chandra for giving valuable suggestions. Thanks are also due to Shri Vishwa Nath for preparing excellent diagrams. The full cooperation and understanding of our Publishers Shri Rajinder Kumar Jain is greatly appreciated.

March, 1973

P.N. MODI
S.M. SETH

Preface to the Fourteenth Edition

In the fourteenth edition, the book has been thoroughly revised and enlarged. In this edition the book has been brought out in A-4 size thereby considerably enhancing the general get-up of the book. Additional typical problems and a large number of additional Multiple Choice Questions have been added. The answers of all the Illustrative Examples and those of Problems have been checked.

The authors thank all the learned Professors as well as the students of the various Universities for their appreciation of the book. The authors also thank their publishers Shri Rajinder Kumar Jain and Shri Sandeep Jain for bringing out the book with very nice get–up.

P.N. MODI

14th Febraury, 2002

S.M. SETH

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Contents

CHAPTER 1. PROPERTIES OF FLUIDS **1–35**

1.1 Introduction 1
1.2 Definition of a Fluid 2
1.3 Development of Fluid Mechanics 2
1.4 Units of Measurement 3
1.5 Mass Density, Specific Weight, Specific Volume 7
1.6 Specific Gravity 8
1.7 Equation of State: The Perfect Gas 9
1.8 Viscosity 10
1.9 Vapour Pressure 12
1.10 Compressibility and Elasticity 13
1.11 Surface Tension and Capillarity 14
Sumary of Main Points *33*
Problems *34*

CHAPTER 2. FLUID PRESSURE AND ITS MEASUREMENT **36–92**

2.1 Fluid Pressure at a Point 36
2.2 Variation of Pressure in a Fluid 36
2.3 Equilibrium of a Compressible Fluid—Atmospheric Equilibrium 40
2.4 Pressure, Same in all Directions — Pascal's Law 47
2.5 Atmospheric, Absolute, Gage and Vacuum Pressures 48
2.6 Mesurement of Pressure 49
2.7 General Comments on Connections for Manometers and Gages 65
Sumary of Main Points *89*
Problems *90*

CHAPTER 3. HYDROSTATIC FORCES ON SURFACES **93–154**

3.1 Total Pressure and Centre of Pressure 93
3.2 Total Pressure on a Plane Surface 93

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

3.3 Pressure Diagram 102
3.4 Total Pressure on Curved Surface 103
3.5 Practical Applications of Total Pressure and Centre of Pressure 105
Sumary of Main Points *151*
Problems *152*

CHAPTER 4. BUOYANCY AND FLOATATION 155–189

4.1 Buoyancy, Buoyant Force and Centre of Buoyancy 155
4.2 Metacentre and Metacentric Height 157
4.3 Stability of Submerged and Floating Bodies 158
4.4 Determination of Metacentric Height 161
4.5 Metacentric Height for Floating Bodies Containing Liquid 165
4.6 Time Period of Transverse Oscillation of a Floating Body 166
Sumary of Main Points *187*
Problems *188*

CHAPTER 5. LIQUIDS IN RELATIVE EQUILIBRIUM 190–228

5.1 Introduction 190
5.2 Fluid Mass Subjected to Uniform Linear Acceleration 190
5.3 Liquid Containers Subjected to Constant Horizontal Acceleration 193
5.4 Liquid Containers Subjected to Constant Vertical Acceleration 196
5.5 Fluid Containers Subjected to Constant Rotation 199
Sumary of Main Points *227*
Problems *227*

CHAPTER 6. FUNDAMENTALS OF FLUID FLOW 229–285

6.1 Introduction 229
6.2 Velocity of Fluid Particles 229
6.3 Types of Fluid Flow 231
6.4 Description of the Flow Pattern 234
6.5 Basic Principles of Fluid Flow 236
6.6 Continutty Equation 236
6.7 Acceleration of a Fluid Particle 246
6.8 Rotational and Irrotational Motions 251
6.9 Circulation and Vorticity 254
6.10 Velocity Potential 256
6.11 Stream Function 257
6.12 Streamlines, Equipotential Lines and Flow Net 260
6.13 Methods of Drawing Flow Nets 262
6.14 Use of the Flow Net 263
6.15 Limitations of Flow Net 265
Sumary of Main Points *281*
Problems *284*

Final Proof/24.10.2009

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

CHAPTER 7. EQUATIONS OF MOTION AND ENERGY EQUATION 286–350

7.1 Introduction 286
7.2 Forces Acting on Fluid in Motion 287
7.3 Euler's Equation of Motion 288
7.4 Integration of Euler's Equations 291
7.5 Bernoulli's Equation from the Principle of Conservation of Energy 297
7.6 Kinetic Energy Correction Factor 301
7.7 Bernoulli's Equation for a Compressible Fluid 302
7.8 Pressure Velocity Realationship 304
7.9 Applications of Bernoulli's Equation 305
7.10 Venturi Meter 305
7.11 Orifice Meter 310
7.12 Nozzle Meter or Flow Nozzle 313
7.13 Other Flow Measurement Devices 313
7.14 Pitot Tube 314
7.15 Free Liquid Jet 317
7.16 Vortex Motion 319
7.17 Radial Flow or Radial Motion 323
7.18 Spiral Vortex Motion 326
Sumary of Main Points *345*
Problems *348*

CHAPTER 8. IMPULSE MOMENTUM EQUATION AND ITS APPLICATIONS 351–382

8.1 Introduction 351
8.2 Impulse-momentum Equations 351
8.3 Momentum Correction Factor 354
8.4 Applications of the Impulse-Momentum Equation 355
8.5 Force on a Pipe Bend 355
8.6 Jet Propulsion—Reaction of Jet 357
8.7 Momentum Theory of Propellers 362
8.8 Angular Momentum Principle—Moment of Momentum Equation 365
Sumary of Main Points *380*
Problems *381*

CHAPTER 9. FLOW THROUGH ORIFICES AND MOUTHPIECES 383–453

9.1 Definition 383
9.2 Classifications of Orifices and Mouthpieces 383
9.3 Sharp-edged Orifice Discharging Free 384
9.4 Experimental Determination of the Coefficients for an Orifice 388
9.5 Flow Through Large Vertical Orifice 394
9.6 Flow Under Pressure Through Orifices 398
9.7 Flow Through Submerged (or Drowned) Orifice 398

Final Proof/24.10.2009

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

9.8 Energy or Head Losses of Flowing Liquid Due to Sudden Change in Velocity 400
9.9 Flow Through an External Cylindrical Mouthpiece 407
9.10 Flow Through A Convergent Divergent Mouthpiece 411
9.11 Flow Through Internal or Re-Entrant or Borda's Mouthpiece 413
9.12 Flow Through an Orifice or a Mouthpiece Under Variable Heads 416
9.13 Flow of Liquid From one Vessel to Another 421
9.14 Time of Emptying and Filling of a Canal Lock 423
Sumary of Main Points *449*
Problems *452*

CHAPTER 10. FLOW OVER NOTCHES AND WEIRS 454–493

10.1 Introduction 454
10.2 Classification of Notches and Weirs 454
10.3 Flow Over a Rectangular Sharp-Crested Weir or Notch 455
10.4 Calibration of Rectangular Weir or Notch 458
10.5 Empirical Fomula for Discharge over Rectangular Weirs 459
10.6 Ventilation of Weirs 461
10.7 Flow Over a Triangular Weir (v-Notch Weir) or Triangular Notch (v-Notch) 463
10.8 Flow Over a Trapezoidal Weir or Notch 465
10.9 Time Required to Empty a Reservoir with Rectangular Weir 467
10.10 Effect on Computed Discharge over a weir or Notch Due to Error in the Measurement of Head 469
10.11 Broad Crested Weir 470
10.12 Submerged Weirs 472
10.13 Spillway and Siphon Spillway 473
10.14 Proportional Weir or Sutro Weir 475
Sumary of Main Points *490*
Problems *492*

CHAPTER 11. FLOW THROUGH PIPES 494–566

11.1 Introduction 494
11.2 Two Types of Flow—Reynolds' Experiment 494
11.3 Laws of Fluid Friction 497
11.4 Froude's Experiments 498
11.5 Equation for Head Loss in Pipes Due to Friction—Darcy-Weisbach Equation 499
11.6 Other Formulae for Head Loss Due to Friction in Pipes 500
11.7 Other Energy Losses in Pipes 502
11.8 Hydraulic Grade Line and Energy Grade Line 503
11.9 Flow Through Long Pipes 507
11.10 Pipes in Series or Compound Pipe 508
11.11 Equivalent Pipe 509
11.12 Pipes in Parallel 510
11.13 Flow Through a Bye-Pass 511

Final Proof/24.10.2009

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

11.14 Branched Pipes 512
11.15 Siphon 515
11.16 Loss of Head Due to Friction in Tapering Pipe 517
11.17 Loss of Head Due to Friction in a Pipe with Side Tappings 519
11.18 Time of Emptying a Reservoir Through Pipe 520
11.19 Transmission of Power Through Pipes 522
11.20 Flow Through Nozzle at the end of Pipe 523
11.21 Water Hammer in Pipes 526
11.22 Pipe Networks 531
Sumary of Main Points *560*
Problems *564*

CHAPTER 12. BOUNDARY LAYER THEORY 567–600

12.1 Introduction 567
12.2 Thickness of Boundary Layer 567
12.3 Boundary Layer along a Long Thin Plate and its Characteristics 569
12.4 Boundary Layer Equations 571
12.5 Momentum Integral Equation of the Boundary Layer 574
12.6 Laminar Boundary Layer 577
12.7 Turbulent Boundary Layer 580
12.8 Laminar Sublayer 582
12.9 Boundary Layer on Rough Surfaces 582
12.10 Separation of Boundary Layer 583
12.11 Methods of Controlling the Boundary Layer 585
12.11.1 Motion of Solid Boundary 585
12.11.2 Acceleration of the Fluid in the Boundary Layer 585
12.11.3 Suction of the Fluid from the Boundary Layer 586
12.11.4 Streamlining of Body Shapes 586
Sumary of Main Points *598*
Problems *600*

CHAPTER 13. LAMINAR FLOW 601–657

13.1 Introduction 601
13.2 Relation between Shear and Pressure Gradients in Laminar Flow 601
13.3 Steady Laminar Flow in Circular Pipes—Hagen–Poiseuille Law 603
13.4 Laminar Flow Through Inclined Pipes 608
13.5 Laminar Flow Through Annulus 610
13.6 Laminar Flow between Parallel Plates–Both Plates at Rest 612
13.7 Laminar Flow between Parallel Flat Plates—one Plate Moving and Other at Rest—Couette Flow 615
13.8 Laminar Flow of Fluid in an Open Channel 619
13.9 Laminar Flow Through Porous Media 620
13.10 Laminar Flow Around a Sphere—Stokes' Law 622
13.11 Lubrication Mechanics 623

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

13.11.1 Slipper Bearing 623
13.11.2 Journal Bearing 627
13.11.3 Properties of Lubricant 629
13.12 Dash-Pot Mechanism 630
13.13 Measurement of Viscosity—Viscometers 633
Sumary of Main Points *653*
Problems *656*

CHAPTER 14. TURBULENT FLOW IN PIPES **658–700**

14.1 Introduction 658
14.2 Shear Stresses in Turbulent Flow 658
14.3 Formation of Boundary Layer in Pipes—Establishment of Flow in Pipes 661
14.4 Hydrodynamically Smooth and Rough Boundaries 662
14.5 Velocity Distribution for Turbulent Flow in Pipes 663
14.6 Velocity Distribution for Turbulent Flow in Hydrodynamically Smooth and Rough Pipes—Karman Prandtl Velocity Distribution Equation 665
14.7 Velocity Distribution Equation for Turbulent Flow in Terms of Mean Velocity, for Smooth and Rough Pipes 669
14.8 Resistance to Flow of Fluid in Smooth and Rough Pipes 671
14.9 Types of Problems in Pipeline Designs 679
14.10 Friction in Non-Circular Conduits 679
Sumary of Main Points *698*
Problems *700*

CHAPTER 15. FLOW IN OPEN CHANNELS **701–781**

15.1 Introduction 701
15.2 Types of Flow in Channles 702
15.3 Geometrical Properties of Channel Section 703
15.4 Velocity Distribution in a Channel Section 705
15.5 Uniform Flow in Channles 706
15.6 Most Economical or Most Efficient Section of Channel 711
15.7 Open Channel Section for Constant Velocity at all Depths of Flow 719
15.8 Computation of Uniform Flow 721
15.9 Specific Energy and Critical Depth 722
15.10 Momentum in Open-Channel Flow-Specific Force 725
15.11 Critical Flow and its Computation 727
15.12 Application of Specific Energy and Discharge Diagrams to Channel Transitions 731
15.13 Metering Flumes 735
15.14 Determination of Mean Velocity of Flow in Channels 738
15.15 Practical Channel Sections 740
15.16 Measurement of Discharge in Rivers 741
Sumary of Main Points *777*
Problems *779*

Final Proof/24.10.2009

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

CHAPTER 16. NON-UNIFORM FLOW IN CHANNELS 782–835

16.1 Introduction 782
16.2 Gradually Varied Flow 782
16.3 Classification of Channel Bottom Slopes 788
16.4 Classification of Surface Profiles 789
16.5 Characteristics of Surface Profiles 790
16.6 Integration of the Varied Flow Equation 796
16.7 Hydraulic Jump 800
16.8 Location of Hydraulic Jump 805
16.9 Surges in Open Channels 808
Sumary of Main Points *833*
Problems *834*

CHAPTER 17. DIMENSIONAL ANALYSIS, HYDRAULIC SIMILITUDE AND MODEL INVESTIGATION 836–891

17.1 Introduction 836
17.2 Dimensions 836
17.3 Dimensional Homogeneity 840
17.4 Methods of Dimensional Analysis 842
17.5 Outline of Procedure for Buckingham Method 846
17.6 Number of Dimensionless Groups in a Complete Set of Variables 847
17.7 Superfluous and Omitted Variables 849
17.8 Use of Dimensional Analysis in Presenting Experimental Data 850
17.9 Model Investigation 851
17.10 Similitude—Types of Similarties 852
17.11 Force Ratios—Dimensionless Numbers 855
17.12 Similarity Laws or Model Laws 857
17.13 Types of Models 860
17.14 Merits and Limitations of Distorted Models 861
17.15 Scale Effect in Models 861
17.16 Application of Dynamic Similarity to Specific Model Investigations 862
Sumary of Main Points *888*
Problems *889*

CHAPTER 18. FLUID FLOW AROUND SUBMERGED OBJECTS—DRAG AND LIFT 892–937

18.1 Introduction 892
18.2 Types of Drag 895
18.3 Dimensional Analysis of Drag and Lift 898
18.4 Drag on a Sphere 899
18.5 Drag on a Cylinder 903
18.6 Drag on a Flat Plate 909

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

18.7 Drag on an Airfoil 910
18.8 Effect of Free Surface on Drag 911
18.9 Effect of Compressibility on Drag 912
18.10 Development of Lift on Immersed Bodies 914
18.11 Induced Drag on an Airfoil of Finite Length 924
18.12 Polar Diagram for Lift and Drag of an Airfoil 927
Sumary of Main Points *935*
Problems *936*

CHAPTER 19. FLOW OF COMPRESSIBLE FLUIDS 938–977

19.1 Introduction 938
19.2 Basic Relationship of Thermodynamics 938
19.3 ContinuIty Equation 941
19.4 Momentum Equation 941
19.5 Energy Equation 941
19.6 Propagation of Elastic Waves Due to Compression of Fluid, Velocity of Sound 943
19.7 Mach Number and its Significance 945
19.8 Propagation of Elastic Waves Due to Disturbance in Fluid 946
19.9 Stagnation Pressure in Ccompressible Flows 947
19.10 Flow of Compressible Fluid with Negligible Friction Through a Pipe of varying Cross-section 949
19.11 Flow of Compressible Fluid in Convergent—Divergent Passages 951
19.12 Normal Shock Waves 956
19.13 Measurement of Compressible Fluid Flow 958
Sumary of Main Points *974*
Problems *976*

CHAPTER 20. IMPACT OF FREE JETS 978–1020

20.1 Introduction 978
20.2 Force Exerted by Fluid Jet on Stationary Flat Plate 978
20.3 Force Exerted by Fluid Jet on Moving Flat Plate 981
20.4 Force Exerted by a Fluid Jet on Stationary Curved Vane 985
20.5 Force Exerted by a Fluid Jet on Moving Curved Vane 989
20.6 Torque Exerted on a Wheel with Radial Curved Vanes 997
Sumary of Main Points *1017*
Problems *1018*

CHAPTER 21. HYDRAULIC TURBINES 1021–1086

21.1 Introduction 1021
21.2 Elements of Hydroelectric Power Plants 1022
21.3 Head and Efficiencies of Hydraulic Turbines 1023
21.4 Classification of Turbines 1026
21.5 Pelton Wheel 1027

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

21.6 Work Done and Efficiencies of Pelton Wheel 1028
21.7 Working Proportions of Pelton Wheel 1032
21.8 Design of Pelton Turbine Runner 1033
21.9 Multiple Jet Pelton Wheel 1033
21.10 Radial Flow Impulse Turbine 1034
21.11 Reaction Turbines 1034
21.12 Francis Turbine 1035
21.13 Work Done and Efficiencies of Francis Turbine 1037
21.14 Working Proportions of Francis Turbine 1039
21.15 Design of Francis Turbine Runner 1039
21.16 Draft Tube Theory 1040
21.17 Shape of Francis Turbine Runner and Development of Kaplan Turbine Runner 1042
21.18 Kaplan Turbine 1043
21.19 Working Proportions of Kaplan Turbine 1044
21.20 New Types of Turbines 1045
21.21 Governing of Turbines 1047
21.22 Runaway Speed 1050
21.23 Surge Tanks 1050
Sumary of Main Points *1082*
Problems *1084*

CHAPTER 22. PERFORMANCE OF TURBINES 1087–1130

22.1 Introduction 1087
22.2 Performance Under Unit Head—Unit Quantities 1087
22.3 Performance Under Specific Conditions 1090
22.4 Expressions for Specific Speeds in Terms of Known Coefficients for Different Turbines 1093
22.5 Performance Characteristic Curves 1096
22.6 Model Testing of Turbines 1101
22.7 Cavitation in Turbines 1105
22.8 Selection of Turbines 1107
Sumary of Main Points *1127*
Problems *1129*

CHAPTER 23. RECIPROCATING PUMPS 1131–1176

23.1 Introduction 1131
23.2 Main Components and Working of a Reciprocating Pump 1131
23.3 Types of Reciprocating Pumps 1133
23.4 Work Done by Reciprocating Pump 1135
23.5 Coefficient of Discharge, Slip, Percentage Slip and Negative Slip of Reciprocating Pump 1137
23.6 Effect of Acceleration of Piston on Velocity and Pressure in the Suction and Delivery Pipes 1137
23.7 Indicator Diagrams 1143
23.8 Air Vessels 1148

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

23.9 Multi-Cylinder Pumps 1157
23.10 Operating Characteristic Curves of Reciprocaing Pumps 1157
Sumary of Main Points *1174*
Problems *1175*

CHAPTER 24. CENTRIFUGAL PUMPS 1177–1245

24.1 Introduction 1177
24.2 Advantages of Centrifugal Pumps over Reciprocating Pumps 1178
24.3 Component Parts of a Centrifugal Pump 1178
24.4 Working of Centrifugal Pump 1179
24.5 Types of Centrifugal Pumps 1181
24.6 Work done by the Impeller 1184
24.7 Head of Pump 1185
24.8 Losses and Efficiencies 1190
24.9 Minimum Starting Speed 1194
24.10 Loss of Head Due to Reduced or Increased Flow 1195
24.11 Diameters of Impeller and Pipes 1196
24.12 Specific Speed 1197
24.13 Model Testing of Pumps 1199
24.14 Pump in Series—Multi-Stage Pumps 1201
24.15 Pumps in Parallel 1202
24.16 Performance of Pumps—Characteristic Curves 1203
24.17 Limitation of Suction Lift 1206
24.18 Net Positive Suction Head (npsh) 1207
24.19 Cavitation in Centrifugal Pumps 1208
24.20 Computation of the Total Head of Pumping—System Head Curves 1209
24.20.1 Operating Point or Operating Range of a Centrifugal Pump 1210
24.20.2 Selection of a Pumping Unit 1212
24.20.3 Pumps Operated in Series 1212
24.20.4 Pumps Operated in Parallel 1213
24.21 Priming Devices 1214
24.22 Centrifugal Pump-Troubles and Remedies 1214
Sumary of Main Points *1241*
Problems *1243*

CHAPTER 25. MISCELLANEOUS HYDRAULIC MACHINES 1246–1277

25.1 Introduction 1246
25.2 Hydraulic Accumulator—Simple and Differential Types 1246
25.3 Hydraulic Intensifier 1248
25.4 Hydraulic Press 1250
25.5 Hydraulic Crane 1251
25.6 Hydraulic Lift 1253
25.7 Hydraulic Ram 1254

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

25.8 Hydraulic Couplings and Torque Converters 1257
25.9 Air Lift Pump 1259
Sumary of Main Points *1275*
Problems *1276*

CHAPTER 26. ELEMENTS OF HYDROLOGY 1278–1297

26.1 Definition 1278
26.2 The Hydrologic Cycle 1278
26.3 Precipitation 1279
26.4 Measurement of Rainfall and Snowfall 1279
26.4.1 Measurement of Rainfall 1279
26.5 Mean Depth of Rainfall over an Area 1282
26.5.1 Arithmetic Mean Method 1282
26.5.2 Theissen Polygon Method 1283
26.5.3 Isohyetal Method 1284
26.6 Evaporation, Transpiration and Evapo-Transpiration 1284
26.7 Infiltration 1286
26.8 Runoff and Factors Affecting Runoff 1287
26.8.1 Factors Affecting Runoff 1288
26.9 Hydrograph 1289
26.10 Methods of Determination of Runoff 1290
Sumary of Main Points *1296*
Problems *1297*

CHAPTER 27. WATER POWER ENGINEERING 1298–1321

27.1 Introduction 1298
27.2 Hydroelectric Power Development of India and The World 1298
27.3 Comparison of Thermal and Hydroelectric Power Costs 1300
27.4 Assessment of Available Power 1300
27.5 Storage and Pondage 1301
27.6 Essential Stream Flow Data for Water Power Studies 1302
27.7 Flow Duration Curve 1302
27.8 Mass Curve 1305
27.9 Types of Hydropower Plants 1307
27.10 Typical Hydroelectric Developments of India 1309
27.10.1 Bhakra-Nangal Hydroelectric Project 1309
27.10.2 Chambal Valley Development Scheme 1311
27.11 Firm (or primary) and Secondary Power 1311
27.12 Load Factor, Utilisation Factor and Capacity Factor 1311
27.13 Components of Hydropower Plants 1312
Sumary of Main Points *1321*
Problems *1321*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

CHAPTER 28. FLUVIAL HYDRAULIC **1322–1339**

28.1 Introduction 1322
28.2 Sediment Transport in Channels 1322
28.3 Sediment Properties 1323
28.4 Modes of Sediment Movement 1324
28.5 Types of Sediment Load 1325
28.6 Initiation of Sediment Motion 1325
28.7 Bed Deformations in Alluvial Streams 1329
28.8 Resistance to Flow in Alluvial Streams 1330
28.9 Design of Unlined Alluvial Channels—Kennedy's and Lacey's Theories 1332
28.9.1 Kennedy's Theory 1332
28.9.2 Lacey's Regime Theory 1333
Sumary of Main Points *1339*

CHAPTER 29. FLOW MEASUREMENT AND LABORATORY EXPERIMENTS **1340–1346**

29.1 Introduction 1340
29.2 Fluid flow Measurements 1340
29.3 Flow Visualization Techniques 1344
29.4 List of Experiments 1344
29.5 Writing of Report 1346

Multiple Choice Questions *1347*
Appendix – I (Main Relations of Fluid Mechanics in Vector Notation) *1373*
Appendix – II (Comparative Study of Flow of Incompreessible and Compressible fluids) *1376*
Appendix – III (Some Important Conversion Factors) *1379*
Appendix – IV (Source, Sink and Doublet) *1382*
Appendix – V (Cavitation) *1385*
Appendix – VI (Flow in Curved Channels) *1387*
Appendix – VII (Control Valves for Pipes) *1389*
Appendix – VIII (Hydraulic Transport of Solid Material in Pipes) *1392*
Bibliography *1394*
Author Index *1396*
Index *1397*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Properties of Fluids

Chapter **1**

1.1 INTRODUCTION

A matter exists in either the solid state or the fluid state. The fluid state is further divided into the liquid and the gaseous states. In fact the same matter may exist in any one of the three states viz., solid, liquid and gaseous. For example water, which ordinarily occurs in a liquid state, may also occur under natural conditions in a solid state as ice and in a gaseous state as vapour. The solids, liquids and gases exhibit different characteristics on account of their different molecular structure. All substances consist of vast numbers of molecules separated by empty space. The molecules are continuously moving within the substance and they have an attraction for each other, but when the distance between them becomes very small (of the order of the diameter of the molecule) there is a force of repulsion between the molecules which pushes them apart. In solids the molecules are very closely spaced, but in liquids the spacing between the molecules is relatively large and in gases the space between the molecules is still larger. As such in a given volume a solid contains a large number of molecules, a liquid contains relatively less number of molecules and a gas contains much less number of molecules. It thus follows that in solids the force of attraction between the molecules is large on account of which there is very little movement of molecules within the solid mass and hence solids possess compact and rigid form. In liquids the force of attraction between the molecules is relatively less due to which the molecules can move freely within the liquid mass, but the force of attraction between the molecules is sufficient to keep the liquid together in a definite volume. In gases the force of attraction between the molecules is much less due to which the molecules of gases have greater freedom of movement so that the gases fill completely the container in which they are placed. It may, however, be stated that inspite of the larger mobility and spacing of the molecules of fluids, for mechanical analysis a fluid is considered to be *continuum* i.e., a continuous distribution of matter with no voids or empty spaces. This assumption is justifiable because ordinarily the fluids involved in most of the engineering problems have large number of molecules and the distances between them are small.

Another difference that exists between the solids and the fluids is in their relative abilities to resist the external forces. A solid can resist tensile, compressive and shear forces upto a certain limit. A fluid has no tensile strength or very little of it, and it can resist the compressive forces only when it is kept in a container. When subjected to a shearing force, a fluid deforms continuously as long as this force is applied. The inability of the fluids to resist shearing stress gives them their characteristic property to change shape or to flow. This, however, does not mean that the fluids do not offer any

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

resistance to shearing forces. In fact as the fluids flow there exists shearing (or tangential) stresses between the adjacent fluid layers which result in opposing the movement of one layer over the other. The amount of shear stress in a fluid depends on the magnitude of the rate of deformation of the fluid element. However, if a fluid is at rest no shear force can exist in it.

The two classes of fluids, viz., gases and liquids also exhibit quite different characteristics. Gases can be compressed much readily under the action of external pressure and when the external pressure is removed the gases tend to expand indefinitely. On the other hand under ordinary conditions liquids are quite difficult to compress and therefore they may for most purposes be regarded as incompressible. Moreover, even if the pressure acting on a liquid mass is removed, still the cohesion between particles holds them together so that the liquid does not expand indefinitely and it may have a free surface, that is a surface from which all pressure except atmospheric pressure is removed.

1.2 DEFINITION OF A FLUID

In view of the above discussion a *fluid* may be defined as a substance which is capable of flowing. It has no definite shape of its own, but conforms to the shape of the containing vessel. Further even a small amount of shear force exerted on a fluid will cause it to undergo a deformation which continues as long as the force continues to be applied.

A *liquid* is a fluid, which possesses a definite volume, which varies only slightly with temperature and pressure. Since under ordinary conditions liquids are difficult to compress, they may be for all practical purposes regarded as incompressible. It forms a free surface or an interface separating it from the atmosphere or any other gas present.

A *gas* is a fluid, which is compressible and possesses no definite volume but it always expands until its volume is equal to that of the container. Even a slight change in the temperature of a gas has a significant effect on its volume and pressure. However, if the conditions are such that a gas undergoes a negligible change in its volume, it may be regarded as incompressible. But if the change in volume is not negligible the compressibility of the gas will have to be taken into account in the analysis.

A *vapour* is a gas whose temperature and pressure are such that it is very near the liquid state. Thus steam may be considered as a vapour because its state is normally not far from that of water.

The fluids are also classified as *ideal fluids* and *practical* or *real fluids*.

Ideal fluids are those fluids which have no viscosity and surface tension and they are incompressible. As such for ideal fluids no resistance is encountered as the fluid moves. However, in nature the ideal fluids do not exist and therefore, these are only imaginary fluids. The existence of these imaginary fluids was conceived by the mathematicians in order to simplify the mathematical analysis of the fluids in motion. The fluids which have low viscosity such as air, water etc., may however be treated as ideal fluids without much error.

Practical or real fluids are those fluids which are actually available in nature. These fluids possess the properties such as viscosity, surface tension and compressibility and therefore a certain amount of resistance is always offered by these fluids when they are set in motion.

1.3 DEVELOPMENT OF FLUID MECHANICS

Fluid mechanics is that branch of science which deals with the behaviour of the fluids at rest as well as in motion. In general the scope of fluid mechanics is very wide which includes the study of all liquids and gases. But usually it is confined to the study of liquids and those gases for which the effects due to compressibility may be neglected. The gases with appreciable compressibility effects are governed by the laws of Thermodynamics which are however dealt with under the subject of Gas dynamics.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The problems, man encountered in the fields of water supply, irrigation, navigation and water power, resulted in the development of the fluid mechanics. However, with the exception of Archimedes (250 B.C.) Principle which is considered to be as true today as some 2250 years ago, little of the scant knowledge of the ancients appears in modern fluid mechanics. After the fall of Roman Empire (476 A.D.) there is no record of progress made in fluid mechanics until the time of Leonardo da Vinci (1500 A.D.), who designed the first chambered canal lock. However, upto da Vinci's time, concepts of fluid motion must be considered to be more art than science.

Some two hundred years ago mankind's centuries of experience with the flow of water began to crystallize in scientific form. Two distinct Schools of thought gradually evolved in the treatment of fluid mechanics. One, commonly known as Classical Hydrodynamics, deals with theoretical aspects of the fluid flow, which assumes that shearing stresses are non-existent in the fluids, that is, ideal fluid concept. The other known as Hydraulics, deals with the practical aspects of fluid flow which has been developed from experimental findings and is, therefore, more of empirical nature. Notable contributions to theoretical hydrodynamics have been made by Euler, D'Alembert, Navier, Lagrange, Stokes, Kirchoff, Rayleigh, Rankine, Kelvin, Lamb and many others. Many investigators have contributed to the development of experimental hydraulics, notable amongst them being Chezy, Venturi, Bazin, Hagen, Poiseuille, Darcy, Weisbach, Kutter, Manning, Francis and several others.

Although the empirical formulae developed in hydraulics have found useful application in several problems, it is not possible to extend them to the flow of fluids other than water and in the advanced field of aerodynamics. As such there was a definite need for a new approach to the problems of fluid flow—an approach which relied on classical hydrodynamics for its analytical development and at the same time on experimental means for checking the validity of the theoretical analysis. The modern Fluid Mechanics provides this new approach, taking a balanced view of both the theorists and the experimentalists. The generally recognized founder of the modern fluid mechanics is the German Professor, Ludwig Prandtl. His most notable contribution being the boundary layer theory which has had a tremendous influence upon the understanding of the problems involving fluid motion. Other notable contributors to the modern fluid mechanics are Blasius, Bakhmeteff, Nikuradse, Von-Karman, Reynolds, Rouse and many others.

In this book the fundamental principles of fluid mechanics applicable to the problems involving the motion of a particular class of fluids called Newtonian fluids (such as water, air, kerosene, glycerine etc.) have been discussed along with the relevant portions of the experimental hydraulics.

1.4 UNITS OF MEASUREMENT

Units may be defined as those standards in terms of which the various physical quantities like length, mass, time, force, area, volume, velocity, acceleration etc., are measured. The system of units used in mechanics are based upon Newton's second law of motion, which states that force equals mass times acceleration or $F = m \times a$, where F is the force, m is the mass and a is the acceleration. There are in general four systems of units, two in metric (C.G.S. or M.K.S.) system and two in the English (F.P.S.) system. Of the two, one is known as the absolute or physicist's system and the other as the gravitational or engineer's system. The difference between the absolute and gravitational systems is that in the former the standard is the unit of mass. The unit of force is then derived by Newton's law. In the gravitational system the standard is the unit of force and the unit of mass is derived by Newton's law. Table 1.1 lists the various units of measurement for some of the basic or fundamental quantities.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

TABLE 1.1

Quantity	*Metric Units*		*English Units*	
	Gravitational	*Absolute*	*Gravitational*	*Absolute*
Length	Metre (m)	Metre (m)	Foot (ft)	Foot (ft)
Time	Second (sec)	Second (sec)	Second (sec)	Second (sec)
Mass	Metric slug (msl)	Gram [gm (mass)]	Slug (sl)	Pound [lb (mass)]
Force	kilogram [kg(f)]	Dyne	Pound [lb (f)]	Poundal (pdl)
Temperature	°C	°C	°F	°F

The units of measurement for the various other quantities may be readily obtained with the help of the Table 1.1. Further Table 1.2 below illustrates all the four systems of units in which the units are defined so that one unit of force equals one unit of mass times one unit of acceleration.

TABLE 1.2

Systems of Units	*Relationships*
Metric Absolute	1 dyne = 1 gram × $\frac{1\text{ cm}}{\text{sec}^2}$
Metric Gravitational	1 kilogram (f) = 1 metric slug × $\frac{1\text{ m}}{\text{sec}^2}$
English Absolute	1 poundal = 1 pound × $\frac{1\text{ ft}}{\text{sec}^2}$
English Gravitational	1 pound (f) = 1 slug × $\frac{1\text{ ft}}{\text{sec}^2}$

Further the following relationships may be utilized to affect the conversion from one system to another.

1 gram-wt. = 981 dynes : 1 metric slug = 9810 gm (mass)

1 lb-wt.= 32.2 poundals : 1 slug = 32.2 lb (mass)

The use of the different systems of units by the scientists and the engineers and also by the different countries of the world often leads to a lot of confusion. Therefore, it was decided at the Eleventh General Conference of Weights and Measures held in Paris in 1960 to adopt a unified, systematically constituted, coherent system of units for international use. This system of units is called the *International System of Units* and is designated by the abbreviation *SI Units.* More and more countries of the world are now adopting this system of units. There are six base units in SI system of units which are given in Table 1.3.

TABLE 1.3

Quantity	*Unit*	*Symbol*
Length	metre	m
Mass	kilogram	kg
Time	second	s
Electric Current	ampere	A
Thermodynamic temperature	kelvin	K
Luminous intensity	candela	cd

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In addition to the above noted base units there are two supplementary units which are given in Table 1.4.

TABLE 1.4

Quantity	*Unit*	*Symbol*
Plane angle	radian	rad
Solid angle	steradian	sr

The unit of a derived quantity is obtained by taking the physical law connecting it with the basic (or primary or fundamental) quantities and then introducing the corresponding units for the basic quantities. Thus in SI units the unit of force is newton (N) which according to Newton's second law of motion is expressed as 1 N = 1 kg × 1 m/s^2, i.e., a force of 1 N is required to accelerate a mass of 1 kg by 1 m/s^2. The units for some of the derived quantities have been assigned special names and symbols. Some of the important derived units with special names, commonly used in Fluid mechanics, in SI and metric gravitational systems of units are given in Table 1.5.

TABLE 1.5

Derived Quantity	*System of Units*			
	SI		*Metric Gravitational*	
	Unit	*Symbol*	*Unit*	*Symbol*
(1)	(2)	(3)	(4)	(5)
Area	square metre	m^2	square metre	m^2
Volume	cubic metre	m^3	cubic metre	m^3
Velocity	metre per second	m/s	metre per second	m/sec
Angular velocity	radian per second	rad/s	radian per second	rad/sec
Acceleration	metre per second square	m/s^2	metre per second square	m/sec^2
Angular acceleration	radian per second square	rad/s^2	radian per second square	rad/sec^2
Frequency	hertz	Hz		1/sec
Discharge	cubic metre per second	m^3/s	cubic metre per second	m^3/sec
Mass density (Specific mass)	kilogram per cubic metre	kg/m^3	metric slug per cubic metre	msl/m^3
Force	newton	N (= kg.m/s^2)	Kilogram (f)	kg(f)-
Pressure, Stress, Elastic Modulus	newton per square metre (= pascal)	N/m^2 (= Pa)	kilogram (f) per square metre	kg(f)/m^2
Weight density (Specific Weight)	newton per cubic metre	N/m^3	kilogram (f) per cubic metre	kg(f)/m^3
Dynamic viscosity	newton second per square metre (= pascal second)	N.s/m^2 (= pa. s)	kilogram (f)-second per square metre	kg(f)-sec/m^2
Kinematic viscosity	square metre per second	m^2/s	square metre per second	m^2/sec
Work, Energy, Torque	joule	J (= N.m)	kilogram(f)-metre	kg(f)-m
Quantity of heat	joule	J	kilo-calorie	kcal
Power	watt	W (= J/s)	kilogram(f) - metre per second	kg(f)-m/sec
Surface tension	newton per metre	N/m	kilogram (f) per metre	kg(f)/m

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 1.5 Contd.

(1)	(2)	(3)	(4)	(5)
Momentum	kilogram metre per second	kg. m/s	metric slug - metre per second	msl-m/sec
Moment of momentum	kilogram square metre per second	kg.m/s	metric slug-square metre per second	msl-m^2/sec
Entropy	(i) joule per kilogram kelvin	J/(kg. K)	(i) kilo-calorie per metric slug deg. Centigrade abs.	k cal/msl °C (abs)
	or		or	
	(ii) joule per kelvin	J/K	(ii) kilo-calorie per deg. Centigrade abs.	k cal/°C (abs)
Specific heat	joule per kilogram kelvin	J/(kg. K)	kilo-calorie per metric slug deg. Centigrade abs.	k cal/msl °C (abs)
Gas constant	joule per kilogram kelvin	J/(kg.K)	kilogram (f) metre per metric slug deg. Centigrade abs.	kg (f)-m/msl °C (abs)
Thermal conductivity	watt per metre kelvin	W/m.K	kilocalorie per second metre deg. Centigrade abs.	kcal/sec-m °C (abs)

Certain units though outside the International System have been retained for general use in this system also. These units are given in Table 1.6.

TABLE 1.6

Quantity	*Unit*	*Symbol*	*Value in SI Units*
Area of land	area	a	100 m^2
– do–	hectare	ha	10 000 m^2
Time	minute	min	60 s
–do–	hour	h	60 min = 3600 s
–do–	day	d	24 h = 86 400 s
Mass	tonne	t	1000 kg
Volume	litre	l	10^{-3} m^3 = 1 dm^3
Dynamic viscosity	poise	P	10^{-1} N.s/m^2
–do–	centipoise	cP	10^{-3} N.s/m^2
Kinematic viscosity	stoke	S	10^{-4} m^2/s
Pressure of fluid	bar	bar	100 kN/m^2 = 10^5 Pa

In using SI units certain rules and conventions are to be followed which are as noted below:

(i) Names of units, even when they are named after persons, are not written with first letter capital when written in the spelled form e.g., newton, joule, watt etc.

(ii) The symbols for the units which are named after persons are written with capital first letter of the name e.g., N for newton, J for joule, W for watt, etc.

(iii) The symbols for all other units are written in lower case (small letters), e.g., m for metre, s for second, kg for kilogram, etc.

(iv) A dot is inserted in the space between the symbols for the compound (or combined) units e.g., N.m (for newton metre), kW-h (for kilowatt hour) etc.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(v) Numbers with more than three digits should be written in groups of three with narrow space in between consecutive groups. However, a sequence of four digits is usually not broken e.g.,

50000	should be	written as	50 000
72345.685	should be	written as	72 345.685
0.13579	should be	written as	0.135 79
9810	may be	written as	9 810 or 9810

(vi) The decimal multiples and sub-multiples of the units are formed by using the prefixes. The various prefixes and the corresponding symbols are given in Table 1.7.

TABLE 1.7

Unit multiplier	*Prefix*	*Symbol*	*Unit multiplier*	*Prefix*	*Symbol*
10^{18}	exa	E	10^{-1}	deci	d
10^{15}	peta	P	10^{-2}	centi	c
10^{12}	tera	T	10^{-3}	milli	m
10^{9}	giga	G	10^{-6}	micro	m
10^{6}	mega	M	10^{-9}	nano	n
10^{3}	kilo	k	10^{-12}	pico	p
10^{2}	hecto	h	10^{-15}	femto	f
10	deca	da	10^{-18}	atto	a

The prefixes hecto (h), deca (da), deci (d) and centi (c) are not commonly used and these should be used only when special necessity arises. Further as far as possible prefixes in steps of $10^{\pm 3}$ should only be used to form multiples and sub-multiples of the units. It may also be noted that compound prefixes (or combination of prefixes) are not to be used. For example the correct form for 10^{-9} is the prefix nano (n) and not the combination of prefixes such as milli micro (mμ) or any other combination which is incorrect and should not be used.

In this book both metric gravitational system of units as well as SI units have been used. Some important conversion factors in various systems of units are given in Appendix V.

1.5 MASS DENSITY, SPECIFIC WEIGHT, SPECIFIC VOLUME

Mass density (or specific mass) of a fluid is the mass which it possesses per unit volume. It is denoted by a symbol ρ (Greek 'rho'). In SI units mass density is expressed in kilogram per cubic metre i.e., kg/m^3. In the metric gravitational system of units mass density is expressed in metric slug per cubic metre i.e., ms1/m^3 and in the metric absolute system of units it is expressed in gm (mass) per cubic centimetre i.e., gm/cm^3 or gm/cc. The corresponding units in the English gravitational and absolute systems of units are slugs per cubic foot i.e., slugs/ft^3 and pound (mass) per cubic foot i.e., lb (m)/ft^3 respectively. The mass density of water at °C in different systems of units is 1000 kg/m^3, or 102 msl/m^3, or 1 gm/cc, or 1.94 slugs/ft^3, or 62.4 lb(m)/ft^3.

Since a molecule of a substance has a certain mass regardless of its state (solid, liquid or gas), it then follows that the mass density is proportional to the number of molecules in a unit volume of the fluid. As the molecular activity and spacing increase with temperature, fewer molecules exist in a given volume of fluid as temperature rises. Therefore, the mass density of a fluid decreases with increasing temperature. Further by application of pressure a large number of molecules can be forced into a given volume, it is to be expected that the mass density of a fluid will increase with increasing pressure.

Specific weight (also called weight density) of a fluid is the weight it possesses per unit volume. It is denoted by a symbol *w* or γ (Greek 'gama'). As it represents the force exerted by gravity on a unit

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

volume of fluid, it has units of force per unit volume. In SI units specific weight is expressed in newton per cubic metre i.e., N/m^3. In the metric gravitational system of units specific weight is expressed in kilogram (f) per cubic metre i.e., $kg(f)/m^3$ and in the metric absolute system of units it is expressed in dynes per cubic centimetre i.e., dynes/cm^3 or dynes/cc. The correponding units in the English gravitational and absolute systems of units are pound (f) per cubic foot i.e., lb (f)/ft^3 and poundal per cubic foot, i.e., pdl/ft^3 respectively. The specific weight of water at 4°C in different units is 9810 N/m^3 (or 9.81 kN/m^3), or 1000 $kg(f)/m^3$, or 981 dynes/cm^3, or 62.4 lb(f)/ft^3, or (62.4 × 32.2) pdl/ft^3.

The mass density ρ and specific weight w are related as indicated below

$$w = \rho g; \rho = \frac{w}{g} \qquad ...(1.1)$$

where g is acceleration due to gravity.

The specific weight depends on the gravitational acceleration and the mass density. Since the gravitational acceleration varies from place to place, the specific weight will also vary. Further as stated earlier the mass density changes with temperature and pressure, hence the specific weight will also depend upon temperature and pressure.

Specific volume of a fluid is the volume of the fluid per unit weight. Thus it is the reciprocal of specific weight. It is generally denoted by v. In SI units specific volume is expressed in cubic metre per newton i.e., m^3/N. In the metric gravitational system of units specific volume is expressed in cubic metre per kilogram (f) i.e., $m^3/kg(f)$ and in the metric absolute system of units it is expressed in cubic centimetre per dyne i.e., cm^3/dyne or cc/dyne. The corresponding units in the English gravitational and absolute systems of units are cubic foot per pound (f) i.e., $ft^3/lb(f)$ and cubic foot per poundal i.e., ft^3/ pdl respectively.

For the problems involving the gas flow *specific volume* is generally defined as the volume of the fluid per unit mass, in which case it is reciprocal of mass density. In SI units the specific volume is then expressed in cubic metre per kilogram i.e., m^3/kg. In the metric gravitational system of units it is expressed in cubic metre per metric slug i.e., m^3/msl and in the metric absolute system of units it is expressed in cubic centimetre per gram(mass) i.e., $cm^3/gm(m)$ or cc/gm(m). The corresponding units in the English gravitational and absolute units are cubic foot per slug i.e., ft^3/slug and cubic foot per pound (mass) i.e., $ft^3/lb(m)$ respectively.

For liquids the mass density, the specific weight and specific volume vary only slightly with the variation of temperature and pressure. It is due to the molecular structure of the liquids in which the molecules are arranged very compactly (in contrast to that of a gas). The presence of dissolved air, salts in solution and suspended matter will slightly increase the values of the mass density and the specific weight of the liquids.

For gases the values of the above properties vary greatly with variation of either temperature, or pressure, or both. It is due to the molecular structure of the gas in which the molecular spacing (i.e., volume) changes considerably on account of pressure and temperature variations.

1.6 SPECIFIC GRAVITY

Specific gravity (sp. gr.) is the ratio of specific weight (or mass density) of a fluid to the specific weight (or mass density) of a standard fluid. For liquids, the standard fluid chosen for comparison is pure water at 4°C (39.2°F). For gases, the standard fluid chosen is either hydrogen or air at some

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

specified temperature and pressure. As specific weight and mass density of a fluid vary with temperature, temperatures must be quoted when specific gravity is used in precise calculations of specific weight or mass density. Being a ratio of two quantities with same units, specific gravity is a pure number independent of the system of units used. The specific gravity of water at the standard temperature (i.e., 4°C), is therefore, equal to 1.0. The specific gravity of mercury varies from 13.5 to 13.6. Knowing the specific gravity of any liquid, its specific weight may be readily calculated by the following relation,

$$w = \text{Sp. gr. of liquid} \times \text{Specific weight of water}$$
$$= (\text{Sp. gr. of liquid}) \times 9\,810\ \text{N/m}^3.$$

1.7 EQUATION OF STATE: THE PERFECT GAS

The density ρ of a particular gas is related to its absolute pressure p and absolute temperature T by the *equation of state,* which for a perfect gas takes the form

$$p = \rho RT; \text{ or } pV = mRT \qquad ...(1.2)$$

in which R is a constant called the *gas constant,* the value of which is constant for the gas concerned, and V is the volume occupied by the mass m of the gas. The absolute pressure is the pressure measured above absolute zero (or complete vacuum) and is given by

$$p_{abs} = p_{gage} + p_{atm} \quad \text{(see also Sec. 2.5)}$$

The absolute temperature is expressed in 'kelvin' i.e., K, when the temperature is measured in °C and it is given by

$$T°_{(abs)} = T\ K = 273.15 + t°C$$

No actual gas is perfect. However, most gases (if at temperatures and pressures well away both from the liquid phase and from dissociation) obey this relation closely and hence their pressure, density and (absolute) temperature may, to a good approximation, be related by Eq.1.2. Similarly air at normal temperature and pressure behaves closely in accordance with the equation of state.

It may be noted that the gas constant R is defined by Eq. 1.2 as $p/\rho T$ and, therefore, its dimensional expression is $(FL/M\theta)$. Thus in SI units the gas constant R is expressed in newton-metre per kilogram per kelvin i.e., (N.m/kg. K). Further, since 1 joule = 1 newton × 1 metre, the unit for R also becomes joule per kilogram per kelvin i.e., (J/kg. K). Again, since 1 N = 1 kg × 1 m/s^2, the unit for R becomes (m^2/s^2 K).

In metric gravitational and absolute systems of units, the gas constant R is expressed in kilogram (f)-metre per metric slug per degree C absolute i.e., [kg(f)-m/msl deg. C abs.], and dyne-centimetre per gram (m) per degree C absolute i.e., [dyne-cm/gm(m) deg. C abs.] respectively.

For air the value of R is 287 N-m/kg K, or 287 J/kg K, or 287 m^2/s^2 K.

In metric gravitational system of units the value of R for air is 287 kg(f)-m/msl deg. C abs. Further, since 1 msl = 9.81 kg (m), the value of R for air becomes (287/9.81) or 29.27 kg(f)-m/kg(m) deg. C abs.

Since specific volume may be defined as reciprocal of mass density, the equation of state may also be expressed in terms of specific vloume of the gas as

$$pv = RT \qquad ...(1.2\ a)$$

in which v is specific volume.

The equation of state may also be expressed as

$$p = wRT \qquad ...(1.2\ b)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in which w is the specific weight of the gas. The unit for the gas constant R then becomes (m/K) or (m/deg. C abs). It may, however, be shown that for air the value of R is 29.27 m/K.

For a given temperature and pressure, Eq. 1.2 indicates that ρR = constant. By Avogadro's hypothesis, all pure gases at the same temperature and pressure have the same number of molecules per unit volume. The density is proportional to the mass of an individual molecule and so the product of R and the 'molecular weight' M is constant for all perfect gases. This product MR is known as the *universal gas constant*. For real gases it is not strictly constant but for monatomic and diatomic gases its variation is slight. If M is the ratio of the mass of the molecule to the mass of a hydrogen atom, MR = 8310 J/kg K.

1.8 VISCOSITY

Viscosity is that property of a fluid by virtue of which it offers resistance to the movement of one layer of fluid over an adjacent layer. It is primarily due to cohesion and molecular momentum exchange between fluid layers, and as flow occurs, these effects appear as shearing stresses between the moving layers of fluid.

Consider two plates (Fig. 1.1) sufficiently large (so that edge conditions may be neglected) placed a small distance Y apart, the space between them being filled with fluid. The lower plate is assumed to be at rest, while the upper one is moved parallel to it with a velocity V by the application of a force F, corresponding to area A, of the moving plate in contact with the fluid. Particles of the fluid in contact with each plate will adhere to it and if the distance Y and velocity V are not too great, the velocity v at a distance y from the lower plate will vary uniformly from zero at the lower plate which is at rest, to V at the upper moving plate. Experiments show that for a large variety of fluids

$$F \sim \frac{AV}{Y}$$

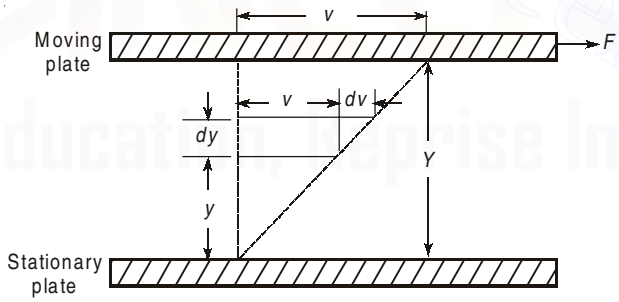


Figure 1.1 *Fluid motion between two parallel plates*

It may be seen from similar triangles in Fig. 1.1 that the ratio V/Y can be replaced by the velocity gradient (dv/dy), which is the rate of angular deformation of the fluid. If a constant of proportionality μ (Greek 'mu') be introduced, the shear stress τ (Greek 'tau') equal to (F/A) between any two thin sheets of fluid may be expressed as

$$\tau = \frac{F}{A} = \mu \frac{V}{Y} = \mu \frac{dv}{dy} \qquad ...(1.3)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 1.3 is called Newton's equation of viscosity, and in the transposed form it serves to define the proportionality constant

$$\mu = \frac{\tau}{dv/dy} \qquad ...(1.3\ a)$$

which is called the *coefficient of viscosity*, or the *dynamic viscosity* (since it involves force), or simply *viscosity* of the fluid. Thus the dynamic viscosity μ, may be defined as the shear stress required to produce unit rate of angular deformation.

In SI units μ is expressed in N.s/m^2, or kg/m.s.

In the metric gravitational system of units, μ is expressed in kg(f)-sec/m^2 or msl/m-sec. In the metric absolute system of units μ is expressed in dyne-sec/m^2 or gm(mass)/cm-sec which is also called 'poise' after Poiseuille. The 'centipoise' is one hundredth of a poise.

In the English gravitational system of units μ is expressed in lb(f)-sec/ft^2 or slug/ft-sec and in the English absolute system of units it is expressed in pdl-sec/ft^2 or lb(m)/ft-sec.

The numerical conversion from one system to another is as follows:

1 N.s/m^2 = 0.102 kg(f)-sec/m^2 = 10 poise

$$1\frac{\text{lb(f)sec}}{\text{ft}^2} = 479\ \frac{\text{dyne-sec}}{\text{cm}^2} = 479 \text{ poise} = 4.88\ \frac{\text{kg(f)-sec}}{\text{m}^2}$$

In many problems involving viscosity there frequently appears a term dynamic viscosity μ divided by mass density ρ. The ratio of the dynamic viscosity μ and the mass density ρ is known as *Kinematic viscosity* and is denoted by the symbol υ (Greek 'nu') so that

$$\upsilon = \frac{\mu}{\rho} \qquad ...(1.4)$$

On analysing the dimensions of the kinematic viscosity it will be observed that it involves only the magnitudes of length and time. The name kinematic viscosity has been given to the ratio (μ/ρ) because kinematics is defined as the study of motion without regard to the cause of the motion and hence it is concerned with length and time only.

In SI units υ is expressed in m^2/s. In the metric system of units υ is expressed in cm^2/sec or m^2/sec. The unit cm^2/sec is termed as 'stoke' after G.G. Stokes and its one-hundredth part is called 'centistoke'. In the English system of units it is expressed in ft^2/sec.

The numerical conversion from one system to another is as follows:

$$1\ \frac{\text{m}^2}{\text{s}} = 10^4\ \frac{\text{cm}^2}{\text{sec}} = 10^4 \text{ stokes} = 10.764\ \frac{\text{ft}^2}{\text{sec}}$$

The dynamic viscosity μ of either a liquid or a gas is practically independent of the pressure for the range that is ordinarily encountered in practice. However, it varies widely with temperature. For gases, viscosity increases with increase in temperature while for liquids it decreases with increase in temperature. This is so because of their fundamentally different intermolecular characteristics. In liquids the viscosity is governed by the cohesive forces between the molecules of the liquid, whereas in gases the molecular activity plays a dominant role. The kinematic viscosity of liquids and of gases at a given pressure, is essentially a function of temperature.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Common fluids such as air, water, glycerine, kerosene etc., follow Eq. 1.3. There are certain fluids which, however, do not follow Newton's law of viscosity. Accordingly fluids may be classified as Newtonian fluids and non-Newtonian fluids. In a Newtonian fluid there is a linear relation between the magnitude of shear stress and the resulting rate of deformation i.e., the constant of proportionality μ in Eq. 1.3 does not change with rate of deformation. In a non-Newtonian fluid there is a non-linear relation between the magnitude of the applied shear stress and the rate of angular deformation. In the case of a plastic substance which is a non-Newtonian fluid an initial yield stress is to be exceeded to cause a continuous deformation. An ideal plastic has a definite yield stress and a constant linear relation between shear stress and the rate of angular deformation. A thixotropic substance, which is a non-Newtonian fluid, has a non-linear relationship between the shear stress and the rate of angular deformation, beyond an initial yield stress. The printer's ink is an example of a thixotropic liquid.

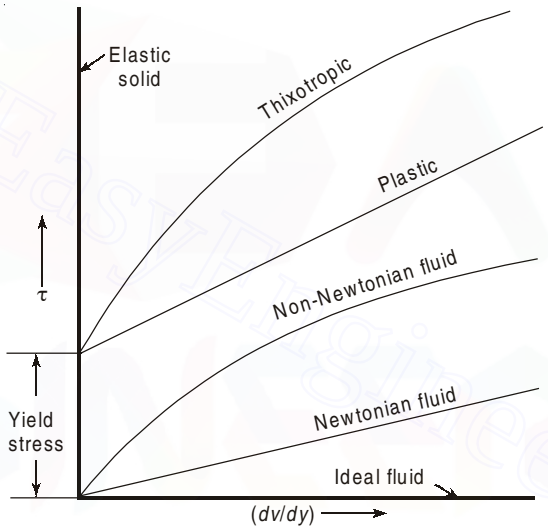


Figure 1.2 *Variation of shear stress with velocity gradient*

An ideal fluid is defined as that having zero viscosity or in other words shear stress is always zero regardless of the motion of the fluid. Thus an ideal fluid is represented by the horizontal axis ($\tau = 0$) in Fig. 1.2 which gives a diagrammatic representation of the Newtonian, non-Newtonian, plastic, thixotropic and ideal fluids. A true elastic solid may be represented by the vertical axis of the diagram.

The fluids with which engineers most often have to deal are Newtonian, that is, their viscosity is not dependent on the rate of angular deformation, and the term 'fluid-mechanics' generally refers only to Newtonian fluids. The study of non-Newtonian fluids is however termed as 'rheology'.

1.9 VAPOUR PRESSURE

All liquids possess a tendency to evaporate or vaporize i.e., to change from the liquid to the gaseous state. Such vaporization occurs because of continuous escaping of the molecules through the free liquid surface. When the liquid is confined in a closed vessel, the ejected vapour molecules get accumulated in the space between the free liquid surface and the top of the vessel. This accumulated vapour of the liquid exerts a partial pressure on the liquid surface which is known as *vapour pressure*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the liquid. As molecular activity increases with temperature, vapour pressure of the liquid also increases with temperature. If the external absolute pressure imposed on the liquid is reduced by some means to such an extent that it becomes equal to or less than the vapour pressure of the liquid, the boiling of the liquid starts, whatever be the temperature. Thus a liquid may boil even at ordinary temperature if the pressure above the liquid surface is reduced so as to be equal to or less than the vapour pressure of the liquid at that temperature.

If in any flow system the pressure at any point in the liquid approaches the vapour pressure, vaporization of liquid starts, resulting in the pockets of dissolved gases and vapours. The bubbles of vapour thus formed are carried by the flowing liquid into a region of high pressure where they collapse, giving rise to high impact pressure. The pressure developed by the collapsing bubbles is so high that the material from the adjoining boundaries gets eroded and cavities are formed on them. This phenomenon is known as cavitation.

Mercury has a very low vapour pressure and hence it is an excellent fluid to be used in a barometer. On the contrary various volatile liquids like benzene etc., have high vapour pressure.

1.10 COMPRESSIBILITY AND ELASTICITY

All fluids may be compressed by the application of external force, and when the external force is removed the compressed volumes of fluids expand to their original volumes. Thus fluids also possess elastic characteristics like elastic solids. Compressibility of a fluid is quantitatively expressed as inverse of the bulk modulus of elasticity K of the fluid, which is defined as:

$$K = \frac{\text{Stress}}{\text{Strain}} = -\frac{dp}{\left(\frac{dV}{V}\right)} = \frac{\text{Change in pressure}}{\left(\frac{\text{Change in volume}}{\text{Original volume}}\right)} \qquad ...(1.5)$$

Thus bulk modulus of elasticity K is a measure of the incremental change in pressure dp which takes place when a volume V of the fluid is changed by an incremental amount dV. Since a rise in pressure always causes a decrease in volume, dV is always negative, and the minus sign is included in the equation to give a positive value of K.

For example, consider a cylinder containing a fluid of volume V, which is being compressed by a piston. Now if the piston is moved so that the volume V decreases by a small amount dV, then the pressure will increase by amount dp, the magnitude of which depends upon the bulk modulus of elasticity of the fluid, as expressed in Eq. 1.5.

In SI units the bulk modulus of elasticity is expressed in N/m^2. In the metric gravitational system of units it is expressed in either $kg(f)/cm^2$ or $kg(f)/m^2$. In the English system of units it is expressed either in $lb(f)/in^2$ or $lb(f)/ft^2$. The bulk modulus of elasticity for water and air at normal temperature and pressure is approximately 2.06×10^9 N/m^2 [or 2.1×10^8 kg (f)/m^2] and 1.03×10^5 N/m^2 [or 1.05×10^4 kg (f)/m^2] respectively. Thus air is about 20,000 times more compressible than water. The bulk (volume) modulus of elasticity of mild steel is about 2.06×10^{11} N/m^2 [or 2.1×10^{10} kg(f)/m^2] which shows that water is about 100 times more compressible than steel.

However, the bulk modulus of elasticity of a fluid is not constant, but it increases with increase in pressure. This is so because when a fluid mass is compressed, its molecules become close together and its resistance to further compression increases i.e., K increases. Thus for example, the bulk modulus of water roughly doubles as the pressure is raised from 1 atmosphere to 3500 atmospheres.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The temperature of the fluid also affects the bulk modulus of elasticity of the fluid. In the case of liquids there is a decrease of K with increase of temperature. However, for gases since pressure and temperature are inter-related and as temperature increases, pressure also increases, an increase in temperature results in an increase in the value of K.

For liquids since the bulk modulus of elasticity is very high, the change of density with increase of pressure is very small even with the largest pressure change encountered. Accordingly in the case of liquids the effects of compressibility can be neglected in most of the problems involving the flow of liquids. However, in some special problems such as rapid closure of valve or water hammer, where the changes of pressure are either very large or very sudden, it is necessary to consider the effect of compressibility of liquids.

On the other hand gases are easily compressible and with the change in pressure the mass density of gases changes considerably and hence the effects of compressibility cannot ordinarily be neglected in the problems involving the flow of gases. However, in a few cases where there is not much change in pressure and so gases undergo only very small changes of density, the effects of compressibility may be disregarded e.g., the flow of air in a ventilating system is a case where air may be treated as incompressible.

1.11 SURFACE TENSION AND CAPILLARITY

Due to molecular attraction, liquids possess certain properties such as cohesion and adhesion. *Cohesion* means inter-molecular attraction between molecules of the same liquid. That means it is a tendency of the liquid to remain as one assemblage of particles. *Adhesion* means attraction between the molecules of a liquid and the molecules of a solid boundary surface in contact with the liquid. The property of cohesion enables a liquid to resist tensile stress, while adhesion enables it to stick to another body. *Surface tension* is due to cohesion between liquid particles at the surface, whereas *capillarity i*s due to both cohesion and adhesion.

(a) Surface Tension. A liquid molecule on the interior of the liquid body has other molecules on all sides of it, so that the forces of attraction are in equilibrium and the molecule is equally attracted on all the sides, as a molecule at point A shown in Fig. 1.3. On the other hand a liquid molecule at the surface of the liquid, (i.e., at the interface between a liquid and a gas) as at point B, does not have any

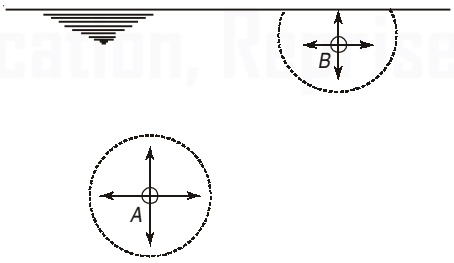


Figure 1.3 *Inter-molecular forces near a liquid surface*

liquid molecule above it, and consequently there is a net downward force on the molecule due to the attraction of the molecules below it. This force on the molecules at the liquid surface, is normal to the liquid surface. Apparently owing to the attraction of liquid molecules below the surface, a film or a special layer seems to form on the liquid at the surface, which is in tension and small loads can be supported over it. For example, a small needle placed gently upon the water surface will not sink but will be supported by the tension at the water surface.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The property of the liquid surface film to exert a tension is called the surface tension. It is denoted by σ (Greek 'sigma') and it is the force required to maintain unit length of the film in equilibrium. In SI units surface tension is expressed in N/m. In the metric gravitational system of units it is expressed in kg(f)/cm or kg(f)/m. In the English gravitational system of units it is expressed in lb(f)/in. or lb(f)/ft.

As surface tension is directly dependent upon inter-molecular cohesive forces, its magnitude for all liquids decreases as the temperature rises. It is also dependent on the fluid in contact with the liquid surface; thus surface tensions are usually quoted in contact with air. The surface tension of water in contact with air varies from 0.0736 N/m [or 0.0075 kg (f)/m] at 19°C to 0.0589 N/m [or 0.006 kg (f)/m] at 100°C. More organic liquids have values of surface tension between 0.0206 N/m [or 0.0021 kg (f)/m] and 0.0304 N/m [or 0.0031 kg (f)/m] and mercury has a value of about 0.4944 N/m [or 0.0504 kg(f)/m], at normal temperature and the liquid in each case being in contact with air.

The effect of surface tension is illustrated in the case of a droplet as well as a liquid jet. When a droplet is separated initially from the surface of the main body of liquid, then due to surface tension there is a net inward force exerted over the entire surface of the droplet which causes the surface of the droplet to contract from all the sides and results in increasing the internal pressure within the droplet. The contraction of the droplet continues till the inward force due to surface tension is in balance with the internal pressure and the droplet forms into sphere which is the shape for minimum surface area. The internal pressure within a jet of liquid is also increased due to surface tension. The internal pressure intensity within a droplet and a jet of liquid in excess of the outside pressure intensity may be determined by the expressions derived below.

(i) *Pressure intensity inside a droplet*. Consider a spherical droplet of radius r having internal pressure intensity p in excess of the outside pressure intensity. If the droplet is cut into two halves, then the forces acting on one half will be those due to pressure intensity p on the projected area (πr^2) and the tensile force due to surface tension σ acting around the circumference ($2\pi r$). These two forces will be equal and opposite for equilibrium and hence we have

$$p(\pi r^2) = \sigma (2\pi r)$$

or
$$p = \frac{2\sigma}{r} \qquad ...(1.6)$$

Equation 1.6 indicates that the internal pressure intensity increases with the decrease in the size of droplet.

(ii) *Pressure intensity inside a soap bubble*. A spherical soap bubble has two surfaces in contact with air, one inside and the other outside, each one of which contributes the same amount of tensile force due to surface tension. As such on a hemispherical section of a soap bubble of radius r the tensile force due to surface tension is equal to $2\sigma (2\pi r)$. However, the pressure force acting on the hemispherical section of the soap bubble is same as in the case of a droplet and it is equal to $p (\pi r^2)$. Thus equating these two forces for equilibrium, we have

$$p (\pi r^2) = 2\sigma (2\pi r)$$

or
$$p = \frac{4\sigma}{r} \qquad ...(1.6\ a)$$

(iii) *Pressure intensity inside a liquid jet*. Consider a jet of liquid of radius r, length l and having internal pressure intensity p in excess of the outside pressure intensity. If the jet is cut into two

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

halves, then the forces acting on one half will be those due to pressure intensity p on the projected area ($2rl$) and the tensile force due to surface tension σ acting along the two sides ($2l$). These two forces will be equal and opposite for equilibrium and hence we have

$$p(2rl) = \sigma(2l)$$

or
$$p = \frac{\sigma}{r} \qquad ...(1.6\ b)$$

(b) Capillarity. If molecules of certain liquid possess, relatively, greater affinity for solid molecules, or in other words the liquid has greater adhesion than cohesion, then it will wet a solid surface with which it is in contact and will tend to rise at the point of contact, with the result that the liquid surface is concave upward and the angle of contact θ is less than 90° as shown in Fig. 1.4. For example, if a glass tube of small diameter is partially immersed in water, the water will wet the surface of the tube and it will rise in the tube to some height, above the normal water surface, with the angle of contact θ, being zero. The wetting of solid boundary by liquid results in creating decrease of pressure within the liquid, and hence the rise in the liquid surface takes place, so that the pressure within the column at the elevation of the surrounding liquid surface is the same as the pressure at this elevation outside the column.

On the other hand, if for any liquid there is less attraction for solid molecule or in other words the cohesion predominates, then the liquid will not wet the solid surface and the liquid surface will be depressed at the point of contact, with the result that the liquid surface is concave downward and the angle of contact θ is greater than 90° as shown in Fig. 1.4. For instance if the same glass tube is

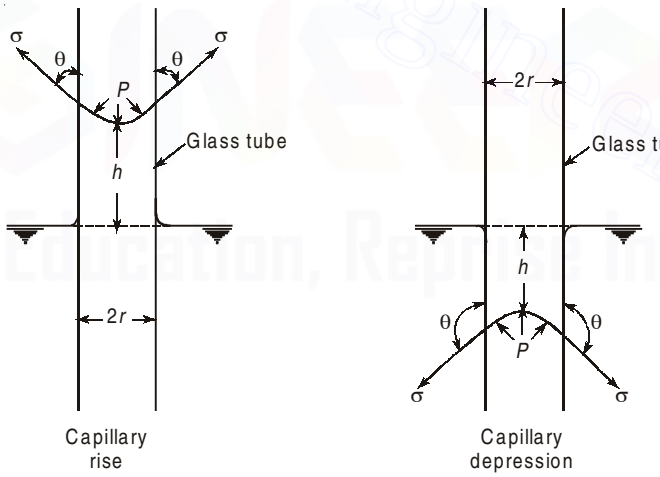


Figure 1.4 *Capillarity in circular glass tubes*

now inserted in mercury, since mercury does not wet the solid boundary in contact with it, the level of mercury inside the tube will be lower than the adjacent mercury level, with the angle of contact θ equal to about 130°. The tendency of the liquids which do not adhere to the solid surface, results in creating an increase of pressure across the liquid surface, (as in the case of a drop of liquid). It is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

because of the increased internal pressure, the elevation of the meniscus (curved liquid surface) in the tube is lowered to the level where the pressure is the same as that in the surrounding liquid.

Such a phenomenon of rise or fall of liquid surface relative to the adjacent general level of liquid is known as *capillarity*. Accordingly the rise of liquid surface is designated as capillary rise and the lowering of liquid surface as capillary depression, and it is expressed in terms of m or mm of liquid in SI units, in terms of cm or mm of liquid in the metric system of units and in terms of inch or ft of liquid in the English system of units.

The capillary rise (or depression) can be determined by considering the conditions of equilibrium in a circular tube of small diameter inserted in a liquid. It is supposed that the level of liquid has risen (or fallen) by h above (or below) the general liquid surface when a tube of radius r is inserted in the liquid, see Fig. 1.4. For the equilibrium of vertical forces acting on the mass of liquid lying above (or below) the general liquid level, the weight of liquid column h (or the total internal pressure in the case of capillary depression) must be balanced by the force, at surface of the liquid, due to surface tension σ. Thus equating these two forces we have

$$sw\pi r^2 h = 2\pi r\sigma \cos\theta$$

where w is the specific weight of water, s is specific gravity of liquid, and θ is the contact angle between the liquid and the tube. The expression for h the capillary rise (or depression) then becomes

$$h = \frac{2\sigma\cos\theta}{swr} \qquad ...(1.7)$$

As stated earlier, the contact angle θ for water and glass is equal to zero. Thus the value of $\cos\theta$ is equal to unity and hence h is given by the expression

$$h = \frac{2\sigma}{wr} \qquad ...(1.8)$$

Equation 1.7 for capillary rise (or depression) indicates that the smaller the radius r the greater is the capillary rise (or depression).

The above obtained expression for the capillary rise (or depression) is based on the assumption that the meniscus or the curved liquid surface is a section of a shpere. This is, however, true only in case of the tubes of small diameters ($r < 2.5$ mm) and as the size of the tube becomes larger, the meniscus becomes less spherical and also gravitational forces become more appreciable. Hence such simplified solution for computing the capillary rise (or depression) is possible only for the tubes of small diameters. However, with increasing diameter of tube, the capillary rise (or depression) becomes much less. It has been observed that for tubes of diameters 6 mm or more the capillary rise (or depression) is negligible. Hence in order to avoid a correction for the effects of capillarity in manometers, used for measuring pressures, a tube of diameter 6 mm or more should be used.

Another assumption made in deriving Eq. 1.7 is that the liquids and tube surfaces are extremely clean. In practice, however, such cleanliness is virtually never encountered and h will be found to be considerably smaller than that given by Eq. 1.7. In respect of this, Eq. 1.7 will provide a conservative estimate of capillary rise (or depression).

If a tube of radius r is inserted in mercury (sp. gr. s_1) above which a liquid of sp. gr. s_2 lies then by considering the conditions of equilibrium it can be shown that the capillary depression h is given by

$$h = \frac{2\sigma\cos\theta}{rw(s_1 - s_2)} \qquad ...(1.9)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

TABLE 1.8 *Properties of Water at Standard Atmospheric Pressure*

Temperature T		*Mass Density* ρ		*Specific Weight* w	*Dynamic Viscosity* μ			*Kinematic Viscosity* υ		*Bulk Modulus of Elasticity K*		*Surface Tension (in contact with air)* σ		*Vapour Pressure Pv*	
°C	K	kg/m³	m.slug/m³ or msl/m³	N/m³	kg(f)/m³	N.s/m² or Pa.s	gm(mass) cm-sec poise	kg(f)-sec m²	m²/s	cm²/s or stokes	N/m²or Pa	kg(f)/cm²	N/m	kg(f)/m	N/m² or Pa	kg(f)/cm²
						Values below to be multiplied by 10^{-3}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{-4}	Values below to be multiplied by 10^{-6}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{9}	Values below to be multiplied by 10^{4}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{-3}	Values below to be multiplied by 10^{3}	Values below to be multiplied by 10^{-2}
0	273.15	999.9	101.96	9 806	999.9	1.792	1.792	1.827	1.792	1.792	2.04	2.08	7.62	7.77	0.6	0.612
5	278.15	1000.0	101.97	9 807	1000.0	1.519	1.519	1.549	1.519	1.519	2.06	2.10	7.54	7.69	0.9	0.918
10	283.15	999.7	101.94	9 804	999.7	1.308	1.308	1.334	1.308	1.308	2.11	2.15	7.48	7.63	1.2	1.224
15	288.15	999.1	101.88	9 798	999.1	1.140	1.140	1.162	1.141	1.141	2.14	2.18	7.41	7.56	1.7	1.734
20	293.15	998.2	101.79	9 789	998.2	1.005	1.005	1.025	1.007	1.007	2.20	2.24	7.36	7.51	2.5	2.549
25	298.15	997.1	101.68	9 778	997.1	0.894	0.894	0.912	0.897	0.897	2.22	2.26	7.26	7.40	3.2	3.263
30	303.15	995.7	101.53	9 764	995.7	0.801	0.801	0.817	0.804	0.804	2.23	2.27	7.18	7.32	4.3	4.385
35	308.15	994.1	101.37	9 749	994.1	0.723	0.723	0.737	0.727	0.727	2.24	2.28	7.10	7.24	5.7	5.812
40	313.15	992.2	101.18	9 730	992.2	0.656	0.656	0.661	0.661	0.661	2.27	2.31	7.01	7.15	7.5	7.648
45	318.15	990.2	100.97	9 711	990.2	0.599	0.599	0.611	0.605	0.605	2.29	2.34	6.92	7.06	9.6	9.789
50	323.15	988.1	100.76	9 690	988.1	0.549	0.549	0.560	0.556	0.556	2.30	2.35	6.82	6.95	12.4	12.644
55	328.15	985.7	100.51	9 666	985.7	0.506	0.506	0.516	0.513	0.513	2.31	2.36	6.74	6.87	15.8	16.112
60	333.15	983.2	100.26	9 642	983.2	0.469	0.469	0.478	0.477	0.477	2.28	2.32	6.68	6.81	19.9	20.292
65	338.15	980.6	99.99	9 616	980.6	0.436	0.436	0.446	0.444	0.444	2.26	2.30	6.58	6.71	25.1	25.595
70	343.15	977.8	99.71	9 589	977.8	0.406	0.406	0.414	0.415	0.415	2.25	2.29	6.50	6.63	31.4	32.019
75	348.15	974.9	99.41	9 561	974.9	0.380	0.380	0.387	0.390	0.390	2.23	2.27	6.40	6.53	38.8	39.565
80	353.15	971.8	99.10	9 530	971.8	0.357	0.357	0.364	0.367	0.367	2.21	2.25	6.30	6.42	47.7	48.640
85	358.15	968.6	98.77	9 499	968.6	0.336	0.336	0.343	0.347	0.347	2.17	2.21	6.20	6.32	58.1	59.246
90	363.15	965.3	98.43	9 466	965.3	0.317	0.317	0.323	0.328	0.328	2.16	2.20	6.12	6.24	70.4	71.788
95	368.15	961.9	98.09	9 433	961.9	0.299	0.299	0.305	0.311	0.311	2.11	2.15	6.02	6.14	84.5	86.166
100	373.15	958.4	97.73	9 399	958.3	0.284	0.284	0.290	0.296	0.296	2.07	2.11	5.94	6.06	101.3	103.297

Standard Atmospheric Pressure = 101.325 kN/m²; 1 kg(f) = 9.806 65 N; 1 msl = 9.806 65 kg.

Downloaded From : www.EasyEngineering.net

TABLE 1.9 *Properties of Air at Standard Atmospheric Pressure*

Temperature T		*Mass Density* ρ		*Specific Weight* w		*Dynamic Viscosity* μ			*Kinematic Viscosity* υ	
°C	K	kg/m³	m.slug/m³ or msl/m³	N/m³	kg(f)/m³	N.s/m² or pa.s	$\frac{\text{gm (mass)}}{\text{cm-sec or poise}}$	$\frac{\text{kg(f)-sec}}{\text{m}^2}$	m²/s	cm²/s or stokes
–50	223.15	1.582	0.1613	15.514	1.582	1.46×10^{-5}	1.46×10^{-4}	1.489×10^{-6}	0.921×10^{-5}	0.921×10^{-1}
–40	233.15	1.514	0.1544	14.847	1.514	1.51×10^{-5}	1.51×10^{-4}	1.540×10^{-6}	0.998×10^{-5}	0.998×10^{-1}
–30	243.15	1.452	0.1481	14.239	1.452	1.56×10^{-5}	1.56×10^{-4}	1.591×10^{-6}	1.08×10^{-5}	1.08×10^{-1}
–20	253.15	1.394	0.1421	13.670	1.394	1.61×10^{-5}	1.61×10^{-4}	1.642×10^{-6}	1.16×10^{-5}	1.16×10^{-1}
–10	263.15	1.342	0.1368	13.161	1.342	1.67×10^{-5}	1.67×10^{-4}	1.703×10^{-6}	1.24×10^{-5}	1.24×10^{-1}
0	273.15	1.292	0.1317	12.670	1.292	1.72×10^{-5}	1.72×10^{-4}	1.754×10^{-6}	1.33×10^{-5}	1.33×10^{-1}
10	283.15	1.247	0.1272	12.229	1.247	1.76×10^{-5}	1.76×10^{-4}	1.795×10^{-6}	1.42×10^{-5}	1.42×10^{-1}
20	293.15	1.204	0.1228	11.807	1.204	1.81×10^{-5}	1.81×10^{-4}	1.846×10^{-6}	1.51×10^{-5}	1.51×10^{-1}
30	303.15	1.164	0.1187	11.415	1.164	1.86×10^{-5}	1.86×10^{-4}	1.897×10^{-6}	1.60×10^{-5}	1.60×10^{-1}
40	313.15	1.127	0.1149	11.052	1.127	1.91×10^{-5}	1.91×10^{-4}	1.948×10^{-6}	1.69×10^{-5}	1.69×10^{-1}
50	323.15	1.092	0.1114	10.709	1.092	1.95×10^{-5}	1.95×10^{-4}	1.988×10^{-6}	1.79×10^{-5}	1.79×10^{-1}
60	333.15	1.060	0.1081	10.395	1.060	2.00×10^{-5}	2.00×10^{-4}	2.039×10^{-6}	1.89×10^{-5}	1.89×10^{-1}
70	343.15	1.030	0.1050	10.101	1.030	2.05×10^{-5}	2.05×10^{-4}	2.090×10^{-6}	1.99×10^{-5}	1.99×10^{-1}
80	353.15	1.000	0.1020	9.807	1.000	2.09×10^{-5}	2.09×10^{-4}	2.131×10^{-6}	2.09×10^{-5}	2.09×10^{-1}
90	363.15	0.937	0.0992	9.542	0.973	2.13×10^{-5}	2.13×10^{-4}	2.172×10^{-6}	2.19×10^{-5}	2.19×10^{-1}
100	373.15	0.946	0.0965	9.277	0.946	2.17×10^{-5}	2.17×10^{-4}	2.213×10^{-6}	2.30×10^{-5}	2.30×10^{-1}
150	423.15	0.834	0.0850	8.179	0.834	2.38×10^{-5}	2.38×10^{-4}	2.427×10^{-6}	2.85×10^{-5}	2.85×10^{-1}
200	473.15	0.746	0.0761	7.316	0.746	2.57×10^{-5}	2.57×10^{-4}	2.621×10^{-6}	3.45×10^{-5}	3.45×10^{-1}
250	523.15	0.675	0.0688	6.619	0.675	2.75×10^{-5}	2.75×10^{-4}	2.804×10^{-6}	4.08×10^{-5}	4.08×10^{-1}
300	573.15	0.616	0.0628	6.041	0.616	2.93×10^{-5}	2.93×10^{-4}	2.988×10^{-6}	4.75×10^{-5}	4.75×10^{-1}

Standard atmospheric pressure = 101.325 kN/m²; 1 kg(f) = 9.806 65 N; 1 msl = 9.806 65 kg

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

TABLE 1.10 *Properties of Some Common Liquids at 20°C and Atmospheric Pressure*

Liquid	*Mass Density* ρ		*Specific Weight* w		*Dynamic Viscosity* μ			*Kinematic Viscosity* υ		*Bulk Modulus of Elasticity* K		*Surface Tension (in contact with air)* σ		*Vapour Pressure* P_v	
	kg/m^3	$m.slug/m^3$ or msl/m^3	N/m^3	$kg(f)/m^3$	$N.s/m^2$ or Pa.s	$\frac{gm(mass)}{cm\text{-}sec}$ or poise	$\frac{kg(f)\text{-}s}{m^2}$	m^2/s	cm^2/s or stokes	N/m^2 or Pa	$kg(f)/cm^2$	N/m	$kg(f)/m$	N/m^2 or Pa	$kg(f)/cm^2$
										Values below to be multiplied by 10^6	Values below to be multiplied by 10^4	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{-3}		
Glycerine	1260	128.48	12356	1260	14.95×10^{-1}	14.95	1.52×10^{-1}	11.87×10^{-4}	11.87	4.35	4.44	6.30	6.42	1.37×10^{-2}	1.40×10^{-7}
Corbon tetra-chloride	1594	162.54	15632	1594	0.97×10^{-3}	0.97×10^{-2}	0.99×10^{-4}	6.04×10^{-7}	6.04×10^{-3}	1.10	1.12	2.67	2.72	1.31×10^{4}	1.34×10^{-1}
Kerosene	800	81.58	7845	800	1.92×10^{-3}	1.92×10^{-2}	1.96×10^{-4}	2.40×10^{-6}	2.40×10^{-2}	1.62	1.65	2.60	2.65	3.30×10^{3}	3.37×10^{-2}
Benzene	879	89.63	8620	879	6.53×10^{-4}	6.53×10^{-3}	6.66×10^{-5}	7.43×10^{-7}	7.43×10^{-3}	1.03	1.05	2.89	2.95	1.00×10^{4}	1.02×10^{-1}
Castor Oil	960	97.89	9414	960	9.80×10^{-1}	9.80	9.99×10^{-2}	10.00×10^{-4}	10.00	1.44	1.47	3.92	4.00	—	—
Ethyl Alcohol	789	80.46	7737	789	1.20×10^{-3}	1.20×10^{-2}	1.22×10^{-4}	1.52×10^{-6}	1.52×10^{-2}	1.21	1.23	2.23	2.27	5.90×10^{3}	6.02×10^{-2}
Mercury	13550	1381.72	132880	13550	1.60×10^{-3}	1.60×10^{-2}	1.63×10^{-4}	1.18×10^{-7}	1.18×10^{-3}	26.20	26.72	51.00	52.00	1.60×10^{-1}	1.63×10^{-6}

Standard atmospheric pressure = 101.325 kN/m^2; 1 kg(f) = 9.806 65 N; 1 msl = 9.806 65 kg.

Downloaded From : www.EasyEngineering.net
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in which σ is the surface tension of mercury in contact with the liquid and rest of the notation are same as defined earlier.

Further if two vertical parallel plates t distance apart and each of width l are held partially immersed in a liquid of surface tension σ and sp. gr. s, then the capillary rise (or depression) h may be determined by equating the weight of the liquid column h (or the total internal pressure in the case of capillary depression) ($swhlt$) to the force due to surface tension ($2\sigma l \cos\theta$). Thus we have

$$swhlt = 2\sigma l \cos\theta$$

or
$$h = \frac{2\sigma \cos\theta}{swt} \qquad ...(1.10)$$

In Tables 1.8 and 1.9 properties of water and air respectively at different temperatures are listed. Table 1.10 gives the properties of some of the common liquids such as glycerine, kerosene, alcohol, mercury etc., at 20°C.

ILLUSTRATIVE EXAMPLES

Example 1.1. *If 5 m^3 of a certain oil weighs 4000 kg(f). Calculate the specific weight, mass density and specific gravity of this oil.*

Solution

Specific weight of oil $= \dfrac{\text{Weight}}{\text{Volume}}$

$$= \frac{4000 \text{ kg(f)}}{5 \text{ m}^3} = 800 \text{ kg(f)/m}^3$$

Mass density of oil $= \dfrac{\text{Specific weight of oil}}{\text{Acceleration due to gravity}}$

$$= \frac{800 \text{ kg(f)/m}^3}{9.81 \text{ m/sec}^2} = 81.55 \text{ msl/m}^3$$

Specific gravity of oil $= \dfrac{\text{Specific weight of oil}}{\text{Specific weight of water}}$

$$= \frac{800 \text{ kg(f)/m}^3}{1000 \text{ kg(f)/m}^3} = 0.8$$

Example 1.2. *If 5 m^3 of a certain oil weighs 40 kN, calculate the specific weight, mass density and specific gravity of this oil.*

Solution

Specific weight of oil $= \dfrac{\text{Weight}}{\text{Volume}} = \dfrac{40 \times 1000 \text{ N}}{5 \text{ m}^3} = 8000 \text{ N/m}^3$

Mass density of oil $= \dfrac{\text{Specific weight of oil}}{\text{Acceleration due to gravity}}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{8000 \text{ N/m}^3}{9.81 \text{ m/s}^2} = 815.49 \text{ kg/m}^3$$

$$\text{Specific gravity of oil} = \frac{\text{Specific weight of oil}}{\text{Specific weight of water}}$$

$$= \frac{8000 \text{ N/m}^3}{9810 \text{ N/m}^3} = 0.815$$

Example 1.3. *Carbon-tetra chloride has a mass density of 1594 kg/m³. Calculate its mass density, specific weight and specific volume in the metric, and the English gravitational systems of units. Also calculate its specific gravity.*

Solution

Mass density of carbon-tetra chloride = 1594 kg/m³

Since $1 \text{ kg} = \frac{1}{9.81} \text{ msl}$

∴ Mass density of carbon tetra chloride in the metric gravitational system of units

$$= \frac{1594}{9.81} = 162.49 \text{ msl/m}^3$$

Acceleration due to gravity = 9.81 m/sec²

∴ Specific weight of carbon-tetra chloride in the metric gravitational system of units = 162.49 × 9.81 = 1594 kg(f)/m³.

Specific volume of carbon-tetra chloride in the metric gravitational system of units

$$= \frac{1}{\text{Specific weight}}$$

$$= \frac{1}{1594} = 6.274 \times 10^{-4} \text{ m}^3/\text{kg(f)}$$

Since $1 \text{ kg(f)} = 2.205 \text{ lb(f)}$

and $1 \text{ m} = 3.281 \text{ ft}$

∴ Specific weight of carbon-tetra chloride in the English gravitational system of units

$$= \frac{1594 \times 2.205}{(3.281)^3} = 99.51 \text{ lb(f)/ft}^3$$

Acceleration due to gravity = 32.2 ft/sec²

∴ Mass density of carbon-tetra chloride in the English gravitational system of units

$$= \frac{99.51}{32.2} = 3.09 \text{ slugs/ft}^3$$

Specific volume of carbon-tetra chloride in the English gravitational system of units

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{1}{\text{Specific weight}} = \frac{1}{99.51} = 1.005 \times 10^{-2}\ \text{ft}^3/\ \text{lb(f)}$$

$$\text{Specific gravity} = \frac{\text{Mass density of carbon tetra chloride}}{\text{Mass density of water}}$$

Mass density of carbon tetra chloride in SI units = 1594 kg/m^3
Mass density of water in SI units = 1000 kg/m^3

$$\therefore \text{ Specific gravity} = \frac{1594\ \text{kg/m}^3}{1000\ \text{kg/m}^3} = 1.594$$

Example 1.4. *A plate 0.0254 mm distant from a fixed plate, moves at 61 cm/sec and requires a force of 0.2 kg(f)/m^2 to maintain this speed. Determine the dynamic viscosity of the fluid between the plates.*

Solution

From Eq. 1.3, shear stress

$$\tau = \frac{F}{A} = \mu \frac{dv}{dy} = \mu \frac{V}{Y}$$

$$\tau = \frac{F}{A} = 0.2\ \text{kg(f)/m}^2$$

$$V = 61\ \text{cm/sec} = 0.61\ \text{m/sec}$$

and $$Y = 0.0254\ \text{mm} = 2.54 \times 10^{-5}\ \text{m}$$

By substituting in the above equation, we get

$$0.2 = \mu \times \frac{0.61}{2.54 \times 10^5}$$

$$\therefore \quad \mu = \frac{0.2 \times 2.54 \times 10^5}{0.61}\ \text{kg(f)-sec/m}^2$$

$$= 8.328 \times 10^{-6}\ \text{kg(f)-sec/m}^2$$

$$= 8.328 \times 10^{-10}\ \text{kg(f)-sec/cm}^2$$

Example 1.5. *At a certain point in castor oil the shear stress is 0.216 N/m^2 and the velocity gradient 0.216s^{-1}. If the mass density of castor oil is 959.42 kg/m^3, find kinematic viscosity.*

Solution

From Eq. 1.3 shear stress

$$\tau = \mu \left(\frac{dv}{dy}\right)$$

$$\tau = 0.216\ \text{N/m}^2;\ \left(\frac{dv}{dy}\right) = 0.216\ \text{s}^{-1}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substitution, we get

$$0.216 = \mu\,(0.216)$$

$$\therefore \quad \mu = 1 \text{ N.s/m}^2$$

$\therefore$ Kinematic viscosity

$$\upsilon = \frac{\mu}{\rho} = \frac{1}{959.42} = 1.042 \times 10^{-3} \text{ m}^2/\text{s}$$

Example 1.6. *If a certain liquid has viscosity 4.9×10^{-4} kg(f)-sec/m^2 and kinematic viscosity 3.49×10^{-2} stokes, what is its specific gravity ?*

Solution

Kinematic viscosity $\upsilon = \dfrac{\mu}{\rho}$

or Mass density $\rho = \dfrac{\mu}{\upsilon}$

$\mu = 4.9 \times 10^{-4}$ kg(f)-sec/m^2

$\upsilon = 3.49 \times 10^{-2}$ stokes

$= 3.49 \times 10^{-6}$ m^2/sec

$$\therefore \quad \rho = \frac{4.9 \times 10^4}{3.49 \times 10^6} = 140.4 \text{ msl/m}^3$$

$$\therefore \text{ Sp. gr. of the liquid} = \frac{\text{Mass density of liquid}}{\text{Mass density of water}}$$

$$= \frac{140.4}{102} = 1.38$$

Example 1.7. *The kinematic viscosity and specific gravity of a certain liquid are 5.58 stokes (5.58×10^{-4} m^2/s) and 2.00 respectively. Calculate the viscosity of this liquid in both metric gravitational and SI units.*

Solution

(a) *Metric gravitational units*

Sp. gr. of the liquid = 2.00

Mass density of water = 102 msl/m^3

$\therefore$ Mass density of the liquid = (2 × 102) = 204 msl/m^3

Kinematic viscosity of the liquid = 5.58 stokes

$= 5.58 \times 10^{-4}$ m^2/sec

$\therefore$ Viscosity of the liquid

$\mu = \upsilon \times \rho$

$= (5.58 \times 10^{-4} \times 204)$ kg(f)-sec/m^2

$= 0.114$ kg(f)–sec/m^2

(b) *SI units*

Specific gravity of the liquid = 2.00

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Mass density of water = 1000 kg/m^3

∴ Mass density of the liquid = (2 × 1000) = 2000 kg/m^3

Kinematic viscosity of the liquid = 5.58×10^{-4} m^2/s

∴ Viscosity of the liquid

$$\mu = \upsilon \times \rho$$
$$= (5.58 \times 10^{-4} \times 2000) \text{ N-s/m}^2$$
$$= 1.116 \text{ N.s/m}^2$$

Example 1.8. *A rectangular plate of size 25 cm by 50 cm and weighing 25 kg(f) slides down a 30° inclined surface at a uniform velocity of 2 m/sec. If the uniform 2 mm gap between the plate and the inclined surface is filled with oil determine the viscosity of the oil.*

Solution

When the plate is moving with a uniform velocity of 2 m/sec, the viscous resistance to the motion is equal to the component of the weight of the plate along the sloping surface. Component of the weight of the plate along the slope = 25 sin 30° = 12.5 kg(f)

Viscous resistance $= (\tau \times A)$

$$= \mu \frac{dv}{dy} \times A = \mu \frac{V}{y} \times A$$

V = 2 m/sec ; $y = 2 \times 10^{-3}$ m; and $A = (0.25 \times 0.5)$ m^2

By substituting these values, we get

$$\text{Viscous resistance} = \mu \times \frac{2}{2 \times 10^3} (0.25 \times 0.5) = 125 \, \mu\text{kg(f)}$$

Equating the two, we get $125\,\mu = 12.5$

∴ $\mu = 0.1$ kg(f)–sec/m^2

Example 1.9. *A cubical block of 20 cm edge and weight 20 kg(f) is allowed to slide down a plane inclined at 20° to the horizontal on which there is thin film of oil of viscosity 0.22 × 10^{-3} kg(f)-s/m^2. What terminal velocity will be attained by the block if the fill thickness is estimated to be 0.025 mm?*

Solution

The force causing the downward motion of the block is

$$F = W \sin 20° = (20 \times 0.3420) = 6.84 \text{ kg(f)}$$

which will be equal and opposite to shear resistance.

$$\therefore \quad \tau = \frac{F}{A} = \frac{6.84}{(0.20 \times 0.20)} = 171 \text{ kg(f)/m}^2$$

Further from Eq. 1.3 we have $\tau = \mu \frac{dv}{dy} = \mu \frac{V}{y}$

$\mu = 0.22 \times 10^{-3}$ kg(f)-s/m^2 ; $y = 0.025$ mm $= 0.025 \times 10^{-3}$ m

Thus by substitution we get $171 = \frac{0.22 \times 10^{-3} V}{0.025 \times 10^{-3}}$

∴ $V = 19.43$ m/sec.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 1.10. *A cylinder of 0.30 m diameter rotates concentrically inside a fixed cylinder of 0.31 m diameter. Both the cylinders are 0.3 m long. Determine the viscosity of the liquid which fills the space between the cylinders if a torque of 0.98 N.m is required to maintain an angular velocity of 2π rad/s* (*or 60 r.p.m., since angular velocity* $\omega = \frac{2\pi N}{60}$ *where N is speed of rotation in r.p.m.*).

Solution

Tangential velocity of the inner cylinder

$$V = r\omega = 0.15 \times 2\pi = 0.942 \text{ m/s}$$

For the small space between the cylinders the velocity profile may be assumed to be linear, then

$$\frac{dv}{dy} = \frac{V}{y} = \frac{0.942}{(0.155 - 0.15)} = 188.4 \text{ s}^{-1}$$

The torque applied to maintain the constant angular velocity is equal to the torque resisted due to shear stress.

Torque resisted $= \tau \times (2\pi \times 0.15 \times 0.30) \times 0.15$

Thus $0.98 = \tau \times (2\pi \times 0.15 \times 0.30) \times 0.15$

$\therefore \quad \tau = 23.11 \text{ N/m}^2$

From Eq. 1.3 $\quad \tau = \mu \frac{dv}{dy}$

$\therefore \quad \mu = \frac{\tau}{(dv/dy)}$

$$= \frac{23.11}{188.4} = 0.123 \text{ N.s/m}^2$$

Example 1.11. *Through a very narrow gap of height h, a thin plate of large extent is pulled at a velocity V. On one side of the plate is oil of viscosity* μ_1 *and on the other side oil of viscosity* μ_2*. Calculate the position of the plate so that* (*i*) *the shear force on the two sides of the plate is equal ;* (*ii*) *the pull required to drag the plate is minimum.*

Solution

Let *y* be the distance of the thin plate from one of the surfaces as shown in Fig. Ex. 11.

(i) Force per unit area on the upper surface of the plate

$$= \mu_1 \left(\frac{dv}{dy}\right) = \mu_1 \frac{V}{(h - y)}$$

Force per unit area on the bottom surface of the plate

$$= \mu_2 \left(\frac{dv}{dy}\right) = \mu_2 \frac{V}{y}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

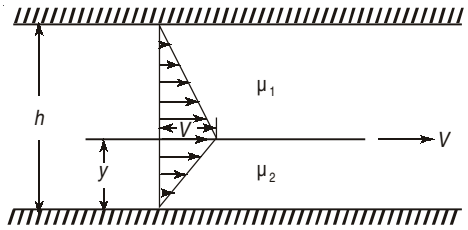


Figure Ex.1.11

Equating the two, we get

$$\mu_1 \frac{V}{h-y} = \mu_2 \frac{V}{y}$$

$$\therefore \quad y = \frac{\mu_2 h}{(\mu_1 + \mu_2)}$$

(ii) Let F be the pull per unit area required to drag the plate, then

$$F = \mu_1 \left(\frac{V}{h-y}\right) + \mu_2 \left(\frac{V}{y}\right)$$

F = Sum of the shear forces per unit area on both the surfaces of the plate

For the force F to be minimum

$$\frac{dF}{dy} = 0$$

or
$$\frac{dF}{dy} = \frac{\mu_1 V}{(h-y)^2} - \frac{\mu_2 V}{y^2} = 0$$

or
$$y = \frac{h}{1+\sqrt{(\mu_1/\mu_2)}}$$

Example 1.12. *If the equation of a velocity profile over a plate is $v = 2y^{2/3}$; in which v is the velocity in m/s at a distance of y metres above the plate, determine the shear stress at $y = 0$ and $y = 0.075$ m (or 7.5 cm). Given $\mu = 0.835$ N.s/m^2 (or 8.35 poise).*

Solution

The velocity profile over the plate is

$$v = 2y^{2/3}$$

$$\therefore \quad \frac{dv}{dy} = 2 \times \frac{2}{3} \times y^{-2/3}\left(\frac{4}{3}\right) y^{-1/3}$$

Shear stress
$$\tau = \mu\left(\frac{dv}{dy}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) SI units $\tau = 0.835 \times \frac{4}{3} y^{-1/3}$

At $y = 0$

$\tau = \infty$ (infinite)

At $y = 0.075$ m

$$\tau = 0.835 \times \frac{4}{3} \times \frac{1}{(0.075)^{1/3}} = 2.64 \text{ N/m}^2$$

(b) Metric gravitational units

$\mu = 8.35$ poise

$$= \frac{8.35 \times 0.102}{10} \text{ kg(f)-s/m}^2$$

$$= 8.517 \times 10^{-2} \text{ kg(f)-s/m}^2$$

$$\tau = 8.517 \times 10^{-2} \times \frac{4}{3} y^{-1/3}$$

At $y = 0$

$\tau = \infty$ (infinite)

At $y = 7.5$ cm $= 0.075$ m

$$\tau = 8.517 \times 10^{-2} \times \frac{4}{3} \times \frac{1}{(0.075)^{1/3}}$$

$$= 0.269 \text{ kg(f)/m}^2$$

Example 1.13. *If the pressure of a liquid is increased from 75 kg(f)/cm^2 to 140 kg(f)/cm^2, the volume of the liquid decreases by 0.147 per cent. Determine the bulk modulus of elasticity of the liquid.*

Solution

From Eq. 1.5, bulk modulus of elasticity

$$K = -\frac{dp}{(dV/V)}$$

$$dp = (140 - 75) = 65 \text{ kg(f)/cm}^2$$

and $$\frac{dV}{V} = -\frac{0.147}{100} = -0.00147$$

$$\therefore \quad K = \frac{65}{0.00147} = 4.42 \times 10^4 \text{ kg(f)/cm}^2$$

Example 1.14. *A liquid compressed in a cylinder has a volume of 0.0113 m^3 at 6.87 × 106 N/m^2 (6.87 MN/m^2) pressure and a volume of 0.0112 m^3 at 13.73 × 10^6 N/m^2 (13.73 MN/m^2) pressure. What is its bulk modulus of elasticity?*

Solution

From Eq. 1.5, bulk modulus of elasticity

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$K = -\frac{dp}{(dV/V)}$$

$$dp = (13.73 \times 10^6 - 6.87 \times 10^6) = 6.86 \times 10^6 \text{ N/m}^2$$

$$dV = (0.0112 - 0.0113) = -0.0001/\text{m}^3$$

and $V = 0.0113 \text{ m}^3$

$$\therefore \quad K = \frac{6.86 \times 10^6 \times 0.0113}{0.0001}$$

$$= 7.75 \times 108 \text{ N/m}^2 \ (0.775 \text{ GN/m}^2)$$

Example 1.15. *At a depth of 2 kilometres in the ocean the pressure is 840 kg(f)/cm^2. Assume the specific weight at surface as 1025 kg(f)/m^3 and that the average bulk modulus of elasticity is 24 × 10^3 kg(f)/cm^2 for that pressure range. (a) What will be the change in specific volume between that at the surface and at that depth? (b) What will be the specific volume at that depth? (c) What will be the specific weight at that depth?*

Solution

Bulk modulus of elasticity

$$K = -\frac{dp}{(dV/V)}$$

$$K = 24 \times 10^3 \text{ kg(f)/cm}^2$$

and $dp = 840 \text{ kg/cm}^2$

$$\therefore \quad \frac{(dV)}{V} = -\frac{840}{24 \times 10^3} = -0.035$$

The negative sign corresponds to a decrease in the volume with increase in pressure.

The specific volume of the water at the surface of the ocean

$$= \frac{1}{1025} \text{ m}^3/\text{kg(f)}$$

∴ The change in specific volume between that at the surface and at that depth is

$$dV = \frac{0.035}{1025} = 3.41 \times 10^{-5} \text{ m}^3/\text{kg(f)}$$

The specific volume at that depth will be thus equal to

$$V_1 = \left(\frac{1}{1025} - \frac{0.035}{1025}\right) = 9.41 \times 10^{-4} \text{ m}^3/\text{kg(f)}.$$

The specific weight of water at that depth is

$$\frac{1}{V_1} = \frac{1}{9.41 \times 10^4} = 1063 \text{ kg(f)/m}^3.$$

Example 1.16. *What should be the diameter of a droplet of water, if the pressure inside is to be 0.0018 kg(f)/cm^2 greater than the outside? Given the value of surface tension of water in contact with air at 20°C as 0.0075 kg(f)/m.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

The internal pressure intensity p in excess of the outside pressure is given by Eq. 1.6 as

$$p = \frac{2\sigma}{r}$$

or $$r = \frac{2\sigma}{p}$$

or $$2r = \frac{4\sigma}{p}$$

$$\sigma = 0.0075 \text{ kg (f)/m}$$

and $$p = 0.0018 \text{ kg(f)/cm}^2$$

By substitution, we get

$$2r = d = 4 \times \frac{0.0075}{100} \times \frac{1}{0.0018} \text{ cm}$$

$$= \frac{4 \times 0.0075 \times 10}{100 \times 0.0018} \text{ mm}$$

$$= 1.67 \text{ mm.}$$

Example 1.17. *What is the pressure within a droplet of water 0.05 mm in diameter at 20°C, if the pressure outside the droplet is standard atmospheric pressure of 1.03 kg(f)/cm² ? Given σ = 0.0075 kg(f)/m for water at 20°C.*

Solution

From Eq. 1.6 the internal pressure intensity p in excess of the outside pressure is given as

$$p = \frac{2\sigma}{r}$$

$$\sigma = 0.0075 \text{ kg(f)/m}$$

$$= \frac{0.0075}{100} \text{ kg(f)/cm}$$

$$r = \frac{0.05}{2} = 0.025 \text{ mm} = \frac{0.025}{10} \text{ cm}$$

By substitution, we get $p = 2 \times \frac{0.0075}{100} \times \frac{10}{0.025} = 0.06 \text{ kg(f)/cm}^2$

The pressure intensity outside the droplet of water = 1.03 kg(f)/cm²

∴ The pressure intensity within the droplet of water = (1.03 + 0.06) = 1.09 kg(f)/cm².

Example 1.18. *Calculate the capillary rise in a glass tube of 2 mm diameter when immersed in (a) water, (b) mercury. Both the liquids being at 20°C and the values of the surface tensions for water and mercury at 20°C in contact with air are respectively 0.0075 kg(f)/m and 0.052 kg(f)/m.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

From Eq. 1.7 the capillary rise (or depression) is given as $h = \dfrac{2\sigma\cos\theta}{swr}$

(a) For water

$\theta = 0, \cos\theta = 1$

$\sigma = 0.0075$ kg(f)/m

$= \dfrac{0.0075}{100}$ kg(f)/cm

$sw = 1000$ kg(f)/m^3

$= 0.001$ kg(f)/cm^3

$r = 1$ mm $= 0.1$ cm

By substitution, we get $h = 2 \times \dfrac{0.0075}{100} \times \dfrac{1}{0.001 \times 0.1} = 1.5$ cm.

(b) For mercury

$\theta = 130° \times \cos\theta = -0.6428$

$\sigma = 0.052$ kg(f)/m

$= \dfrac{0.052}{100}$ kg(f)/cm

$sw = (13.6 \times 1000)$ kg(f)/m^3

$= (13.6 \times 0.001)$ kg(f)/cm^3

$r = 1$ mm $= 0.1$ cm

By substitution, we get $h = 2 \times \dfrac{0.052}{100} \times \dfrac{(-0.6428)}{13.6 \times 0.001 \times 0.1} = -0.492$ cm.

The negative sign in the case of mercury indicates that there is capillary depression.

Note. Often the value of contact angle for mercury is taken as 180°; in which case $\cos\theta = -1$ and the capillary depression becomes

$$h = -\frac{2 \times 0.052 \times 1}{100 \times 13.6 \times 0.001 \times 0.1} = -0.765 \text{ cm.}$$

Example 1.19. *Determine the minimum size of glass tubing that can be used to measure water level, if the capillary rise in the tube is not to exceed 0.25 cm. Take surface tension of water in contact with air as 0.0075 kg(f)/m.*

Solution

From Eq. 1.8 capillary rise $h = \dfrac{2\sigma}{swr}$ $\sigma = 0.0075$ kg(f)/m $= \dfrac{0.0075}{100}$ kg(f)/cm

$sw = 1000$ kg(f)/m$^3 = 0.001$ kg(f)/cm^3

$h = 0.25$ cm

By substitution, we get $0.25 = \dfrac{2 \times 0.0075}{0.001 \times 100 \times r}$

$\therefore \quad r = 0.6$ cm

Thus minimum diameter of the tube is 1.2 cm.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 1.20 *In measuring the unit surface energy of a mineral oil (sp. gr. 0.85) by the bubble method, air is forced to form a bubble at the lower end of a tube of internal diameter 1.5 mm immersed at a depth of 1.25 cm in the oil. Calculate the unit surface energy if the maximum bubble pressure is 15 kg(f)/m².*

Solution

The effective pressure attributable to surface tension is

$$p = \left[15 - \frac{(0.85 \times 1000 \times 1.25)}{100}\right] = 4.375 \text{ kg(f)/m}^2$$

From Eq. 1.6 $p = \dfrac{2\sigma}{r}$

The radius of the bubble is taken equal to that of the tube, thus by substitution, we get

$$4.375 = \frac{2 \times \sigma}{0.75 \times 10^{-3}}$$

$$\therefore \quad \sigma = 0.001\,64 \text{ kg(f)/m.}$$

Example 1.21. *Calculate the capillary effect in mm in a glass tube 3 mm in diameter when immersed in (a) water (b) mercury. Both the liquids are at 20°C and the values of the surface tensions for water and mercury at 20°C in contact with air are respectively 0.0736 N/m and 0.51 N/m. Contact angle for water = 0° and for mercury = 130°.*

Solution

From Eq. 1.7 the capillary rise (or depression) is given as $h = \dfrac{2\sigma \cos\theta}{swr}$

(a) For water $\theta = 0$, $\cos\theta = 1$

$\sigma = 0.0736$ N/m

$sw = 9810$ N/m³

$$r = \frac{3}{2} = 1.5 \text{ mm} = 1.5 \times 10^{-3} \text{ m}$$

By substitution, we get $h = \dfrac{2 \times 0.0736 \times 1}{9810 \times 1.5 \times 10^{-3}}$

$= 1.00 \times 10^{-2}$ m

$= 10$ mm

(b) For mercury $\theta = 130°$, $\cos\theta = -0.6428$

$\sigma = 0.51$ N/m

$sw = (13.6 \times 9810)$ N/m³

$$r = \frac{3}{2} = 1.5 \text{ mm} = 1.5 \times 10^{-3} \text{ m}$$

By substitution, we get $h = \dfrac{2 \times 0.51 \times (-0.6428)}{13.6 \times 9810 \times 1.5 \times 10^{-3}}$

$= -3.276 \times 10^{-3}$ m

$= -3.276$ mm

The negative (–) sign in the case of mercury indicates that there is capillary depression.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. Mass density or specific mass of a fluid is the mass per unit volume of the fluid. It is denoted by ρ.
2. Specific weight or weight density of a fluid is the weight per unit volume of the fluid. It is denoted by w or γ. Thus
$$w \text{ (or } \gamma) = \rho g$$
3. Specific volume of a fluid is the volume per unit weight (or volume per unit mass) of the fluid. It is denoted by v and it is reciprocal of specific weight (or specific mass).
Thus
$$v = \frac{1}{w}; \quad \text{or } v = \frac{1}{\rho}$$
4. Shear stress τ is proportional to the velocity gradient $\frac{dv}{dy}$. Thus
$$\tau = \mu \frac{dv}{dy}$$
where μ is the constant of proportionality which is called coefficient of viscosity, or dynamic viscosity.
The SI units of μ are N.s/m^2 , kg/m.s, or poise (= g/cm.s) or Pa.s,
(where Pa = pascal = N/m^2).
5. Kinematic viscosity υ is given by $\upsilon = \frac{\mu}{\rho}$.
The SI units of υ are m^2/s, or stoke (= cm^2/s).
6. The mass density ρ of a perfect gas is related to its absolute pressure p and absolute temperature T by equation of state as
$$p = \rho RT$$
where R = gas constant. For air in SI units
$$R = 287 \frac{\text{N.m}}{\text{kg°K}} = 287 \frac{\text{J}}{\text{kg°K}} = 287 \frac{\text{m}^2}{\text{S}^2\text{°K}}$$
and in metric units
$$R = (287/9.81) \text{ or } 29.27 \frac{\text{kg(f)°m}}{\text{kg(m)°K}}$$
7. Compressibility of a fluid is reciprocal of the bulk modulus of elasticity K of the fluid which is defined as $K = \frac{dp}{\left(\frac{dV}{V}\right)}$.
8. Surface tension is the force required to maintain unit length of a film at the liquid surface in equlibrium. It is denoted by σ.
The relation between surface tension σ and difference of pressure p between inside and outside of :
(i) a liquid drop of radius r is given as
$$p = \frac{2\sigma}{\gamma};$$
(ii) a soap bubble of radius r is given as
$$p = \frac{2\sigma}{r}; \text{ and}$$
(iii) a liquid jet of radius r is given as
$$p = \frac{\sigma}{r}$$
9. Capillary rise (or depression) h of a liquid of specific weight w in a tube of radious r is given by
$$h = \frac{2\sigma\cos\theta}{wr}.$$
The value of contact angle θ for water and glass is equal to zero and for mercury and glass is 130°.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

PROBLESMS

1.1 If a certain liquid has a mass density of 129 msl/m^3, what are the values of its specific weight, specific gravity and specific volume in metric gravitational and metric absolute systems of units.

[*Ans.*1266 kg(f)/m^3; 1.266; 7.9 × 10^{-4} m^3/kg(f); 1242 dynes/cm^3; 1266; 8.05 × 10^{-4} cm^3/dyne]

1.2 If 5.27 m^3 of a certain oil weighs 44 kN, calculate the specific weight ,mass density and specific gravity of the oil.

[*Ans.* 8349 N/m^3; 851.09 kg/m^3; 0.851]

1.3 The specific gravity of a liquid is 3.0, what are its specific weight, specific mass and specific volume.

[*Ans.* 3000 kg(f)/m^3; 305.8 msl/m^3; 0.33 × 10^{-3} m^3/kg(f); 29.43 kN/m^3; 3000 kg/m^3; 3.398 × 10^{-5} m^3/N]

1.4 A certain liquid has a dynamic viscosity of 0.073 poise and specific gravity of 0.87. Compute the kinematic viscosity of the liquid in stokes and also in m^2/s.

[*Ans.* 0.0839 stokes; 0.0839 × 10^{-4} m^2/s]

1.5 If a certain liquid has a viscosity of 0.048 poise and kinematic viscosity 3.50 × 10^{-2} stokes, what is its specific gravity? [*Ans.* 1.371]

1.6 In a stream of glycerine in motion at a certain point the velocity gradient is 0.25 s^{-1}. If for fluid ρ = 129.3 msl/m^3 (1268.4 kg/m^3) and v = 6.30 × 10^{-4} $m^{2/s}$, calculate the shear stress at the point.

[*Ans.* 0.02036 kg (f)/m^2 ; 0.19977 N/m^2]

1.7 If the equation of a velocity distribution over a plate is given by $v = 2y - y^2$, in which v is the velocity in m/s at a distance y, measured in metres above the plate, what is the velocity gradient at the boundary and at 7.5 cm and 15 cm from it? Also determine the shear stress at these points if absolute viscosity μ = 8.60 poise.

[*Ans.* 2 sec^{-1}; 1.85 sec^{-1}; 1.70 sec^{-1}; 0.175 kg(f)/m^2; 0.162 kg(f)/m^2; 0.149 kg(f)m^2]

1.8 A body weighing 441.45 N with a flat surface area of 0.093 m^2 slides down lubricated inclined plane making a 30° angle with the horizontal. For viscosity of 0.1 N.s/m^2 and body speed of 3m/s, determine the lubricant film thickness.

[*Ans.* 0.126 mm]

1.9 A hydraulic lift consists of a 25 cm diameter ram which slides in a 25.015 cm diameter cylinder, the annular space being filled with oil having a kinematic viscosity of 0.025 cm^2/sec and specific gravity of 0.85. If the rate of travel of the ram is 9.15 m/min, find the frictional resistance when 3.05 m of the ram is engaged in the cylinder. [*Ans.* 1.055 kg(f)]

1.10 A cylinder 0.1 m diameter rotates in an annular sleeve 0.102 m internal diameter at 100 r.p.m. The cylinder is 0.2 m long. If the dynamic viscosity of the lubricant between the two cylinders is 1.0 poise, find the torque needed to drive the cylinder against viscous resistance. Assume that Newton's Law of viscosity is applicable and the velocity profile is linear.

[*Ans.* 0.1645 N.m]

1.11 A fluid compressed in a cylinder has a volume of 0.011 32 m^3 at a pressure of 70.30 kg(f)/cm^2. What should be the new pressure in order to make its volume 0.011 21 m^3 ? Assume bulk modulus of elasticity K of the liquid as 703 0 kg(f)/cm^2. [*Ans.* 138.61 kg(f)/cm^2]

1.12 If the volume of a liquid decreases by 0.2 per cent for an increase of pressure from 6.867 MN/m^2 to 15.696 MN/m^2, what is the value of the bulk modulus of the liquid ?

[*Ans.* 44.145 × 108 N/m^2]

1.13 Obtain an expression for the bulk modulus of elasticity K of a fluid in terms of the pressure P, and the mass density ρ. [*Ans.* K = ρ($dp/d\rho$)]

1.14 A soap bubble 51 mm in diameter has an internal pressure in excess of outside pressure of 0.000 21 kg(f)/cm^2. Calculate the tension in the soap film. [**Hint.** ρ = (4σ/r)]

[*Ans.* 0.0134 kg(f)/m]

1.15. If the pressure inside a droplet of water is 196.2 N/m^2 in excess of the external pressure, what is the diameter of the droplet? Given the value of surface tension of water in contact with air at 20°C as 0.073 58 N/m. [*Ans.* 1.5 mm]

1.16 A glass tube 0.25 mm in diameter contains mercury column with air above the mercury at 20°C. The surface tension of mercury in contact with air is 0.051 kg(f)/m. What will be the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

capillary depression of the mercury if angle of contact θ = 130° and sp. gr. of mercury = 13.6. [*Ans.* 3.86 cm]

1.17 A glass tube 0.25 mm in diameter contains a mercury column with water above the mercury. The temperature is 20°C at which the surface tension of mercury in contact with water is 0.037 kg(f)/m. What will be the capillary depression of the mercury? Take angle of contact θ = 130°. [**Hint**. Use Eq. 1.9] [*Ans.* 3.02 cm]

1.18 Calculate the capillary rise h in a glass tube of 3 mm diameter when immersed in water at 20°C. Take s for water at 20°C as 0.007 5 kg(f)/m. What will be the percentage increase in the value of h if the diameter of the glass tube is 2 mm? [*Ans.* 10 mm; 50 per cent]

1.19 Show that for two vertical parallel plates t distance apart, held partially immersed in a liquid of surface tension σ and specific weight w, the capillary rise h is given by the expression

$$h = \frac{2\sigma\cos\theta}{wt}$$

in which θ is the angle of contact.

Hence calculate the maximum capillary rise of water at 20°C to be expected between two vertical clean glass plates spaced 1 mm apart. Take σ = 0.073 58 N/m. [*Ans.* 15 mm]

1.20 A capillary tube having inside diameter 5 mm is dipped in water at 20°C. Determine the height of water which will rise in the tube. Take σ = 0.075 gm(f)/cm and θ = 60°. Specific weight of water at 20°C = 998 kg(f)/m^3. [*Ans.* 3 mm]

1.21 By how much does the pressure in a cylindrical jet of water 4 mm in diameter exceed the pressure of the surrounding atmosphere if σ = 0.0075 kg(f)/m.

[**Hint.** Use Eq. 1.6 b]

[*Ans.* 3.75 kg(f)m^2]

1.22 Calculate the capillary effect in mm in a glass tube of 4 mm diameter when immersed in (i) water, and (ii) mercury, both at 20°C. The values of s of water and mercury at 20°C in contact with air are respectively 0.007 5 kg(f)/m and 0.052 kg(f)/m. The contact angle for water θ = 0° and for mercury θ = 130°.

[*Ans.* (i) 7.5 mm; (ii) – 2.46 mm]

1.23 Name the characteristic fluid properties to which the following phenomena are attributable: (i) rise of sap in a tree; (ii) spherical shape of a drop of a liquid; (iii) cavitation; (iv) flow of a jet of oil in an unbroken stream; (v) water hammer.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Fluid Pressure and its Measurement

Chapter 2

2.1 FLUID PRESSURE AT A POINT

'Pressure' or 'intensity of pressure' may be defined as the force exerted on a unit area. If F represents the total force uniformly distributed over an area A, the pressure at any point is $p = (F/A)$. However, if the force is not uniformly distributed, the expression will give the average value only. When the pressure varies from point to point on an area, the magnitude of pressure at any point can be obtained by the following expression

$$p = \frac{dF}{dA}$$

where dF represents the force acting on an infinitesimal area dA. In SI units pressure is expressed in N/m^2 (or pascal), and in metric gravitational units it is expressed in $kg(f)/cm^2$ or $kg(f)/m^2$.

A fluid is a substance which is capable of flowing. As such when a certain mass of fluid is held in static equilibrium by confining within solid boundaries, it exerts forces against boundary surfaces. The forces so exerted always act in the direction normal to the surface in contact. This is so because a fluid at rest cannot sustain shear stress and hence the forces cannot have tangential components. The normal force exerted by a fluid per unit area of the surface is called the fluid pressure. It may, however, be noted that even if an imaginary surface is assumed within a fluid body, the fluid pressure and pressure force on the imaginary surface are exactly the same as those acting on any real surface. This is in accordance with Newton's third law of motion, viz., action and reaction exist in pairs.

2.2 VARIATION OF PRESSURE IN A FLUID

Consider a small fluid element of size $\delta x \times \delta y \times \delta z$ at any point in a static mass of fluid as shown in Fig. 2.1. Since the fluid is at rest, the element is in equilibrium under the various forces acting on it. The forces acting on the element are the pressure forces on its faces and the self-weight of the element. Let p be the pressure intensity at the mid point O of the element. Then the pressure intensity on the left hand face of the element is $\left[p - \left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]$ and the pressure intensity on the right hand face of the element is $\left[p + \left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]$. The corresponding pressure forces on the left hand and the right hand

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

faces of the element are $\left[p-\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right].\ \delta y\delta z$ and $\left[p+\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]\delta y\delta z$ respectively. Likewise the pressure intensities and the corresponding pressure forces on the other faces of the element may be obtained as shown in Fig. 2.1. Further if w is the specific weight of the fluid then the weight of the element acting vertically downwards is ($w\delta x\ \delta y\ \delta z$). Since the element is in equilibrium under these forces, the algebraic sum of the forces acting on it in any direction must be zero. Thus considering the forces acting on the element along x,y and z axes the following Equations are obtained

$$\Sigma F_x = 0$$

or $$\left(p-\frac{\partial p}{\partial x}\frac{\delta x}{2}\right).\ \delta y\ \delta z-\left(p+\frac{\partial p}{\partial x}\frac{\delta x}{2}\right).\ \delta y\delta z=0$$

or $$\frac{\partial p}{\partial x} = 0 \qquad ...(2.1)$$

$$\Sigma F_y = 0$$

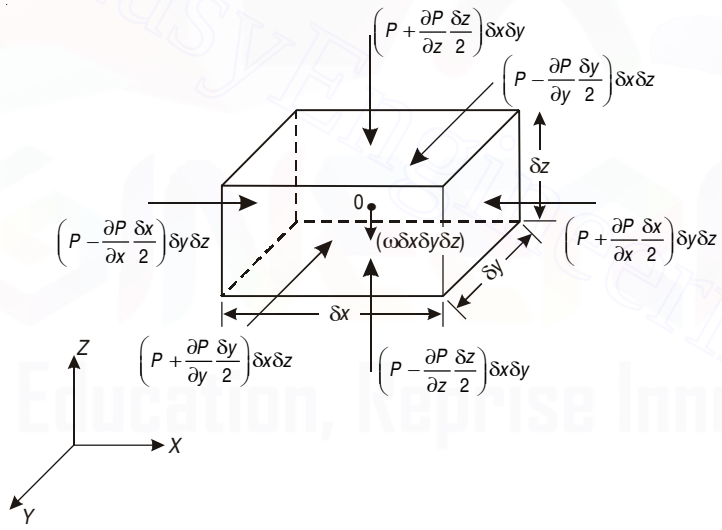


Figure 2.1 *Fluid element with forces acting on it in a static mass of fluid*

or $$\left(p-\frac{\partial p}{\partial y}\frac{\delta y}{2}\right).\ \delta x\ \delta z-\left(p+\frac{\partial p}{\partial y}\frac{\delta y}{2}\right).\ \delta x\ \delta z=0$$

or $$\frac{\partial p}{\partial y} = 0 \qquad ...(2.2)$$

$$\Sigma F_z = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\left(p - \frac{\partial p}{\partial z}\frac{\delta z}{2}\right).\ \delta x\ \delta y - \left(p + \frac{\partial p}{\partial z}\frac{\delta z}{2}\right).\ \delta x\ \delta y - (w\ \delta x\ \delta y\ \delta z) = 0$$

or $$\frac{\partial p}{\partial z} = -w \qquad ...(2.3)$$

Equations 2.1, 2.2 and 2.3 indicate that the pressure intensity p at any point in a static mass of fluid does not vary in x and y directions and it varies only in z direction. Hence the partial derivative in eq. 2.3 may be reduced to total (or exact) derivative as follows.

$$\frac{dp}{dz} = -w = -\rho g \qquad ...(2.4)$$

In vector notation Eq. 2.4 may be expressed as

$$-\text{grad}\ p = wk = \rho g k$$

where k is unit vector parallel to z axis.

The minus sign (–) in the above equation signifies that the pressure decreases in the direction in which z increases i.e., in the upward direction.

Equation 2.4 is the basic differential equation representing the variation of pressure in a fluid at rest, which holds for both compressible and incompressible fluids. Equation 2.4 indicates that within a body of fluid at rest the pressure increases in the downward direction at the rate equivalent to the specific weight w of the liquid. Further if $dz = 0$, then dp is also equal to zero; which means that the pressure remains constant over any horizontal plane in a fluid. Integration of Eq. 2.4 yields the pressure at any point in a fluid at rest, which is discussed separately for the incompressible fluids such as liquids having constant density and compressible fluids such as gases having variable density.

Pressure at a Point in a Liquid

A liquid may be considered as incompressible fluid for which w is constant and hence integration of Eq. 2.4 gives

$$p = -wz + C \qquad ...(2.5)$$

in which p is the pressure at any point at an elevation z in the static mass of liquid and C is the constant of integration. Liquids have a free surface at which the pressure of atmosphere acts. Thus as shown in Fig. 2.2 for a point lying in the free surface of the liquid $z = (H + z_0)$ and if p_a is the atmospheric pressure at the liquid surface then from Eq. 2.5 the constant of integration $C = [p_a + w\ (H + z_0)]$. Substituting this value of C in Eq. 2.5, it becomes

$$P = -\ wz + [p_a + w\ (H + z_0)] \qquad ...(2.6)$$

Now if a point is lying in the liquid mass at a vertical depth h below the free surface of the liquid then as shown in Fig. 2.2 for this point $z = (H + z_0 - h)$ and from Eq. 2.6

$$p = p_a + wh \qquad ...(2.7)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

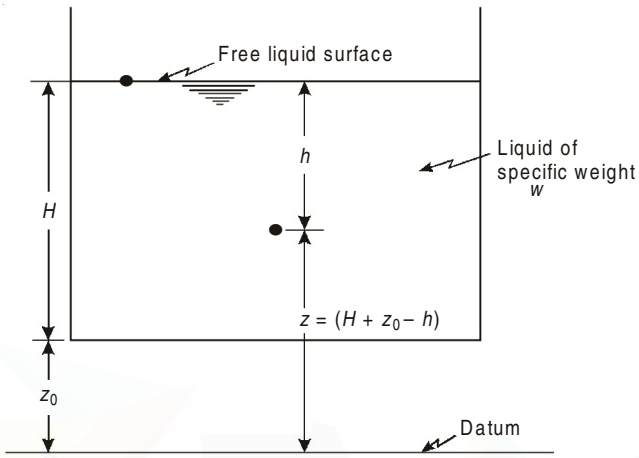


Figure 2.2 *Pressure at a point in a liquid*

It is evident from Eq. 2.7 that the pressure at any point in a static mass of liquid depends only upon the vertical depth of the point below the free surface and the specific weight of the liquid, and it does not depend upon the shape and size of the bounding container. This fact is illustrated in Fig. 2.3 in which although the containers of different shapes are interconnected, so that the total weight

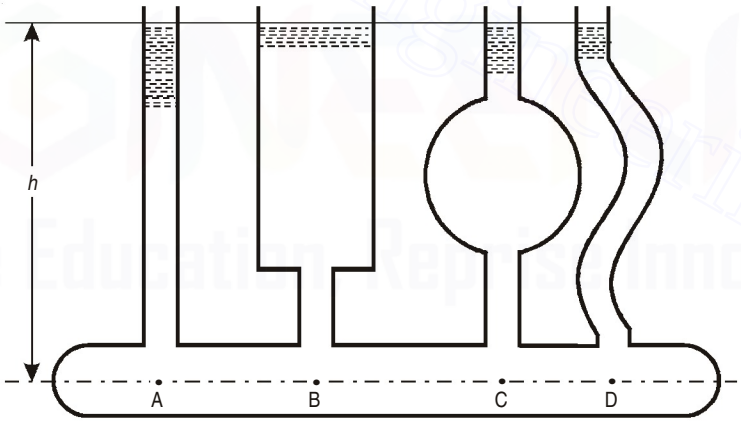


Figure 2.3 *Interconnected containers of different shapes*

of the liquid in each part differs, yet the pressures at points A, B, C and D lying on the same horizontal level and at the same vertical depth h below the free surface of the liquid will be same.

Since the atmospheric pressure at a place is constant, at any point in a static mass of liquid, often only the pressure in excess of the atmospheric pressure is considered, in which case Eq. 2.7 becomes

$$p = wh \qquad ...(2.8)$$

Pressure Head. The vertical height of the free surface above any point in a liquid at rest is known as *pressure head.* From Eq. 2.8 the pressure head may be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h = \frac{p}{w} \qquad ...(2.9)$$

Since the pressure at any point in a liquid depends on the height of the free surface above the point, it is convenient to express a fluid pressure in terms of pressure head. The pressure is then expressed in terms of metres (or centimetres) of a liquid column.

Equation 2.8 can be used to obtain a relationship between the heights of columns of different liquids which would develop the same pressure at any point. Thus, if h_1 and h_2 are the heights of the columns of liquids of specific weights w_1 and w_2 required to develop the same pressures p, at any point; then from Eq. 2.8

$$p = w_1 h_1 = w_2 h_2 \qquad ...(2.10)$$

If S_1 and S_2 are the specific gravities of the two liquids and w is the specific weight of water then since $w_1 = S_1 w$ and $w_2 = S_2 w$, Eq. 2.10 may also be written as

$$S_1 h_1 = S_2 h_2 \qquad ...(2.10\ a)$$

Further if h_1 and h_2 are the depths of two points below the free surface in a static mass of liquid of specific weight w and p_1 and p_2 are the respective pressure intensities at these points, then from Eq. 2.8 the pressure difference between these points is obtained as

$$(p_1 - p_2) = w\,(h_1 - h_1)$$

Thus it may be stated that the difference in pressure at any two points in a static mass of liquid varies directly as the difference in depth (or elevation) of the two points.

Pressure at any Point in a Compressible Fluid. For a compressible fluid since the density varies with the pressure, Eq. 2.4 can be integrated only if the relation between w and p is known. Moreover, in a compressible fluid there being no free surface, the integration of Eq. 2.4 gives the variation of pressure between any two points lying in a static mass of fluid. Thus if p_1 and p_2 are the pressure intensities at two points which are at elevations z_1 and z_2 above an arbitrarily assumed datum, then integration of Eq. 2.4 yields

$$\int_{p_1}^{p_2} \frac{dp}{w} = -\int_{z_1}^{z_2} dz = (z_1 - z_2)$$

The left hand side of the above expression has been evaluated using different relations between w and p as indicated in the Section 2.3.

2.3 EQUILIBRIUM OF A COMPRESSIBLE FLUID—ATMOSPHERIC EQUILIBRIUM

Equation 2.4 expresses the condition for equilibrium of any fluid, which may be written as

$$dp = -wdz = -\rho g dz \qquad ...(2.4)$$

This is a general relationship which can be applied to both incompressible as well as compressible fluids. However, for a compressible fluid the above equation can be integrated, to obtain the value of pressure at any point in the fluid, provided it can be assumed that the mass density ρ is either constant or it is a function of pressure (or elevation) only. The mass density ρ may be assumed to be constant if the variation in elevation is not great. But if the variation in elevation is great, ρ can no longer be considered constant, in which case the manner in which ρ varies with pressure p (or elevation) must be known. The variation of pressure and mass density of the air in the atmosphere

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

with elevation is one such problem which is encountered in aeronautics and meteorology. Similarly in oceanography too the problem of the variation of pressure and mass density with depth is of concern, since at great depth there is a small increase in the density of the fluid.

In order to obtain the variation of pressure with elevation in the atmosphere, some of the relations between p and ρ which may be used in Eq. 2.4 are as described below.

(a) Isothermal State for Atmosphere or Isothermal Atmosphere. If it is assumed that the variation of p and ρ is according to isothermal condition, i.e., the temperature of atmosphere is assumed to have a constant value (which may however be true only over a relatively small vertical distance), then according to Boyle's law

$$\frac{p}{\rho} = \frac{p_0}{\rho_0} \qquad \text{... (2.11)}$$

where p_0 and ρ_0 are the values of the pressure and density of the gas at initial condition at some reference level, for example at the earth's surface i.e., at $z = 0$.

From Eq. 2.4

$$\frac{dp}{\rho} = -g dz$$

By substituting for ρ from Eq. 2.11

$$\frac{dp}{p}\frac{p_0}{\rho_0} = -g dz$$

Since the value of g decreases by only about 0.1% for about 300 m increase in altitude it may be assumed constant. Thus integration of above equation gives

$$-gz = \frac{p_0}{\rho_0}\log_e p + C$$

If p_1 is the pressure at height z_1, then

$$-gz_1 = \frac{p_0}{\rho_0}\log_e p_1 + C$$

Thus eliminating C from these equations

$$(z - z_1) = \frac{p_0}{\rho_0 g}\log_e\left(\frac{p_1}{p}\right) \qquad \text{...(2.12)}$$

Equation 2.12 expresses the relation between altitude and pressure when the air is isothermal. The ratio $(p_0/\rho_0 g)$ represents the height of a fluid column of constant specific weight $(\rho_0 g)$. It is called the *equivalent height of a uniform atmosphere.*

Further at $z = 0$ since $p = p_0$ the integration constant

$$C = -\frac{p_0}{\rho_0}\log_e p_0$$

Hence

$$z = -\frac{p_0}{\rho_0 g}\log_e\left(\frac{p}{p_0}\right) \qquad \text{...(2.13)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For a perfect gas from equation of state

$$p_0 = \rho_0 R T_0 \; ; \text{ or } \frac{p_0}{\rho_0} = R T_0$$

Thus substituting the values of (p_0/ρ_0) in Eq. (2.13)

$$z = -\frac{RT_0}{g} \log_e \left(\frac{p}{p_0} \right)$$

or
$$\frac{p}{p_0} = \exp\left(-\frac{gz}{RT_0} \right) \qquad ...(2.14)$$

From Eq. 2.14 it may be seen that when (gz/RT_0) is small this expression approximates to that for a fluid of constant density.

(b) Adiabatic State for Atmosphere or Adiabatic (or Isentropic) Atmosphere. It is well-known fact that the temperature in the atmosphere decreases rapidly as one goes to higher altitude. Thus if it is assumed that no heat is added or taken away from a certain column of air, then this column is said to be under adiabatic conditions.

For an adiabatic process

$$\frac{p}{\rho^k} = \frac{p_0}{\rho_0^k} \; ; \text{ or } \rho = \rho_0 \left(\frac{p}{p_0} \right)^{1/k} \qquad ...(2.15)$$

in which k is the adiabatic exponent (or adiabatic index) defined as the ratio of the specific heat at constant pressure C_p to the specific heat at constant volume C_v

Substituting this value in Eq. 2.4 one obtains

$$-g dz = \frac{dp}{\rho_0} \left(\frac{p_0}{p} \right)^{1/k}$$

or
$$-dz = \frac{1}{\rho_0 g} \left(\frac{p_0}{p} \right)^{1/k} dp$$

or
$$-dz = \frac{p_0}{\rho_0 g} \left(\frac{p}{p_0} \right)^{-\frac{1}{k}} d\left(\frac{p}{p_0} \right)$$

As stated earlier g may be assumed to be constant and hence by integrating the above expression one obtains

$$-z = \frac{p_0}{\rho_0 g} \left(\frac{p}{p_0} \right)^{1-\frac{1}{k}} \left(\frac{1}{1-\frac{1}{k}} \right) + C$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

if $p = p_0$ when $z = z_0$ then integration constant

$$C = -z_0 - \left(\frac{k}{k-1}\right)\frac{p_0}{\rho_0 g}$$

Hence $$(z - z_0) = \left(\frac{k}{k-1}\right)\frac{p_0}{\rho_0 g}\left[1-\left(\frac{p}{p_0}\right)^{\frac{k-1}{k}}\right] \quad ...(2.16)$$

from which

$$\frac{p}{p_0} = \left[1 - \Delta z \frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right]^{\frac{k}{k-1}} \quad ...(2.17)$$

where $(z - z_0) = \Delta z$

Substituting the value of (p/p_0) in Eq. (2.15)

$$\frac{\rho}{\rho_0} = \left[1 - \Delta z \frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right]^{\frac{1}{k-1}} \quad ...(2.18)$$

Again for a perfect gas from equation of state

$$p_0 = \rho_0 RT_0 ; \text{ or } \frac{p_0}{\rho_0} = RT_0$$

Substituting the value of (p_0/ρ_0) in Eq. (2.17)

$$\frac{p}{p_0} = \left[1 - \Delta z \frac{g}{RT_0}\left(\frac{k-1}{k}\right)\right]^{\frac{k}{k-1}} \quad ...(2.19)$$

If z is small, all but the first two terms of the binomial expression of the right hand side of Eq. 2.17 may be neglected and then

$$\frac{p}{p_0} = 1 - \frac{\rho_0 g \Delta z}{p_0}$$

or $$p = p_0 - \rho_0 g \, \Delta z$$

This corresponds to the relationship $p + wz$ = constant (eq. 2.5) for fluid of constant density. Thus for small differences of height say less than about 300 m in the atmosphere, the fluid may be considered to be of constant density without an appreciable error being introduced.

Further dividing Eq. 2.17 by Eq. 2.18 one obtains

$$\frac{p}{\rho} = \frac{p_0}{\rho_0}\left[1 - \Delta z \frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$RT = \frac{p_0}{\rho_0}\left[1 - \Delta z \frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right] \quad ...(2.20)$$

since $\frac{p}{\rho} = RT$ from the eq. of state.

Equation 2.20 gives the relation between the absolute temperature T and altitude z. Now if $T = T_0$ at $z = z_0$ (or $\Delta z = 0$) then from Eq. 2.20

$$RT_0 = \frac{p_0}{\rho_0}$$

Further if $(T_0 - T) = \Delta T$ for a difference in elevation Δz then

$$R(\Delta T) = g(\Delta z)\left(\frac{k-1}{k}\right) \quad ...(2.21)$$

For air, R = 29.27 m-kg(f)/kg(m)°C absolute and k =1.405, then for Δz = 100 m, the variation in temperature ΔT may be obtained from Eq. 2.21 as

$$R(\Delta T) = \frac{9.81 \times 100 \times 0.405}{1.405}$$

$$R = 29.27 \text{ m-kg (f)/kg (m)°C absolute}$$

$$= 29.27 \times 9.81 \text{ m}^2/\text{sec}^2 \text{ °C absolute}$$

$$\therefore \quad \Delta T = \frac{9.81 \times 100 \times 0.405}{29.27 \times 9.81 \times 1.405} = 0.985 \text{ °C.}$$

This shows that under adiabatic condition, the absolute temperature decreases by about 0.985°C for each 100 m increase in elevation. However, if the atmosphere is stable, the temperature drop with altitude is generally somewhat less than that computed above.

(c) Polytropic State for Atmosphere or Polytropic Atmosphere. In a more generalized way the variation of atmospheric pressure p with ρ may be considered according to a polytropic process, in which case, we have

$$\frac{p}{\rho^n} = \frac{p_0}{\rho_0^n}; \text{ or } \rho = \rho_0 \left(\frac{p}{p_0}\right)^{1/n}$$

where n is a positive constant.

Again substituting this value in Eq. 2.4 and integrating as indicated earlier the following expression may be obtained

$$\frac{p}{p_0} = \left[1 - \frac{g(z - z_0)}{RT_0}\frac{n-1}{n}\right]^{\frac{n}{n-1}} \quad ...(2.22)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Evidently Eq. 2.22 is more general being applicable for any value of n, except for the particular case of $n = 1.0$; and from this, eq. 2.19 can be obtained by considering $n = k$. Equation 2.22 represents the variation of pressure with elevation, which has been plotted in Fig. 2.4 for different values of n. The actual variation of pressure with elevation in the atmosphere is also plotted in Fig 2.4, from which it may be seen that in the atmosphere the actual value of n usually varies between $n = 1.2$ (wet adiabatic process) to $n = 1.4$ (dry adiabatic process). As shown in Fig. 2.4 the value of n depends on the *temperature lapse rate* ($\partial T/\partial z$). Combining Eq. 2.22 with the equation of state $p = \rho RT$ and the equation (p/ρ^n) = constant, we have

$$\frac{T}{T_0} = \left[1 - \frac{g(z - z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right] \qquad ...(2.23)$$

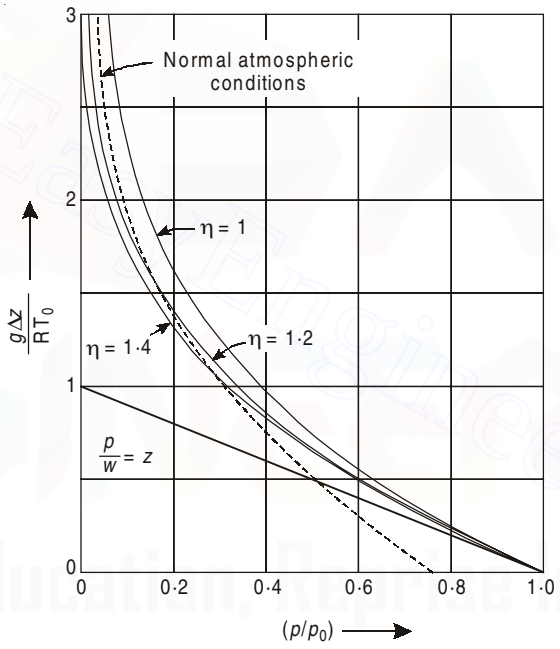


Figure 2.4 *Variation of pressure with elevation in the atmosphere*

Differentiating with respect to z, we obtain

$$\frac{1}{T_0}\frac{\partial T}{\partial z} = -\frac{g}{RT_0}\left(\frac{n-1}{n}\right)$$

or $$\frac{g}{R}\left(\frac{n-1}{n}\right) = -\frac{\partial T}{\partial z} = \lambda$$

where $\lambda = -\left(\frac{\partial T}{\partial z}\right)$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad n = \frac{1}{1-(R\lambda/g)} \qquad ...(2.24)$$

For the first 11 000 m above the ground the temperature in the atmosphere decreases uniformly, that is $-(\partial T/\partial z) = \lambda$ = constant. The observed value of λ in this region of atmosphere is about 6.56°C/1000 m

$$\therefore \quad \frac{R\lambda}{g} = \frac{29.27 \text{ m kg(f)}}{\text{kg(m)°C}} \times \frac{6.56°\text{C}}{1000 \text{ m}} \times \frac{1}{9.81} \frac{\text{sec}^2}{\text{m}}$$

$$= \frac{29.27 \times 0.00656}{9.81} \frac{\text{kg(f)-sec}^2}{\text{kg(m) m}}$$

Since 1 kg(f) = 9.81 kg (m) $\frac{\text{m}}{\text{sec}^2}$

$$\therefore \quad \frac{R\lambda}{g} = 29.27 \times 0.006\,56 = 0.192$$

and $$n = \frac{1}{1-0.192} = 1.238.$$

From about 11 000 m to 32 000 m the temperature in the atmosphere remains constant at about –56.5°C in which case $n = 1$, that is, isothermal condition may be assumed and then Eq. 2.14 applies. Beyond 32 000 m the temperature rises again. Since n depends on the temperature lapse rate λ, which varies with the altitude, it is obvious that in the atmosphere n varies with altitude. As such the assumption of a constant value of n for all the altitudes in the atmosphere may lead to erroneous results.

(d) Standard Atmosphere. It is very well known that the densities and the temperatures in the atmosphere vary continuously. They change from day to day and from place to place on the earth. In addition, moisture too plays a part in this variation, though it has a relatively small influence on density ρ and hence it is generally neglected in practical calculations. For aeronautical purpose, particularly for comparison of aircraft performance at different locations and on different days the difficulties that may arise from variations in ρ must be avoided. Therefore, an international *standard atmosphere* has been chosen. The standard atmosphere is defined by certain values of n, p_0 and T_0 which provides a set of data reasonably representative of the actual atmosphere. The standard atmosphere as approved by ICAO (International Civil Aviation Organisation) is based on the following values at sea level : p_o = 10 332 kg(f)/m^2 ; t_0 = 15°C ; w_0 = 1.226 kg(f)/m^3 ; ρ_0 = 0.125 msl/m^3 ; R = 29.27 m.kg(f)/kg(m) °C. In the lower atmosphere or *troposphere* the temperature of the air decreases linearly with altitude at an average rate of λ = 6.56° C/1000 m, which continues upto an altitude of about 11 000 m. For a considerable distance above this elevation the temperature of air remains constant at about –56.5°C. This upper region where constant temperature prevails is usually referred to as the *stratosphere,* which extends from about 11000 m to 32000 m. However, there is no sharp demarcation between these two strata, but the transition from one to the other is rather gradual. This zone of transition is called the *tropopause*. As discussed earlier, in the troposhpere the functional relation between p and ρ may be assumed to be polytropic with n = 1.238 while in the stratosphere on account of the temperature being constant the functional relation between p and ρ is isothermal.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

2.4 PRESSURE, SAME IN ALL DIRECTIONS — PASCAL'S LAW

The pressure at any point in a fluid at rest has the same magnitude in all directions. In other words, when a certain pressure is applied at any point in a fluid at rest, the pressure is equally transmitted in all the directions and to every other point in the fluid. This fact was established by B. Pascal, a French Mathematician in 1653, and accordingly it is known as Pascal's Law.

To prove this statement, consider an infinitesimal wedge shaped element of fluid at rest as a free body. The element is arbitrarily chosen and has the dimensions as shown in Fig. 2.5. Since in a fluid at rest there can be no shear forces, the only forces acting on the free-body are the normal pressure

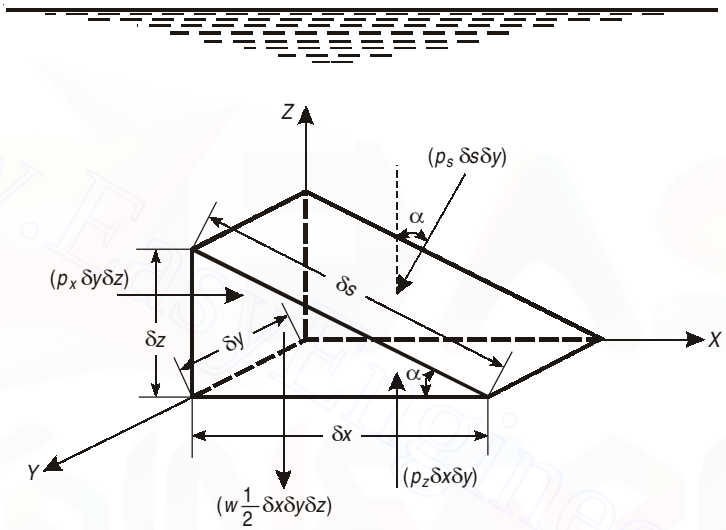


Figure 2.5 *Free-body diagram of a wedge-shaped element of fluid*

forces exerted by the surrounding fluid on the plane surfaces, and the weight of the element. As the element is in equilibrium, the sum of the force components on the element in any direction must be equal to zero. So, the equations of equilibrium in the x and z directions are respectively,

$$p_x\ \delta y\ \delta z - p_s\ \delta s\ \delta y\ \sin\alpha = 0$$

$$p_z\ \delta x\ \delta y - p_s\ \delta s\ \delta y\ \cos\alpha - w\frac{1}{2}\ \delta x\ \delta y\ \delta z\ = 0$$

in which p_x, p_z, p_s are the average pressures on the three faces and w is the specific weight of the fluid. Since $\delta z = \delta s \sin\alpha$ and $\delta x = \delta s \cos\alpha$ the above equations simplify to

$$p_x\ \delta y\ \delta z\ -\ p_s\ \delta y\ \delta z\ = 0$$

$$p_z\ \delta x\ \delta y - p_s\ \delta x\ \delta y - \frac{1}{2}\ w\ \delta x\ \delta y\ \delta z\ = 0$$

The third term of the second equation is much smaller than the other two terms (since it involves a product of three infinitesimal quantities), and hence it may be neglected. Now by dividing the equations by $\delta y \delta z$ and $\delta x \delta y$ respectively and taking the limit, so that the element is reduced to a point, it follows from the equations that $p_s = p_x = p_z$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since angle α is chosen arbitrarily, this equation proves that the pressure is the same in all directions at a point in a static fluid.

The Pascal's law is made use of in the construction of machines such as hydraulic press, hydraulic jack, hydraulic lift, hydraulic crane, hydraulic rivetter etc., in which by the application of relatively small forces considerably larger forces are developed.

2.5 ATMOSPHERIC, ABSOLUTE, GAGE AND VACUUM PRESSURES

The atmospheric air exerts a normal pressure upon all surfaces with which it is in contact, and it is known as *atmospheric pressure.* The atmospheric pressure varies with the altitude and it can be measured by means of a barometer. As such it is also called the barometric pressure. At sea level under normal conditions the equivalent values of the atmospheric pressure are 10.1043×10^4 N/m^2 or 1.03 kg(f)/cm^2 ; or 10.3 m of water ; or 76 cm of mercury.

Fluid pressure may be measured with respect to any arbitrary datum. The two most common datums used are (i) absolute zero pressure and (ii) local atmospheric pressure. When pressure is measured above absolute zero (or complete vacuum), it is called an *absolute pressure.* When it is measured either above or below atmospheric pressure as a datum, it is called *gage pressure.* This is because practically all pressure gages read zero when open to the atmosphere and read only the difference between the pressure of the fluid to which they are connected and the atmospheric pressure.

If the pressure of a fluid is below atmospheric pressure it is designated as vacuum pressure (or suction pressure on negative gage pressure) ; and its gage value is the amount by which it is below that of the atmospheric pressure. A gage which measures vacuum pressure is known as vacuum gage.

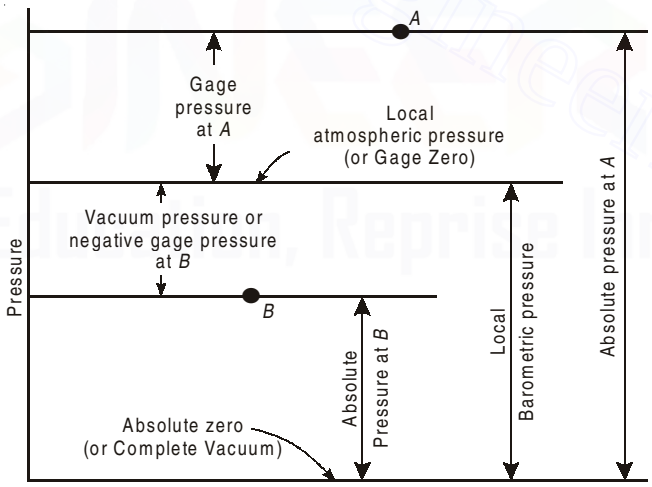


Figure 2.6 *Relationship between absolute, gage and vacuum pressures*

All values of absolute pressure are positive, since in the case of fluids the lowest absolute pressure which can possibly exist corresponds to absolute zero or complete vacuum. However, gage pressures are positive if they are above that of the atmosphere and negative if they are vacuum pressures. Figure 2.6 illustrates the relation between absolute, gage and vacuum pressures.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From the foregoing discussion it can be seen that the following relations hold :

Absolute Pressure = Atmospheric Pressure + Gage Pressure ...(2.25)

Absolute Pressure = Atmospheric Pressure – Vacuum Pressure ...(2.26)

2.6 MESUREMENT OF PRESSURE

The various devices adopted for measuring fluid pressure may be broadly classified under the following two heads:

(1) Manometers

(2) Mechanical Gages.

Manometers. Manometers are those pressure measuring devices which are based on the principle of balancing the column of liquid (whose pressure is to be found) by the same or another column of liquid. The manometers may be classified as

(a) Simple Manometers.

(b) Differential Manometers.

Simple Manometers are those which measure pressure at a point in a fluid contained in a pipe or a vessel. On the other hand *Differential Manometers* measure the difference of pressure between any two points in a fluid contained in a pipe or a vessel.

Simple Manometers. In general a simple manometer consists of a glass tube having one of its ends connected to the gage point where the pressure is to be measured and the other remains open to atmosphere. Some of the common types of simple manometers are as noted below:

(i) Piezometer.

(ii) U-tube Manometer.

(iii) Single Column Manometer.

(i) *Piezometer.* A piezometer is the simplest form of manometer which can be used for measuring moderate pressures of liquids. It consists of a glass tube (Fig. 2.7) inserted in the wall of a pipe or a vessel, containing a liquid whose pressure is to be measured. The tube extends vertically upward to such a height that liquid can freely rise in it without overflowing. The pressure at any point in the liquid is indicated by the height of the liquid in the tube above that point, which can be read on the scale attached to it. Thus, if w is the specific weight of the liquid, then the pressure at point m in Fig. 2.7 (a) is $p_m = wh_m$. In other words, h_m is the pressure head at m. Piezometers measure gage pressure only, since the surface of the liquid in the tube is subjected to atmospheric pressure.

From the foregoing principles of pressure in homogeneous liquid at rest, it is obvious that the location of the point of insertion of a piezometer makes no difference. Hence as shown in Fig. 2.7 (a) piezometers may be inserted either in the top, or the side, or the bottom of the container, but the liquid will rise to the same level in the three tubes.

Negative gage pressures (or pressures less than atmospheric) can be measured by means of the piezometer shown in Fig. 2.7 (b). It is evident that if the pressure in the container is less than the atmospheric no column of liquid will rise in the ordinary piezometer. But if the top of the tube is bent downward and its lower end dipped into a vessel containing water (or some other suitable liquid) [Fig. 2.7 (b)], the atmospheric pressure will cause a column of the liquid to rise to a height h in the tube, from which the magnitude of the pressure of the liquid in the container can be obtained.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

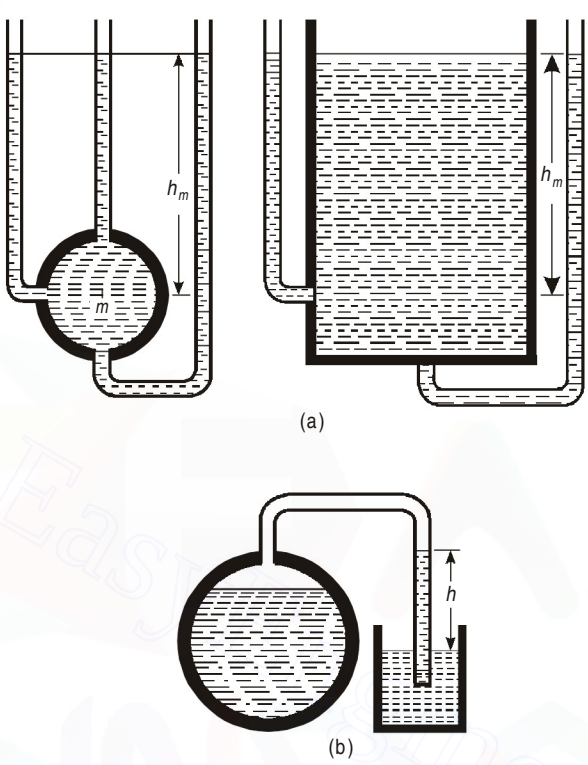


Figure 2.7 *Piezometers*

Neglecting the weight of the air caught in the portion of the tube, the pressure on the free surface in the container is the same as that at free surface in the tube which from Eq. 2.8 may be expressed as $p = -wh$, where w is the specific weight of the liquid used in the vessel. Conversely $-h$ is the pressure head at the free surface in the container.

Piezometers are also used to measure pressure heads in pipes where the liquid is in motion. Such tubes should enter the pipe in a direction at right angles to the direction of flow and the connecting end should be flush with the inner surface of the pipe. All burrs and surface roughness near the hole must be removed, and it is better to round the edge of the hole slightly. Also, the hole should be small, preferably not larger than 3 mm.

In order to prevent the capillary action from affecting the height of the column of liquid in a piezometer, the glass tube having an internal diameter less than 12 mm should not be used. Moreover for precise work at low heads the tubes having an internal diameter of 25 mm may be used.

(ii) *U-tube Manometer.* Piezometers cannot be used when large pressures in the lighter liquids are to be measured, since this would require very long tubes, which cannot be handled conveniently. Furthermore gas pressures cannot be measured by means of piezometers because a gas forms no free atmospheric surface. These limitations imposed on the use of piezometers may be overcome by the use of U-tube manometers. A U-tube manometer consists of a glass tube bent in U-shape, one

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

end of which is connected to the gage point and the other end remains open to the atmosphere (Fig. 2.8). The tube contains a liquid of specific gravity greater than that of the fluid of which the pressure is to be measured.

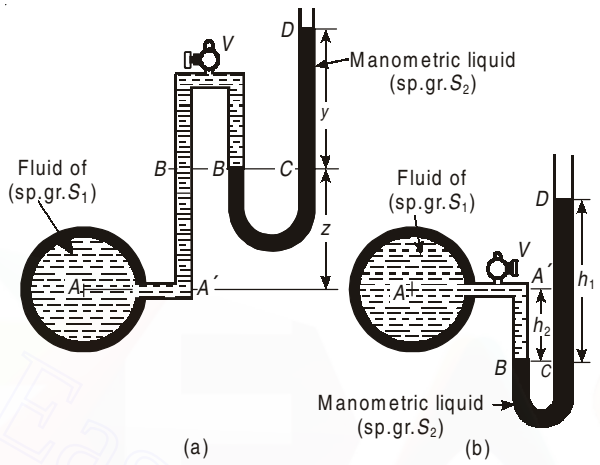


Figure 2.8 *U-tube simple manometer*

Sometimes more than one liquid may also be used in the manometer. The liquids used in the manometers should be such that they do not get mixed with the fluids of which the pressures are to be measured. Some of the liquids that are frequently used in the manometers are mercury, oil, salt solution, carbon disulphide, carbon tetrachloride, bromoform and alcohol. Water may also be used as a manometric liquid when the pressures of gases or certain coloured liquids (which are immiscible with water) are to be measured. The choice of the manometric liquid, however, depends on the range of pressure to be measured. For low pressure range, liquids of lower specific gravities are used and for high pressure range, generally mercury is employed.

When one of the limbs of the U-tube manometer is connected to the gage point, the fluid from the container or pipe *A* will enter the connected limb of the manometer thereby causing the manometric liquid to rise in the open limb as shown in Fig. 2.8. An air relief valve *V* is usually provided at the top of the connecting tube which permits the expulsion of all air from the portion *A′B* and its place taken by the fluid in *A*. This is essential because the presence of even a small air bubble in the portion *A′B* would result in an inaccurate pressure measurement.

In order to determine the pressure at *A*, a gage equation may be written as described below. Although any units of pressure or head may be used in the gage equation, it is generally convenient to express all the terms in metres of the fluid whose pressure is to be measured. The following general procedure may be adopted to obtain the gage equation :

(1) Start from either *A* or from the free surface in the open end of the manometer and write the pressure there in an appropriate unit (say metres of water or other fluid or N/m^2 or $kg(f)/cm^2$ or $kg(f)/m^2$). If the pressure is unknown (as at *A*) it may be expressed in terms of an appropriate symbol. On the other hand the pressure at the free surface in the open end (which is equal to atmospheric pressure) may be taken as zero. So that equation formed in each case will represent the gage pressure.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(2) To the pressure found above, add the change in pressure (in the same units) which will be caused while proceeding from one level to another adjacent level of contact of liquids of different specific gravities. Use positive sign if the next level of contact is lower than the first and negative if it is higher. The pressure heads in terms of the heights of columns of same liquid may be obtained by using Eq. 2.10 (a).

(3) Continue the process as in (2) until the other end of the gage is reached and equate the expression to the pressure at that point, known or unknown.

The expression will contain only one unknown viz., the pressure at A, which may thus be evaluated.

Thus for the manometer arrangement shown in Fig. 2.8 (a) the gage equation may be written as mentioned below.

Starting from A, if p_A is the unknown pressure intensity at A, w is the specific weight of water and S_1 is the specific gravity of the liquid in the container, then the pressure head at $A = \dfrac{p_A}{wS_1}$ (in terms of liquid at A). Since all points lying at the same horizontal level in the same continuous static mass of liquid have same pressure.

Pressure at A = Pressure at A'.

From A' to B' there being increase in elevation, pressure head decreases, so that pressure head at $B' = \left(\dfrac{p_A}{wS_1} - z\right)$. Again from the above enunciation,

Pressure at B' = Pressure at B = Pressure at C.

From C to D there being increase in elevation, pressure head decreases. If S_2 represents the specific gravity of the manometric liquid then from Eq. 2.10(a) the pressure head, in terms of liquid at A, equivalent to the column CD $(= y)$ of the manometric fluid $= y\left(\dfrac{S_2}{S_1}\right)$. Thus pressure head at $D = \left(\dfrac{p_A}{wS_1} - z - y\dfrac{S_2}{S_1}\right)$. But at D there being atmospheric pressure, the pressure head = 0, in terms of the gage pressure. As such equating the pressure heads at D, the gage equation becomes

$$\frac{p_A}{wS_1} - z - \frac{yS_2}{S_1} = 0$$

or
$$\frac{p_A}{wS_1} = z + \frac{yS_2}{S_1} \qquad ...(2.27)$$

Equation 2.27 represents the pressure heads in terms of the liquid at A. However, if the pressure heads are expressed in terms of water the following equation is obtained

$$\frac{p_A}{w} = zS_1 + yS_2 \qquad ...(2.28)$$

Evidently Eq. 2.28 may be obtained directly from Eq. 2.27 by multiplying its both the sides by S_1, the specific gravity of liquid at A.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If A contains a gas, its specific weight being quite small, the specific gravity S_1 of the gas (with respect to water) will be so small that zS_1 may be neglected. In which case the pressure head of the gas at A in terms of water is given by

$$\frac{p_A}{w} = yS_2 \qquad ...(2.29)$$

Figure 2.8 (b) shows another arrangement for measuring pressure at A by means of a U-tube manometer. By following the same procedure as indicated above the gage equation for this arrangement can also be written, which will be in terms of liquid at A as,

$$\frac{p_A}{wS_1} = h_1 \frac{S_2}{S_1} - h_2 \qquad ...(2.30)$$

and in terms of water as

$$\frac{p_A}{w} = h_1S_2 - h_2S_1 \qquad ...(2.31)$$

Again if A contains gas, the specific gravity S_1 of the gas (with respect to water) being so small that h_2S_1 may be neglected. In which case Eq. 2.31 becomes

$$\frac{p_A}{w} = h_1S_2 \qquad ...(2.32)$$

A U-tube manometer can also be used to measure negative or vacuum pressure. For measurement of small negative pressure, a U-tube manometer without any manometric liquid may be used, which is as shown in Fig. 2.9 (a). It is evident that for the pressure at A being negative (i.e., less than atmospheric pressure) the liquid surface in the open limb of the manometer will be below A. The pressure at A may be determined from the gage equation as obtained below.

At C the pressure head = 0, in terms of gage pressure. Further, pressure at C = pressure at B. From B to A', there being increase in elevation, the pressure head decreases, so that pressure head at $A' = 0 - h$ (in terms of liquid at A). Again, Pressure at A' = Pressure at A.

Now if p_A is the pressure intensity at A, w is the specific weight of water, and S_1 is the specific gravity of the liquid at A, then the pressure head at $A = \frac{p_A}{wS_1}$ (in terms of liquid at A). Thus in terms of liquid at A,

$$\frac{p_A}{wS_1} = -h \qquad ...(2.33)$$

and in terms of water

$$\frac{p_A}{w} = -S_1h \qquad ...(2.34)$$

For measuring negative pressures of larger magnitude a manometric liquid having higher specific gravity is employed, for which the arrangement as shown in Fig. 2.9 (b) or Fig. 2.9 (c) may be employed.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

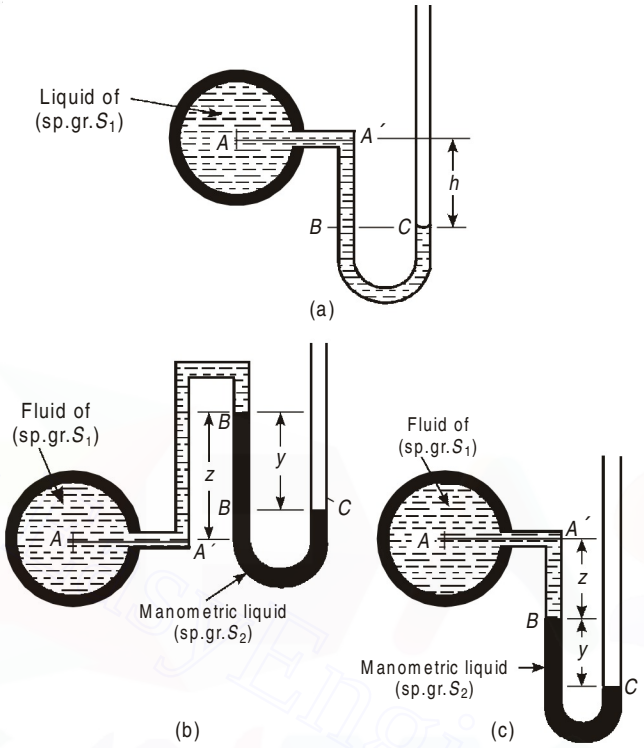


Figure 2.9 *Measurement of negative pressure by U-tube simple manometer*

If the specific gravity of the fluid at A is S_1 (with respect to water) and the specific gravity of the manometric liquid is S_2, the gage equation for the arrangement of Fig. 2.9 (b) may be written in terms of liquid at A as,

$$\frac{p_A}{wS_1} = z - y\,\frac{S_2}{S_1} \qquad ...(2.35)$$

and in terms of water as

$$\frac{p_A}{w} = zS_1 - yS_2 \qquad ...(2.36)$$

Similarly, for the arrangement shown in Fig. 2.9 (c) the gage equation may be written in terms of liquid at A as,

$$\frac{p_A}{wS_1} = -z - y\,\frac{S_2}{S_1} \qquad ...(2.37)$$

and in terms of water as

$$\frac{p_A}{w} = -zS_1 - yS_2 \qquad ...(2.38)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The arrangement of Fig. 2.9 (c) is generally preferred to that of Fig. 2.9 (b).

(iii) *Single Column Manometer.* The U-tube manometers described above usually require readings of fluid levels at two or more points, since a change in pressure causes a rise of liquid in one limb of the manometer and a drop in the other. This difficulty may however be overcome by using single column manometers. A single column manometer is a modified form of a U-tube manometer in which a shallow reservoir having a large cross-sectional area (about 100 times) as compared to the area of the tube is introduced into one limb of the manometer, as shown in Fig. 2.10. For any variation in pressure, the change in the liquid level in the reservoir will be so small that it may be neglected, and the pressure is indicated approximately by the height of the liquid in the other limb. As such only one reading in the narrow limb of the manometer need be taken for all pressure measurements. The narrow limb of the manometer may be vertical as in Fig. 2.10 (a) or it may be inclined as in Fig. 2.10 (b) . The inclined type is useful for the measurement of small pressures. As explained later since no reading is required to be taken for the level of liquid in the reservoir, it need not be made of transparent material.

When the manometer is not connected to the container, the surface of the manometric liquid in the reservoir will stand at level 0 – 0, and since it is subjected to a pressure due to a column of fluid of height y and specific gravity S_1 , the surface of the manometric liquid in the tube will stand at B, at a height h_1 above 0 – 0, such that from Eq. 2.10 (a)

$$yS_1 = h_1S_2 \qquad ...(2.39)$$

where S_2 is the specific gravity of the manometric liquid. This is known as the normal position of the manometric liquid. On being connected to the container at the gage point, the high pressure fluid will enter the reservoir, due to which there will be a drop in the manometric liquid surface in the reservoir by a distance Δy and a consequent rise in the tube by a distance h_2, above B. If A and a are the cross-sectional areas of the reservoir and the tube respectively, then,

$$A\,(\Delta y) = ah_2 \qquad ...(2.40)$$

If p_A is the pressure intensity at A and w is the specific weight of water, then starting from D the following gage equation in terms of water is obtained

$$0 + (h_2 + h_1 + \Delta y)S_2 - (\Delta y)S_1 - yS_1 = \frac{p_A}{w} \qquad ...(2.41)$$

Introducing Eqs (2.39) and (2.40) in Equation (2.41) it becomes

$$\frac{p_A}{w} = h_2\left[S_2 + (S_2 - S_1)\frac{a}{A}\right] \qquad ...(2.42)$$

By making the reservoir sufficiently large, the ratio $\left(\frac{a}{A}\right)$ can be made so small that Δy is negligible and height $(h_1 + h_2)$ is a measure of the pressure head at the contact level $C - C$ in the reservoir. In which case Eq. 2.42 reduces to

$$\frac{p_A}{w} = h_2S_2 \qquad ...(2.43)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

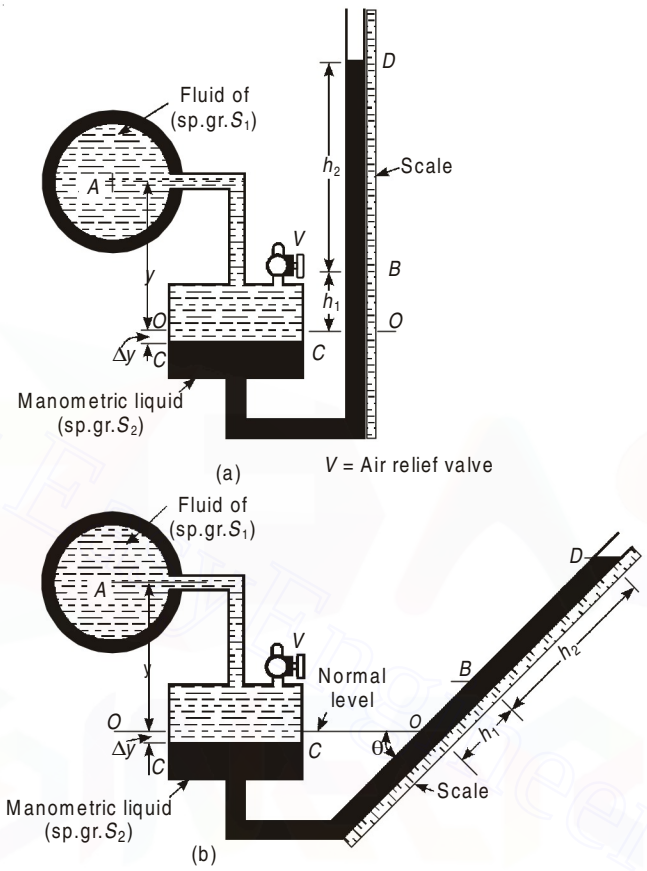


Figure 2.10 *Single column manometer*

so that only one reading of the height of level of liquid in the narrow tube is required to be taken to obtain the pressure head at A. However, if Δy is appreciable, then since the terms within brackets on the right side of Eq. 2.42 are constant, the scale on which h_2 is read can be so graduated as to correct for Δy so that again only one reading of the height of liquid level in the narrow tube is required to be taken, which will directly give the pressure head at A.

A single tube manometer can be made more sensitive by making its narrow tube inclined as shown in Fig. 2.10 (b). With this modification the distance moved by the liquid in the narrow tube shall be comparatively more, even for small pressure intensity at A. As before when the manometer is not connected to the container, the manometric liquid surface in the reservoir will stand at level 0 – 0 and that in the tube will stand at B, such that

$$yS_1 = (h_1 \sin \theta)\, S_2 \qquad ...(2.44)$$

Due to high pressure fluid entering the reservoir, the manometric liquid surface will drop to level $C - C$ by a distance Δy, and it will travel a distance BD equal to h_2 in the narrow tube. Thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$A\,(\Delta y) = ah_2 \qquad \text{...(2.45)}$$

Again starting from D the gage equation in this case becomes

$$0 + (h_1 + h_2)\,(\sin\theta)\,S_2 + \Delta y S_2 - \Delta y S_1 - y S_1 = \frac{p_A}{w} \qquad \text{...(2.46)}$$

Introducing Eqs 2.44 and 2.45 in Eq. 2.46 it becomes

$$\frac{p_A}{w} = h_2\left[S_2 \sin\theta + (S_2 - S_1)\frac{a}{A}\right] \qquad \text{...(2.47)}$$

Again if the ratio is negligible then Eq. 2.47 reduces to

$$\frac{p_A}{w} = (h_2 \sin\theta)\,S_2 \qquad \text{...(2.48)}$$

Single column manometers can also be employed to measure the negative gage pressures. If the pressure at A in the container is negative, the manometric liquid surface in the reservoir will be raised by a certain distance and consequently there will be drop in the liquid surface in the tube. Again by adopting the same procedure the gage equations for the negative pressure measurement can also be obtained.

Differential Manometers. For measuring the difference of pressure between any two points in a pipeline or in two pipes or containers, a differential manometer is employed. In general a differential manometer consists of a bent glass tube, the two ends of which are connected to each of the two gage points between which the pressure difference is required to be measured. Some of the common types of differential manometers are as noted below:

(i) Two–Piezometer Manometer.

(ii) Inverted U-Tube Manometer.

(iii) U- Tube Differential Manometer.

(iv) Micromanometer.

(i) *Two-Piezometer Manometer.* As the name suggests this manometer consists of two separate piezometers which are inserted at the two gage points between which the difference of pressure is required to be measured. The difference in the levels of the liquid raised in the two tubes will denote the pressure difference between the two points. Evidently this method is useful only if the pressure at each of the two points is small. Moreover it cannot be used to measure the pressure difference in gases, for which the other types of differential manometers described below may be employed.

(ii) *Inverted U-tube Manometer.* It consists of a glass tube bent in U-shape and held inverted as shown in Fig. 2.11. Thus it is as if two piezometers described above are connected with each other at top. When the two ends of the manometer are connected to the points between which the pressure difference is required to be measured, the liquid under pressure will enter the two limbs of the manometer, thereby causing the air within the manometer to get compressed. The presence of the compressed air results in restricting the heights of the columns of liquids raised in the two limbs of the manometer. An air cock as shown in Fig. 2.11, is usually provided at the top of the inverted U-tube which facilitates the raising of the liquid columns to suitable level in both the limbs by driving out a portion of the compressed air. It also permits the expulsion of air bubbles which might have been entrapped somewhere in the pipeline.

If p_A and p_B are the pressure intensities at points A and B between which the inverted U-tube manometer is connected, then corresponding to these pressure intensities the liquid will rise above

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

points A and B upto C and D in the two limbs of the manometer as shown in Fig. 2.11. Now if w represents the specific weight of water and S_1 represents the specific gravity of the liquid at A or B,

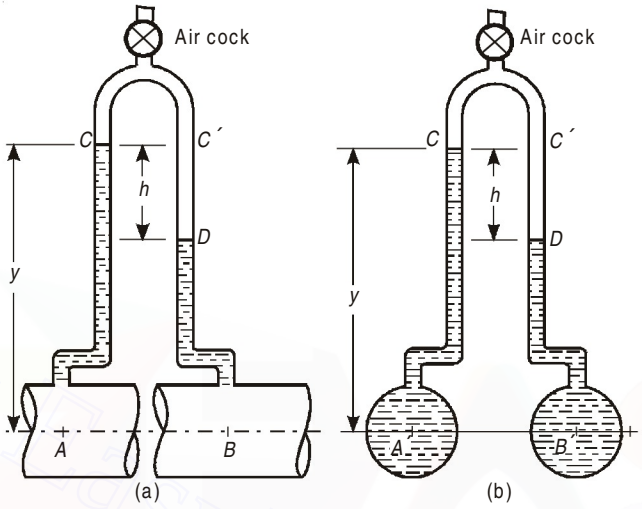


Figure 2.11 *Inverted U-Tube differential manometer*

then commencing from A, the pressure head at C in terms of water is equal to $\left(\frac{p_A}{w} - yS_1\right)$. Since points C and C' are at the same horizontal level and in the same continuous static mass of fluid

Pressure at C = Pressure at C'

Between points C' and D there is a column of compressed air. Since the specific weight of air is negligible as compared with that of liquid, the weight of air column between C' and D may be neglected. Hence,

Pressure at C' = Pressure at D

From D to B there being decrease in elevation, the pressure head increases so that the pressure head at B is equal to pressure head at D plus $(y - h)\, S_1$. Thus the gage equation may be expressed as

$$\frac{p_A}{w} - yS_1 + (y - h)\, S_1 = \frac{p_B}{w}$$

or
$$\frac{p_A}{w} - \frac{p_B}{w} = hS_1 \qquad ...(2.49)$$

Inverted U-tube manometers are suitable for the measurement of small pressure difference in liquids. Sometimes instead of air, the upper part of this manometer is filled with a manometric liquid which is lighter than the liquid for which the pressure difference is to be measured and is immiscible with it. Such an arrangement is shown in Fig. 2.12. As explained later the use of manometric liquid in this manometer results in increasing the sensitivity of the manometer.

Again if p_A and p_B are the pressure intensities at points A and B between which the inverted U-tube manometer is connected, then corresponding to these pressure intensities the liquid will rise above points A and B upto C and D in the two limbs of the manometer as shown in Fig. 2.12. Now

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

if w represents the specific weight of water and S_1 and S_2 are the specific gravities of the liquid at A or B and the manometric liquid (in the upper part of the manometer) respectively, then commencing

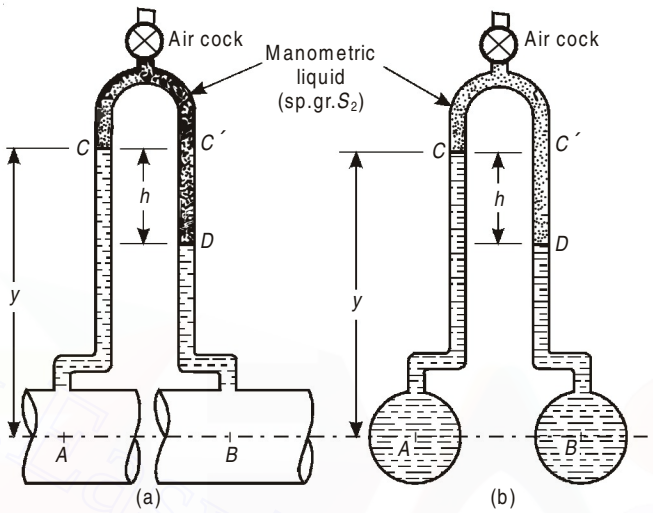


Figure. 2.12 *Inverted U-tube differential manometer with light manometric liquid*

from A, the pressure head at C in terms of water is equal to $\left(\frac{p_A}{w} - yS_1\right)$. Since points C and C' are at the same horizontal level and in the same continuous static mass of liquid,

Pressure at C = Pressure at C'

From C' to D there being decrease in elevation, the pressure head increases, so that the pressure head at D in terms of water is equal to

$$\left[\left(\frac{p_A}{w} - yS_1\right) + hS_2\right]$$

Further from D to B there is decrease in elevation and hence the gage equation becomes

$$\frac{p_A}{w} - yS_1 + hS_2 + (y - h)\, S_1 = \frac{p_B}{w}$$

or
$$\frac{p_A}{w} - \frac{p_B}{w} = h\,(S_1 - S_2) \qquad ...(2.50)$$

It is evident from Eq. 2.50 that as the specific gravity of the manometric liquid approaches that of the liquid at A or B $(S_1 - S_2)$ approaches zero and large values of h will be obtained even for small pressure differences, thus increasing the sensitivity of the manometer. Another arrangement for increasing the sensitivity of these manometers is to incline the gage tubes so that a vertical gage difference h is transposed into a reading which is magnified by $\frac{1}{\sin\theta}$, where θ is angle of inclination with the horizontal.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) *U-Tube Differential Manometer.* It consists of glass tube bent in U-shape, the two ends of which are connected to the two gage points between which the pressure difference is required to be measured. Figure 2.13 shows such an arrangement for measuring the pressure difference between any two points *A* and *B*. The lower part of the manometer contains a manometric liquid which is heavier than the liquid for which the pressure difference is to be measured and is immiscible with it.

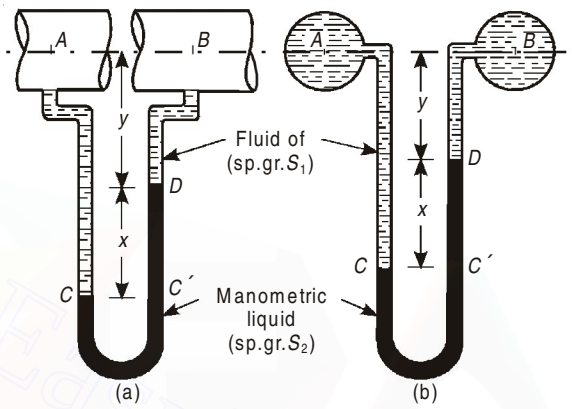


Figure 2.13 *U-tube Differential Manometer*

When the two limbs of the manometer are connected to the gage points *A* and *B*, then corresponding to the difference in the pressure intensities p_A and p_B the levels of manometric liquid in the two limbs of the manometer will be displaced through a distance *x* as shown in Fig. 2.13. By measuring this difference in the levels of the manometric liquid, the pressure difference $(p_A - p_B)$ may be computed as indicated below.

If S_1 and S_2 are the specific gravities of the liquid at *A* or *B* and the manometric liquid respectively, then by commencing at *A* where the pressure is p_A, the pressure head at *C* in terms of water is equal to $\left[\frac{p_A}{w} + (y + x)S_1\right]$, in which *w* is the specific weight of water. Further, since points *C* and *C′* are at the same level and are lying in the same continuous static mass of liquid,

Pressure at *C* = Pressure at *C′*

Further from *C′* to *D* there being an increase in the elevation, the pressure head decreases, so that the pressure head at *D* in terms of water is equal to $\left[\frac{p_A}{w} + (y + x)S_1 - xS_2\right]$. Similarly from *D* to *B* there is an increase in elevation and hence the gage equation becomes

$$\left[\frac{p_A}{w} + (y + x)S_1 - xS_2 - yS_1\right] = \frac{p_B}{w}$$

or
$$\frac{p_A}{w} - \frac{p_B}{w} = x\,(S_2 - S_1) \qquad ...(2.51)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If, for example, the manometric liquid is mercury ($S_2 = 13.6$) and the liquid at A or B is water ($S_1 = 1$) then the difference in pressure heads at the points A and B is 12.6 times the deflection x of the manometric liquid in the two limbs of the manometer. As such the use of mercury as manometric liquid in U-tube manometer is suitable for measuring large pressure differences. However, for small pressure differences, mercury makes precise measurement difficult, and hence for such cases it is common to use a liquid which is only slightly heavier than the liquid for which the pressure difference is to be measured.

Often the points A and B between which the pressure difference is to be measured are not at the same level, as shown in Fig. 2.14. For such cases also, by adopting the same procedure, the following gage equation may be obtained in order to compute the pressure difference between the points A and B.

$$\left[\frac{p_A}{w} + (z + y + x)S_1 - xS_2 - yS_3\right] = \frac{p_B}{w}$$

or
$$\frac{p_A}{w} - \frac{p_B}{w} = [\, x\,(S_2 - S_1) + y\,(S_3 - S_1) - zS_1\,] \qquad ...(2.52)$$

Equation 2.52 is, however, a general equation, which may be modified to derive the equations for different conditions. Thus, for example, if there is same liquid at A and B, then since $S_1 = S_3$, Eq. 2.52 becomes

$$\frac{p_A}{w} - \frac{p_B}{w} = [x\,(S_2 - S_1) - zS_1\,] \qquad ...(2.53)$$

Further if A and B are at the same level, then since $z = 0$, Eq. 2.53 becomes same as Eq. 2.51 which is quite obvious.

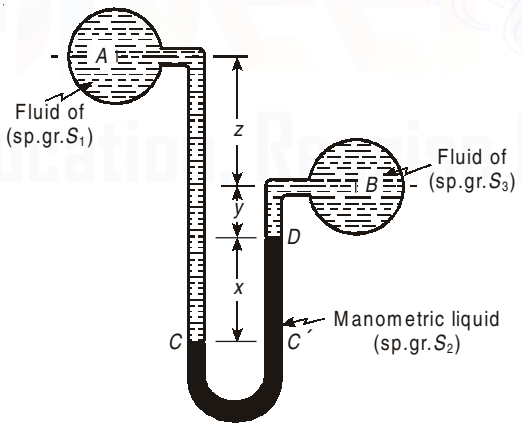


Figure 2.14 *U-tube differential manometer between two points at different levels*

(iv) *Micromanometers.* For the measurement of very small pressure differences, or for the measurement of pressure differences with very high precision, special forms of manometers called micromanometers are used. A wide variety of micromanometers have been developed, which either magnify the readings or permit the readings to be observed with greater accuracy. One simple type

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of micromanometer consists of a glass U-tube, provided with two transparent basins of wider sections at the top of the two limbs, as shown in Fig. 2.15. The manometer contains two manometric liquids of different specific gravities and immiscible with each other and with the fluid for which the pressure difference is to be measured. Before the manometer is connected to the pressure points A and B, both the limbs are subjected to the same pressure. As such the heavier manometric liquid of sp. gr. S_1 will occupy the level DD' and the lighter manometric liquid of sp. gr. S_2 will occupy the level CC'. When the manometer is connected to the pressure points A and B where the pressure intensities are p_A and p_B respectively, such that $p_A > p_B$ then the level of the lighter manometric liquid will fall in the left basin and rise in the right basin by the same amount Δy. Similarly the level of the heavier manometric liquid will fall in the left limb to point E and rise in the right limb to point F. If A and a are the cross-sectional areas of the basin and the tube respectively, then since the volume of the liquid displaced in each basin is equal to the volume of the liquid displaced in each limb of the tube the following expression may be readily obtained

$$A\,(\Delta y) = a\left(\frac{x}{2}\right) \qquad ...(2.54)$$

Further if w is specific weight of water, then, starting from point A the following gage equation in terms of water column may be obtained,

$$\frac{p_A}{w} + (y_1 + \Delta y)\,S_3 + \left(y_2 - \Delta y + \frac{x}{2}\right) S_2 - xS_1 - \left(y_2 - \frac{x}{2} + \Delta y\right) S_2 - (y_1 - \Delta y)\,S_3 = \frac{p_B}{w}$$

Substituting the value of Δy from Eq. 2.54 and simplifying the above equation it becomes

$$\frac{p_A}{w} - \frac{p_B}{w} = x\left[S_1 - S_2\left(1 - \frac{a}{A}\right) - S_3\frac{a}{A}\right] \qquad ...(2.55)$$

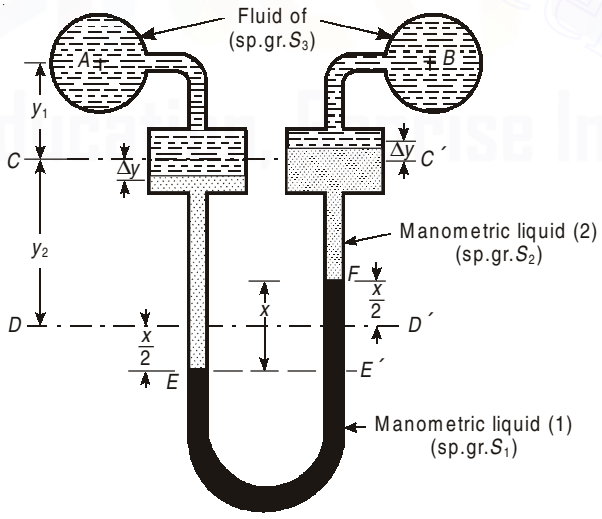


Figure 2.15 *Micromanometer*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The quantities within brackets on right side of Eq. 2.55 are constant for a particular manometer. Thus by measuring x and substituting in Eq. 2.55 the pressure difference between any two points can be known.

If the cross-sectional area of the basin is large as compared with the cross-sectional area of the tube, then the ratio a/A is very small and the Eq. 2.55 reduces to

$$\frac{p_A}{w} - \frac{p_B}{w} = x\,(S_1 - S_2) \qquad ...(2.56)$$

By selecting the two manometric liquids such that their specific gravities are very nearly equal then a measurable value of x may be achieved even for a very small pressure difference between the two points.

In a number of other types of micromanometers the pressure difference to be measured is balanced by the slight raising or lowering (on a micrometer screw) of one arm of the manometer whereby a meniscus is brought back to its original position. The micromanometers of this type are those invented by Chattock, Small and Krell, which are sensitive to pressure differences down to less than 0.0025 mm of water. However the disadvantage with such manometers is that an appreciable time is required to take a reading and they are therefore suitable only for completely steady pressures.

Mechanical Gages. Mechanical gages are those pressure measuring devices, which embody an elastic element, which deflects under the action of the applied pressure, and this movement mechanically magnified, operates a pointer moving against a graduated circumferential scale. Generally these gages are used for measuring high pressures and where high precision is not required. Some of the mechanical pressure gages which are commonly used are as noted below:

(i) Bourdon Tube Pressure Gage (ii) Diaphragm Pressure Gage

(iii) Bellows Pressure Gage (iv) Dead-weight Pressure Gage

(i) *Bourdon Tube Pressure Gage.* It is the most common type of pressure gage which was invented by E. Bourdon (1808–84). The pressure responsive element in this gage is a tube of steel or bronze which is of elliptic cross-section and is curved into a circular arc. The tube is closed at its outer end,

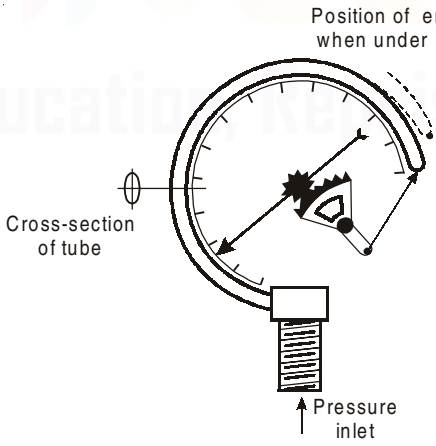


Figure 2.16 *Bourdon tube pressure gage*

and this end of the tube is free to move. The other end of the tube, through which the fluid enters, is rigidly fixed to the frame as shown in Fig. 2.16. When the gage is connected to the gage point, fluid

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

under pressure enters the tube. Due to increase in internal pressure, the elliptical cross-section of the tube tends to become circular, thus causing the tube to straighten out slightly. The small outward movement of the free end of the tube is transmitted, through a link, quadrant and pinion, to a pointer which by moving clockwise on the graduated circular dial indicates the pressure intensity of the fluid. The dial of the gage is so calibrated that it reads zero when the pressure inside the tube equals the local atmospheric pressure, and the elastic deformation of the tube causes the pointer to be displaced on the dial in proportion to the pressure intensity of the fluid. By using tubes of appropriate stiffness, gages for wide range of pressures may be made. Further by suitably modifying the graduations of the dial and adjusting the pointer Bourdon tube vacuum gages can also be made. When a vacuum gage is connected to a partial vacuum, the tube tends to close, thereby moving the pointer in anti-clockwise direction, indicating the negative or vacuum pressure. The gage dials are usually calibrated to read newton per square metre (N/m^2),or pascal (Pa), or kilogram (f) per square centimetre [$kg(f)/cm^2$]. However other units of pressure, such as metres of water or centimetres of mercury, are also frequently used.

(ii) *Diaphragm Pressure Gage.* The pressure responsive element in this gage is an elastic steel corrugated diaphragm. The elastic deformation of the diaphragm under pressure is transmitted to a pointer by a similar arrangement as in the case of Bourdon tube pressure gage (see Fig. 2.17). However, this gage is used to measure relatively low pressure intensities. The Aneroid barometer operates on a similar principle.

(iii) *Bellows Pressure Gage.* In this gage the pressure responsive element is made up of a thin metallic tube having deep circumferential corrugations. In response to the pressure changes this elastic element expands or contracts, thereby moving the pointer on a graduated circular dial as shown in Fig. 2.18.

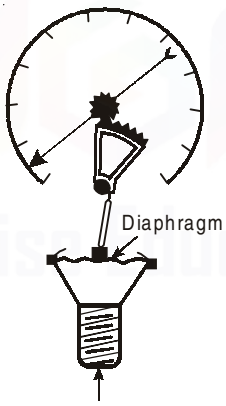


Figure 2.17 *Diaphragm pressure gage*

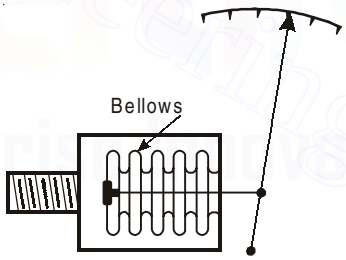


Figure 2.18 *Bellows pressure gage*

(iv) *Dead -Weight Pressure Gage.* A simple form of a dead-weight pressure gage consists of a plunger of diameter d, which can slide within a vertical cylinder, as shown in Fig. 2.19. The fluid under pressure, entering the cylinder, exerts a force on the plunger, which is balanced by the weights loaded on the top of the plunger. If the weight required to balance the fluid under pressure is W, then the pressure intensity p of the fluid may be determined as,

$$p = W \Big/ \left(\frac{\pi}{4} d^2 \right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The only error that may be involved is due to frictional resistance offered to motion of the plunger in the cylinder. But this error can be avoided if the plunger is carefully ground, so as to fit with the least permissible clearance in the cylinder. Moreover, the whole mass can be rotated by hand before final readings are taken.

Dead-weight gages are generally not used so much to measure the pressure intensity at a particular point as to serve as standards of comparison. Hence as shown in Fig. 2.19, a pressure gage which is to be checked or calibrated is set in parallel with the dead-weight gage. Oil under pressure is pumped into the gages, thereby lifting the plunger and balancing it against the oil pressure by loading it with known weights. The pressure intensity of the oil being thus known, the attached pressure gage can either be tested for its accuracy or it can be calibrated.

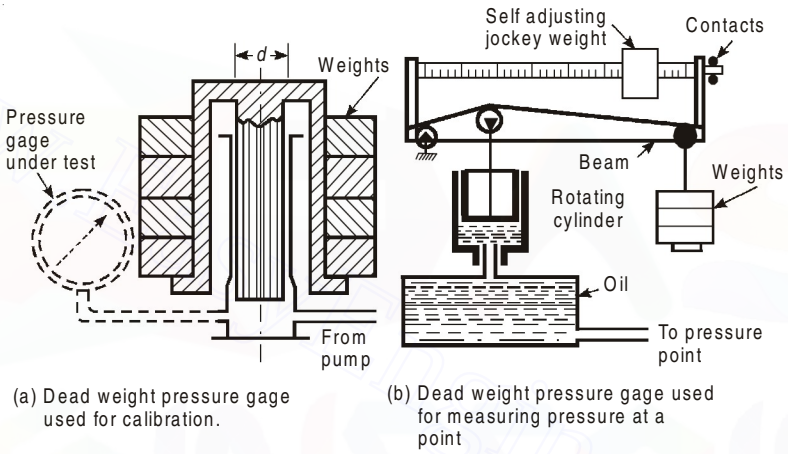


Figure 2.19 *Dead-weight pressure gage*

A dead-weight gage which can be used for measuring pressure at a point with more convenience is also shown in Fig. 2.19. In this gage a lever, same as in some of the weighing machines, is provided to magnify the pull of the weights. The load required to balance the force due to fluid pressure is first roughly adjusted by hanging weights from the end of the main beam. Then a smaller jockey weight is slided along to give precise balance. In more precise type of gage the sliding motion may be contrived automatically by an electric motor.

2.7 GENERAL COMMENTS ON CONNECTIONS FOR MANOMETERS AND GAGES

The following points should be kept in view while making connections for the various pressure measuring devices:

(i) At the gage point the hole should be drilled normal to the surface and it should flush with the inner surface.

(ii) The diameters of the holes at the gage points should be about 3 mm to 6 mm.

(iii) The holes should not disturb the internal surface and no burrs or irregularities must be left.

(iv) There should be no air pockets left over in the connecting tubes, which should be completely filled with the liquid. The presence of air bubbles can easily be detected if the connecting tubes are made of polythene or similar transparent material.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ILLUSTRATIVE EXAMPLES

Example 2.1. *Express pressure intensity of 7.5 kg (f)/cm² in all pressure units. Take the barometer reading as 76 cm of mercury.*

Solution

(A) *Gage Units*

(a) $p = 7.5$ kg(f)/cm²

(b) $p = 7.5 \times 10^4$ kg(f)/m²

(c) $h = \frac{p}{w} = \frac{7.5 \times 10^4}{1000} = 75$ m of water

(d) $h = \frac{p}{w} = \frac{7.5 \times 10^4}{13.6 \times 1000} = 5.51$ m of mercury

(e) $p = 9\,810 \times 75 = 73.575 \times 10^4$ N/m²

(B) *Absolute Units*

Absolute pressure = Gage pressure + Atmospheric pressure

Atmospheric pressure = 76 cm of mercury

$= \frac{76 \times 13.6}{100} = 10.34$ m of water

$= \frac{76 \times 13.6 \times 1000}{100} = 1.034 \times 10^4$ kg(f)/m²

$= \frac{76 \times 13.6 \times 1000}{100 \times 10^4} = 1.034$ kg(f)/cm²

$= \frac{76 \times 13.6 \times 9810}{100} = 10.14 \times 10^4$ N/m²

(a) Absolute pressure = (7.5 + 1.034)
= 8.534 kg(f)/cm²

(b) Absolute pressure = $(7.5 \times 10^4 + 1.034 \times 10^4)$
= 8.534×10^4 kg(f)/m²

(c) Absolute pressure head = (75 + 10.34)
= 85.34 m of water

(d) Absolute pressure head = (5.51 + 0.76)
= 6.27 m of mercury

(e) Absolute pressure = $\frac{6.27}{0.76}$
= 8.25 atmospheres

(f) Absolute pressure = $(73.58 \times 10^4 + 10.14 \times 10^4)$
= 83.72×10^4 N/m²

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 2.2. *Find the depth of a point below water surface in sea where pressure intensity is 1.006 MN/m². Specific gravity of sea water = 1.025.*

Solution

Depth of sea water above the point is

$$h = \frac{p}{w}$$

$$p = 1.006 \text{ MN/m}^2$$

$$= 1.006 \times 10^6 \text{ N/m}^2$$

$$w = (1.025 \times 9\,810) \text{ N/m}^2$$

$$= 1.006 \times 10^4 \text{ N/m}^3$$

$$h = \frac{1.006 \times 10^6}{1.006 \times 10^4} = 100 \text{ m}$$

Example 2.3. *Convert a pressure head of 100 m of water to (a) kerosene of specific gravity 0.81, (b) carbon tetrachloride of specific gravity 1.6.*

Solution

From Eq. 2.10 (a)

$$h_1 S_1 = h_2 S_2$$

Thus by substitution

(a) $$100 \times 1 = h_2 \times 0.81$$

$$h_2 = \frac{100}{0.81}$$

$$= 123.46 \text{ m of kerosene}$$

(b) $$100 \times 1 = h_2 \times 1.6$$

$$h_2 = \frac{100}{1.6} = 62.5 \text{ m of carbon tetrachloride}$$

Example 2.4. *The left leg of a U-tube mercury manometer is connected to a pipe-line conveying water, the level of mercury in the leg being 0.6 m below the center of pipe-line, and the right leg is open to atmosphere. The level of mercury in the right leg is 0.45 m above that in the left leg and the space above mercury in the right leg contains Benzene (specific gravity 0.88) to a height of 0.3 m. Find the pressure in the pipe.*

Solution

In the accompanying figure the *pressures at C and C′* are equal. Thus computing the pressure heads at *C* and *C′* from either side and equating the same, we get

$$\frac{p_A}{w} + 0.6 = 0.45 \times 13.6 + 0.3 \times 0.88$$

or $$\frac{p_A}{w} = 5.784 \text{ m of water}$$

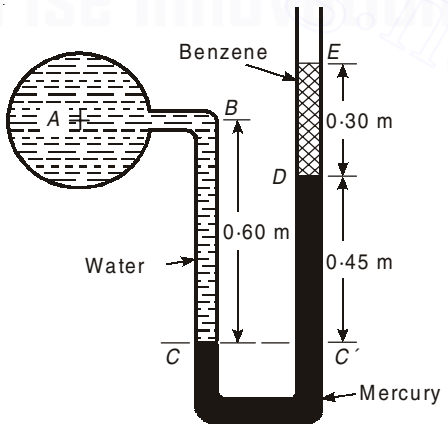


Figure Ex. 2.4

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore$ $p_A = (5.784 \times 9\,810)$

$= 5.674 \times 10^4\ \text{N/m}^2$

or $p_A = (5.784 \times 1000)$

$= 5.784 \times 10^3\ \text{kg (f)/m}^2$

$= 0.578\,4\ \text{kg (f)/cm}^2$

Example 2.5. *As shown in the accompanying figure, pipe M contains carbon tetrachloride of specific gravity 1.594 under a pressure of 1.05 kg (f)/cm² and pipe N contains oil of specific gravity 0.8. If the pressure in the pipe N is 1.75 kg (f)/cm² and the manometric fluid is mercury, find the difference x between the levels of mercury.*

Solution

Equate the pressure heads at Z and Z′ as shown in the fig. Ex. 2.5

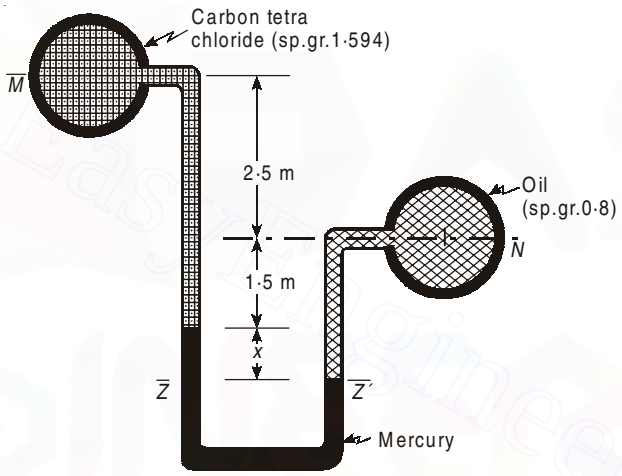


Figure Ex. 2.5

Pressure head at Z in terms of water column

$$= \left[\frac{1.05\times10^4}{1000} + (2.5+1.5)\times1.594 + x(13.6)\right] = 16.876 + 13.6x$$

Similarly pressure head at Z′ in terms of water column

$$= \left[\frac{1.75\times10^4}{1000} + (1.5\times0.8) + x(0.8)\right] = 18.7 + 0.8x$$

Thus equating the two, we get

$16.876 + 13.6x = 18.7 + 0.8x$

or $12.8x = 1.824$

$\therefore$ $x = \dfrac{1.824}{12.8} = 0.1425\ \text{m} = 14.25\ \text{cm}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 2.6. *The pressure between two points A and B in a pipe conveying oil of specific gravity 0.8 is measured by an inverted U-tube. The column connected to point B stands 1.6 m higher than that at point A. A commercial pressure gage attached directly to the pipe at A reads 1.125 kg (f)/cm^2; determine its reading when attached directly to the pipe at B.*

Solution

The difference of pressure head between the two points *A* and *B* is equal to 1.6 m of oil.

$$\therefore \quad \frac{p_B}{w} - \frac{p_A}{w} = (1.6 \times 0.8) = 1.28 \text{ m of water}$$

But $p_A = 1.125 \text{ kg (f)/cm}^2$

$$\therefore \quad \frac{p_A}{w} = \frac{1.125 \times 10^4}{1000} = 11.25 \text{ m of water}$$

Thus by substitution, we get

$$\frac{p_B}{w} - 11.25 = 1.28$$

or $$\frac{p_B}{w} = 12.53 \text{ m of water}$$

$$\therefore \quad p_B = (12.53 \times 1000) \text{ kg (f)/m}^2$$

$$= \frac{12.53 \times 1000}{10^4} = 1.253 \text{ kg (f)/cm}^2$$

Example 2.7. *Two pipes as shown in Fig. Ex. 2.7 convey toluene of specific gravity 0.875 and water respectively. Both the liquids in the pipes are under pressure. The pipes are connected to a U-tube manometer and the hoses connecting the pipes to the tubes are filled with the corresponding liquids. Find the difference of pressure in two pipes if the level of manometric liquid having a specific gravity 1.25 is 2.25 m higher in the right limb than the lower level of toluene in the left limb of the manometer.*

Solution

Let $p_{A'}$ and p_B be the pressure intensities at the centre of the pipes *A* and *B* respectively.

Since points *D* and *D′* lie at the same horizontal plane and in the same continuous static mass of liquid, the pressure heads at these two points will be equal.

$$\text{Pressure head at } D = \frac{p_A}{w} + (1.5 \times 0.875)$$

$$\text{Pressure head at } D' = \frac{p_B}{w} - 2.25 + (2.25 \times 1.25)$$

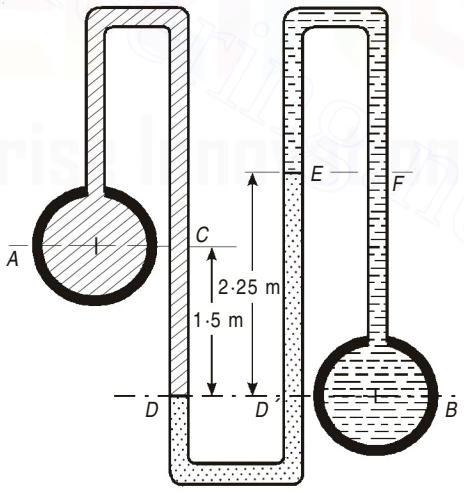


Figure Ex. 2.7

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The pressure heads have been calculated in terms of water column. Equating the pressure heads, we get

$$\frac{p_A}{w} + (1.5 \times 0.875) = \frac{p_B}{w} - 2.25 + (2.25 \times 1.25)$$

or $$\frac{p_A}{w} + 1.3125 = \frac{p_B}{w} + 0.5625$$

or $$\frac{p_A}{w} - \frac{p_B}{w} = 0.75$$

or $$(p_B - p_A) = (0.75 \times 1000)$$
$$= 750 \text{ kg(f)/m}^2$$

or $$(p_B - p_A) = 0.75 \times 9\,810$$
$$= 7\,357.5 \text{ N/m}^2$$
$$= 7.357\,5 \text{ kN/m}^2$$

Example 2.8. *Two tanks are filled with water of specific weight 9.81 kN/m^3. The bottoms of the tanks are connected to an inverted U-tube containing oil weighing 7.85 kN/m^3. Find the difference in pressure between the two tanks when the manometer gives a reading of 0.8 m.*

Solution

Using Eq. 2.50, the difference of pressure head in the two tanks is given by

$$\frac{p_1}{w} - \frac{p_2}{w} = x\,(S_1 - S_2)$$

$$= 0.8\left(1 - \frac{7.85}{9.81}\right)$$

$$= 0.16 \text{ m of water} = 16 \text{ cm of water}$$

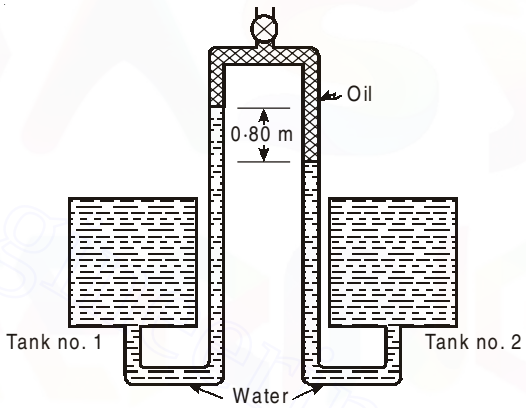


Figure Ex. 2.8

Example 2.9. *In the accompanying figure, fluid A is water, fluid B is oil of specific gravity 0.85, Z = 0.7 m and y = 1.5 m. Compute pressure difference between m and n.*

Solution

Let the height of the common surface above the point m be x. Since pressure head at T = pressure head at T' ; we have

$$\frac{p_m}{w} - x - (Z \times 0.85) = \frac{p_n}{w} - (Z + x - y)$$

or $$\frac{p_m}{w} - \frac{p_n}{w} = y - Z\,(1 - 0.85)$$
$$= 1.5 - 0.7(0.15)$$
$$= (1.5 - 0.105)$$
$$= 1.395 \text{ m of water}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$(p_m - p_n) = \frac{1.395 \times 1000}{10^4}$$

$$= 0.1395 \text{ kg(f)/cm}^2$$

or $$(p_m - p_n) = 1.395 \times 9.810$$

$$= 13.685 \text{ kN/m}^2$$

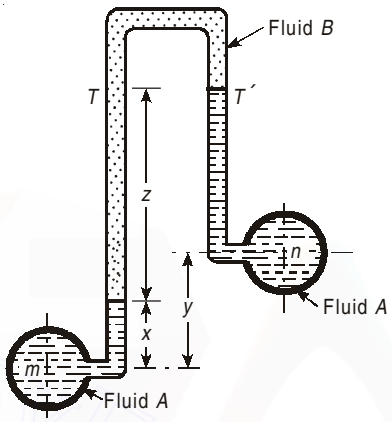


Figure Ex. 2.9

Example 2.10 (*a*) *Water fills the vessels shown in the Fig. Ex. 2.10 and a portion of the connecting tube. If the manometric liquid is oil of specific gravity 0.9, find the diference in pressure intensity at m and n when h = 1.25 m and Z = 0.3 m.*

(*b*) *If in the same figure instead of water there is mercury and the manometric liquid used has a specific gravity of 1.6, find the difference in pressure intensity at m and n when h = 0.6 m and Z = 1.0 m.*

Solution

(a) Pressure head at X = Pressure head at X'

Thus if the height of the common surface between oil and water in the right limb above point m is y, then

$$\frac{p_m}{w} - Z - y = \frac{p_n}{w} - (h + y) - Z\,(0.9) \qquad \text{...(i)}$$

or $$\frac{p_n}{w} - \frac{p_m}{w} = h - Z\,(1 - 0.9)$$

$$= 1.25 - 0.3\,(1 - 0.9)$$

$$= 1.22 \text{ m of water}$$

or $$p_n - p_m = \frac{1.22 \times 1000}{10^4} = 0.122 \text{ kg(f)/cm}^2$$

or $$p_n - p_m = 1.22 \times 9\,810$$

$$= 11\,968.2 \text{ N/m}^2 = 11.968\,2 \text{ kN/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) Substituting the corresponding values in Eq. (i) noted above, we get

$$\frac{p_m}{w} - (Z + y) \times 13.6 = \frac{p_n}{w} - (h + y) \times 13.6 - Z\,(1.6)$$

or
$$\frac{p_m}{w} - \frac{p_n}{w} = Z\,(13.6 - 1.6) - h\,(13.6)$$
$$= 1.0\,(13.6 - 1.6) - 0.6\,(13.6)$$

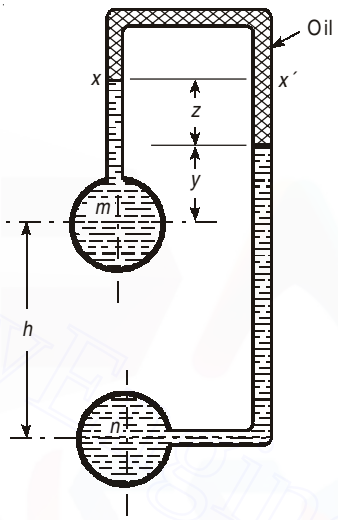


Figure Ex.2.10

$$= (12.0 - 8.16) = 3.84 \text{ m of water}$$

or
$$(p_m - p_n) = \frac{3.84 \times 1000}{10^4} = 0.384 \text{ kg(f)/cm}^2$$

or
$$(p_m - p_n) = 3.84 \times 9\,810 = 37\,670.4 \text{ N/m}^2 = 37.670\,4 \text{ kN/m}^2.$$

Example 2.11. *Figure Ex. 2.11 shows a differential gage. X and Y are connected to two different sources of pressure. With the equal pressure at X and Y, tops of kerosene columns stand at the common level J-J and water at 0-0. Points X and Y are at the same level. Find the difference of pressure head between X and Y in mm of water if h is 0.3 m. Take the reservoir cross-section 100 times that of glass tube.*

Solution

Let A and a be the cross-sectional areas of the reservoir and the glass tube respectively.

$\therefore \quad (A/a) = 100$

Let l be the vertical height of kerosene column above 0–0 in the right limb. Also let Z and Z' be the two points in the left and the right limbs respectively at the same level and at height $(h/2)$ above J-J as shown in Fig. Ex. 2.11.

$\therefore$ Pressure head at Z = Pressure head at Z'.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The pressure head at Z will be equal to $\frac{p_x}{w} - t - y - \left(l - \frac{y}{2}\right) 0.82 - h\,(0.82)$

where t is the vertical depth of the points X and Y below the common surface between kerosene and water in the right limb.

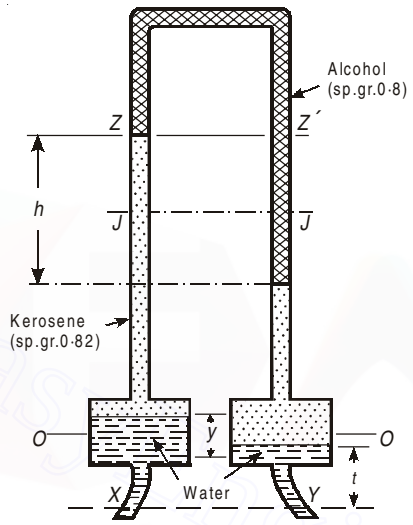


Figure Ex.2.11

Similarly the pressure head at point Z' will be equal to $\frac{p_y}{w} - t - \left(l + \frac{y}{2}\right)) 0.82 - h\,(0.80)$

Thus, we have

$$\frac{p_x}{w} - t - y - \left(l - \frac{y}{2}\right) 0.82 - h\,(0.82)$$

$$\frac{p_y}{w} - t - \left(l + \frac{y}{2}\right) 0.82 - h\,(0.8)$$

or $$\frac{p_x}{w} - \frac{p_y}{w} = h\,(0.82 - 0.80) + y(1 - 0.82)$$

Further

$$A \times y = a \times h$$

and $$h = 0.3 \text{ m}$$

$$\therefore \quad y = \frac{a}{A} \times h = \frac{0.3}{100} = 0.003 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substituting these values, we get

$$\frac{p_x}{w} - \frac{p_y}{w} = 0.3(0.82 - 0.80) + 0.003(1 - 0.82)$$

$$= 6.54 \times 10^{-3} \text{ m} = 6.54 \text{ mm of water.}$$

Example 2.12. *A two-liquid double column enlarged -ends manometer is used to measure a high precision pressure difference between two points of a pipeline containing gas under pressure. The basins are partly filled with methyl alcohol of specific gravity 0.78 and the lower portion of the U–tube is filled with mercury of specific gravity 13.6. The specific weight of the gas which is methane is 0.476 kg(f)/m^3. Find the pressure difference if the U-tube reading is 30 mm and the diameter of the basin is 15 times that of the U-tube.*

Solution

Let p_1 and p_2 be the pressure intensities at the two points 1 and 2 in the pipeline. Thus equating the pressure heads at the two points Z and Z', as shown in the accompanying figure, we get

$$\frac{p_1}{w} + (l + y)\left(\frac{0.476}{1000}\right) + m\,(0.78) + 30\,(0.78)$$

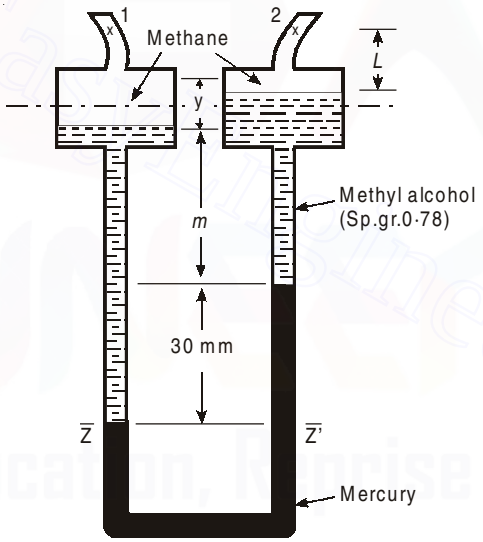


Figure Ex. 2.12

$$= \frac{p_2}{w} + l\left(\frac{0.476}{1000}\right) + (m + y)\,0.78 + 30\,(13.6)$$

or
$$\frac{p_1}{w} - \frac{p_2}{w} = 30(13.6 - 0.78) + y\,(0.78 - 0.476 \times 10^{-3})$$

$$= 384.6 + y\,(0.779\,5)$$

Further

$$A \times y = 30 \times a$$

or
$$y = 30\,\frac{a}{A} \text{ mm}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

But $A = (15)^2 a = 225\, a$

$\therefore \quad y = \dfrac{30}{225}$

By substituting these values, we get

$$\frac{p_1}{w} - \frac{p_2}{w} = 384.6 + \frac{30}{225}(0.7795)$$

$$= 384.704 \text{ mm of water}$$

$$= 38.470\ 4 \text{ cm of water}$$

Example 2.13. *A manometer consists of an inclined glass tube which is connected to a metal cylinder standing upright, and manometric liquid fills the apparatus to a fixed zero mark on the tube when both cylinder and the tube are open to the atmosphere. The upper end of the cylinder is then connected to a gas supply at a pressure p and the liquid rises in the tube.*

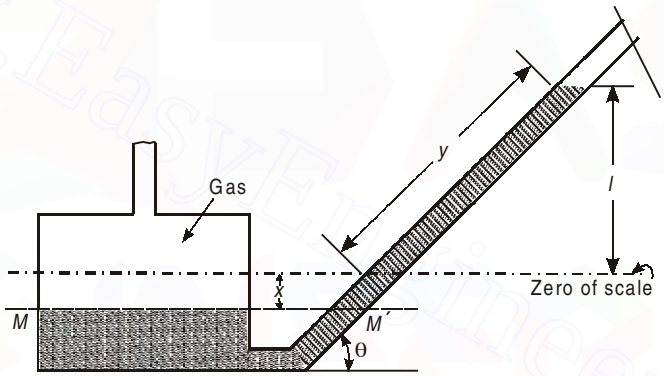


Figure Ex. 2.13

Find an expression for the pressure p in cm of water when the liquid reads y cm in the tube, in terms of the inclination θ of the tube, the specific gravity of the liquid S, and the ratio ρ of the diameter of the cylinder to the diameter of the tube. Hence determine the value of ρ so that the error due to disregarding the change in level in the cylinder will not exceed 0.1% when θ = 30°.

Solution

When the level of liquid rises in the tube by y cm, the level of the liquid will fall in the reservoir by certain amount; say x, which will be given by

$$x\left(\frac{\pi D^2}{4}\right) = y\left(\frac{\pi d^2}{4}\right)$$

or $$x = y\left(\frac{d}{D}\right)^2 = \frac{y}{\rho^2}$$

Further the vertical rise corresponding to y cm of rise $= l = (y \sin \theta)$.

Thus equating the pressure heads at the two points M and M', and neglecting the height of the gas column, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p}{w} = xS + (y \sin\theta)\, S$$

$$= \left(\frac{y}{\rho^2} + y \sin\theta\right) S$$

$$= y\,(\rho^{-2} + \sin\theta)S$$

where p and w are the pressure intensity of the gas and the specific weight of water respectively.

When the change in the level of the liquid in the cylinder is disregarded, the pressure head of the gas will become

$$\frac{p}{w} = (y \sin\theta)S$$

For the error due to disregarding the change in level of liquid in the cylinder not to exceed 0.1%,

$$\frac{99.9}{100}\left(\sin\theta + \frac{1}{\rho^2}\right) yS = (y \sin\theta)\, S$$

But $\theta = 30°$

Thus

$$\frac{99.9}{100}\left(\frac{1}{2} + \frac{1}{\rho^2}\right) = \frac{1}{2}$$

or $$\frac{0.999}{\rho^2} = \frac{0.1}{2 \times 100}$$

or $$\rho^2 = 1998$$

$\therefore$ $$\rho = 44.7.$$

Example 2.14. *For a gage pressure at A of –0.15 kg(f)/cm², determine the specific gravity of the gage liquid B in the Fig. Ex. 2.14.*

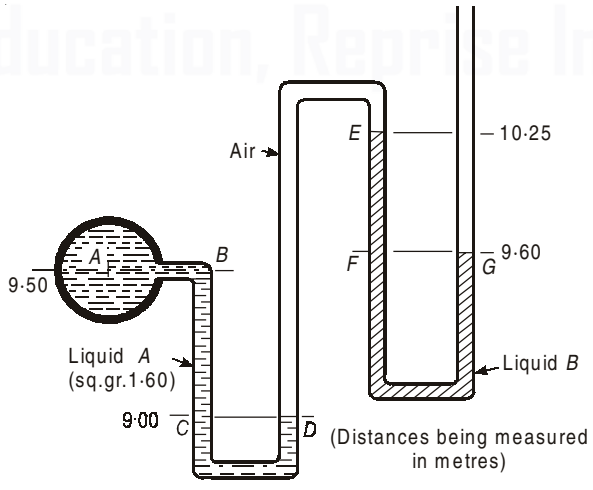


Figure Ex. 2.14

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Pressure at C = Pressure at D

Thus

$-(0.15) \times 10^4 + (1000 \times 1.6 \times 0.5) = p_D$

or $p_D = -0.7 \times 103 \text{ kg(f)/m}^2$

Between points D and E, since there is an air column which can be neglected. Thus

$p_D = p_E$

Also, pressure at F = pressure at G

But point G being at atmospheric pressure,

$p_G = 0 = p_F$

Thus $p_F = p_E + S \times 1000(10.25 - 9.60) = 0$

or $-0.7 \times 103 + S \times 1000\ (0.65) = 0$

$\therefore \quad S = \dfrac{0.70}{0.65} = 1.077.$

Example 2.15. *The pressure head at level A – A is 0.1 m of water and the unit weight of gas and air are 0.5607 kg(f) m³ and 1.2608 kg(f)/m³ respectively. Determine the reading of the water in the U-tube gage which measures the gas pressure at level B in the Fig. 2.15.*

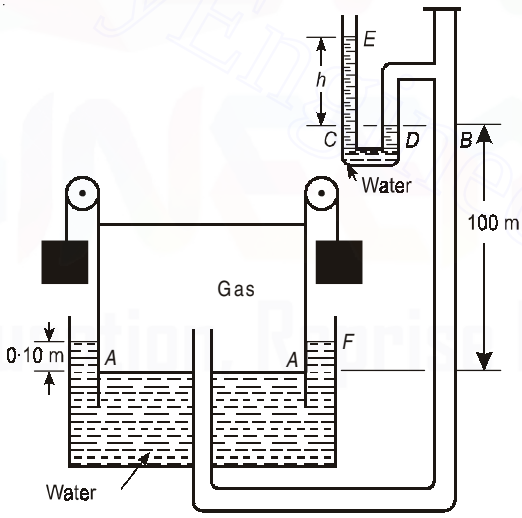


Figure Ex. 2.15

Solution

Assume that the values of specific weight for air and gas remain constant for the 100 m difference of elevation. Because the specific weights of gas and air are of the same order of magnitude, the change in atmospheric pressure with altitude must be taken into account.

Thus Absolute pressure at C = Absolute pressure at D or Atmospheric pressure $p_E + 1000 \times h$ = Absolute pressure at $A - (0.5607 \times 100)$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The absolute pressure at A will now be evaluated in terms of the atmospheric pressure at E, obtaining first the atmospheric pressure at F and then p_A.

Absolute pr. p_A = [atmospheric pr. P_E +1.2608 (h +100 – 0.10)] + (0.10 × 1000)

By substituting the value of absolute pr. p_A in the above equation and cancelling atmospheric pr. p_E, we get

$$1000 \times h = 1.2608\,(h + 99.9) + (1000 \times 0.10) - (0.5607 \times 100)$$

or $$(1000 - 1.2608)h = (1.2608 \times 99.9) + 100 - 56.07 = 169.884$$

or $$h = \frac{169.884}{998.7392} \text{ m of water} = \frac{169.884 \times 100}{998.7392} \text{ cm of water}$$

$$= 17.0098 \text{ cm of water.}$$

Example 2.16. *Point A is 0.25 m below the surface of the liquid of specific gravity 1.25, in the vessel as shown in Fig. Ex. 2.16. What is the pressure at A if a liquid of specific gravity 1.36 rises 2.1 m in the tube?*

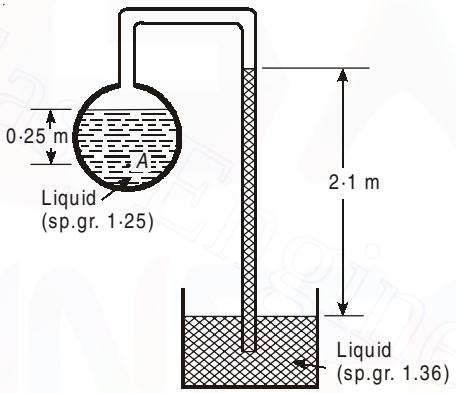


Figure Ex. 2.16

Solution

The pressure of the air in the vessel

$$= -(2.1 \times 1.36) \text{ m of water}$$

$$= -2.856 \text{ m of water.}$$

Thus pressure at point A in the vessel

$$= [-2.856 + 0.25 \times 1.25] \text{ m of water} = [-2.856 + 0.312\,5] \text{ m of water}$$

$$= -2.543\,5 \text{ m of water}$$

$$= -(2.543\,5 \times 9\,810)\text{N/m}^2$$

$$= -24.952 \text{ kN/m}^2$$

$$= -254\,3.5 \text{ kg(f)/ m}^2$$

$$= -0.254\,35 \text{ kg(f)/cm}^2$$

Example 2.17. *The cylinder and tubing shown in Fig. Ex. 2.17 contain oil of specific gravity 0.902. For a gage reading of 2 kg (f)/cm^2, what is the total weight of the piston and the slab placed on it ?*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

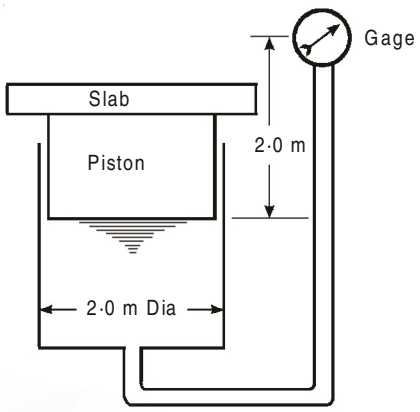


Figure Ex. 2.17

Solution

Let W be the total weight of the piston and the slab.

Since pr. below the piston = pr. at the same level in the vertical tube, we have

$$\frac{W}{\frac{\pi}{4}(2)^2} = 2 \times 10^4 + (2 \times 1000 \times 0.902)$$

$$\therefore \quad W = \pi(2 \times 10^4 + 0.180\ 4 \times 10^4)$$

$$= 6.85 \times 10^4 \text{ kg(f)}.$$

Example 2.18. *The compartments B and C shown in Fig. Ex. 2.18 are closed and filled with air. The barometer reads 99.572 kN/m^2. When the gages A and D read as indicated, what should be the value of x for gage E, if mercury is the manometric liquid in each U-tube gage ?*

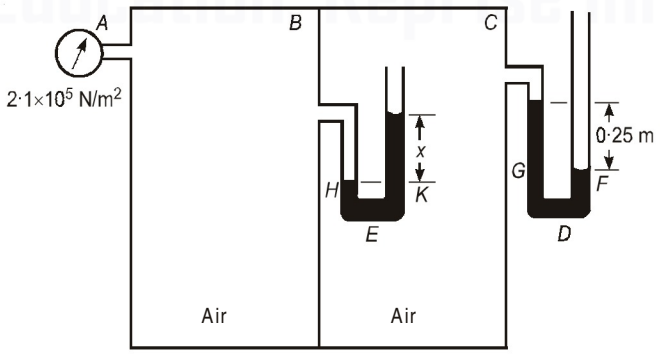


Figure Ex. 2.18

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Pressure at F = Pressure at G. Thus if p is the pressure intensity of air in the compartment C in terms of absolute units, then

$$\frac{p_c}{w} + 0.25 \times 13.6 = \frac{99.572 \times 10^3}{9810}$$

or
$$\frac{p_c}{w} = (10.15 - 3.4)$$
$$= 6.75 \text{ m of water (absolute)}$$

Further pressure at K = Pressure at H

Thus in terms of absolute units

$$6.75 + x\,(13.6) = \left[\frac{2.1 \times 10^5}{9810} + \frac{99.572 \times 10^3}{9810}\right]$$

or
$$x = \frac{1}{13.6}(31.56 - 6.75) = 1.824 \text{ m}$$

Example 2.19. *The pressure of illuminating gas is measured as shown in Fig. 2.19. The U-tube at the gas main shows a pressure of 75 mm of water. What difference should the upper tube show in mm of water? The right hand columns of both gages are open to the atmosphere. The density of both air and gas may be assumed constant throughout the 100 m of elevation. Air weighs 1.33 kg(f)/m^3 and the gas weighs 0.56 kg(f)/m^3 .*

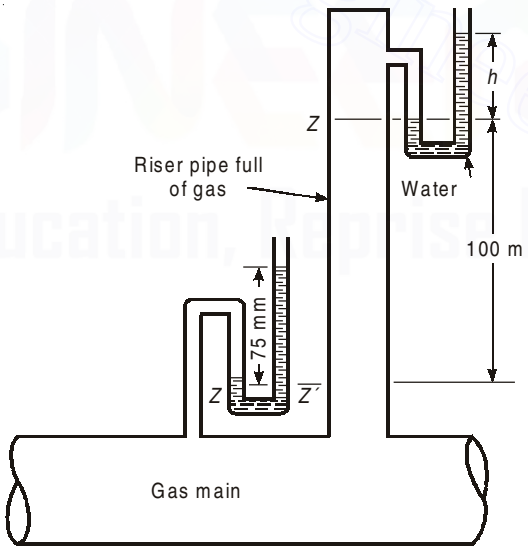


Figure Ex. 2.19

Solution

Let p_a represent the intensity of the atmospheric pressure in kg(f)/m^2 at an elevation of 100 m.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus the absolute pressure of the gas at point Z will be equal to

$$(p_a + h \times 1000) \text{ kg(f)/m}^2$$

The pressure at point Z' in the riser pipe at 100 m below Z will be given by

$$\text{Pressure at } Z' = \text{Pressure at } Z + (100 \times 0.56) = p_a + h \times 1000 + (100 \times 0.56)$$

Further

$$\text{Pressure at } Z' = \text{Pressure at } Z''$$

Thus

$$(p_a + h \times 1000) + (100 \times 0.56) = [p_a + (1.33 \times 100)] + \frac{75}{1000} \times 1000$$

or $$1000h = (133 + 75 - 56)$$

$\therefore$ $$h = 0.152 \text{ m}$$

Example 2.20. *For the multiple differential manometer shown in Fig. Ex. 2.20, if points A, B and C are at the same elevation, what is the difference in pressure heads in terms of water column between A and B, between A and C and between B and C?*

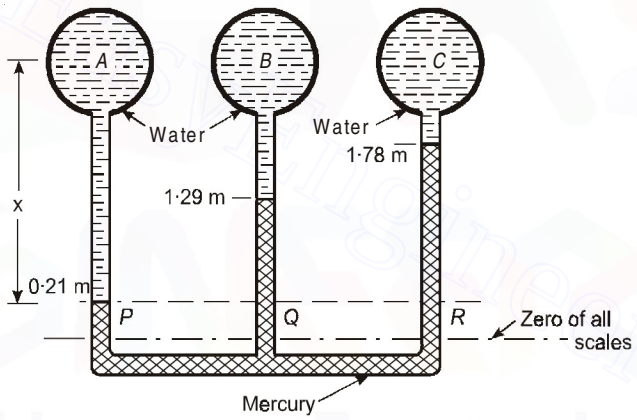


Figure Ex. 2.20

Solution

$$\text{Pr. at } P = \text{Pr. at } Q = \text{Pr. at } R$$

Thus if the depth of the point P below $A = x$, then

$$\frac{p_A}{w} + x = \frac{p_B}{w} + (x - 1.08) + (1.08 \times 13.6)$$

$$\frac{p_C}{w} = + (x - 1.57) + (1.57 \times 13.6)$$

Thus

$$\frac{p_A}{w} - \frac{p_B}{w} = 1.08\ (13.6 - 1)$$

$$= 13.608 \text{ m of water}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Similarly

$$\frac{p_A}{w} - \frac{p_C}{w} = 1.57\ (13.6 - 1)$$

$$= 19.782 \text{ m of water}$$

and

$$\frac{p_B}{w} - \frac{pc}{w} = (19.782 - 13.608)$$

$$= 6.174 \text{ m of water}$$

Example 2.21. *For a compound manometer shown in Fig. Ex. 2.21, what is gage pressure at C if the manometric fluid is mercury and if the fluid in the pipe and in the tubing which connects the two U-tubes is water?*

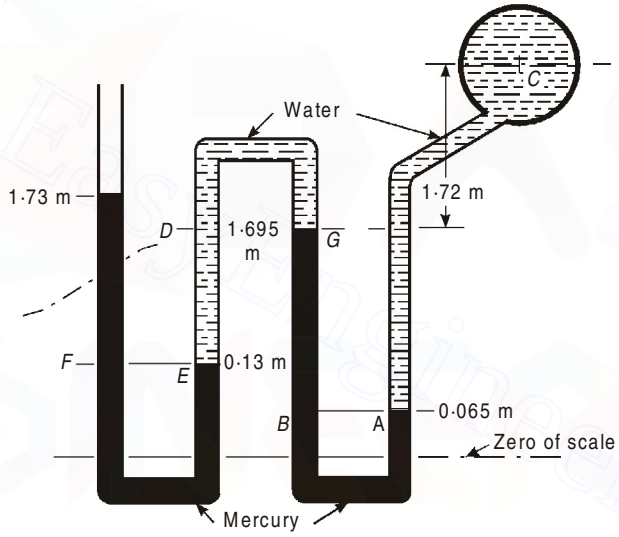


Figure Ex. 2.21

Solution

Pressure at A, p_A = Pressure at B, p_B

Thus we have

$$\frac{p_A}{w} = \frac{p_c}{w} + 1.72 + 1.695 - 0.065 = \frac{p_B}{w}$$

Further

Pressure at G, $$\frac{p_G}{w} = \frac{p_B}{w} - (1.695 - 0.065)\ 13.6$$

$$= \frac{p_C}{w} + (1.72 + 1.695 - 0.065) - (1.695 - 0.065) \times 13.6$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Also

Pressure at G, P_G = Pressure at D, p_D

and Pressure at E, p_E = Pressure at F, p_F

But

$$\frac{p_E}{w} = \frac{p_D}{w} + (1.695 - 0.13)$$

and $$\frac{p_F}{w} = (1.73 - 0.13) \times 13.6$$

Thus by substitution, we get

$$= \frac{p_C}{w} + (1.72 + 1.695 - 0.065) - (1.695 - 0.065) \times 13.6$$

$$= (1.73 - 0.13) \times 13.6 - (1.695 - 0.13)$$

or $$\frac{p_C}{w} + 3.35 - 22.168 = 21.76 - 1.565$$

$$\therefore \quad \frac{p_C}{w} = (43.928 - 4.915)$$

$$= 39.013 \text{ m of water}$$

i.e., $$p_C = 39.013 \times 10^3 \text{ kg(f)/m}^2$$

$$= 3.901\ 3 \text{ kg(f)/cm}^2$$

Example 2.22. *An empty cylindrical bucket, 0.3 m in diameter and 0.5 m long whose wall thickness and weight can be considered as negligible is forced with its open end first into water until its lower edge is 4 m below the surface. What force will be required to maintain position, assuming the trapped air to remain at constant temperature during the entire operation? Atmospheric pressure = 1.03 kg(f) / cm².*

Solution

Let p_a be the atmospheric pressure and p_1 be the absolute pressure of the compressed air trapped in the bucket. If the depth of water raised in the bucket is x, then since the temperature of the air remains constant, according to isothermal condition.

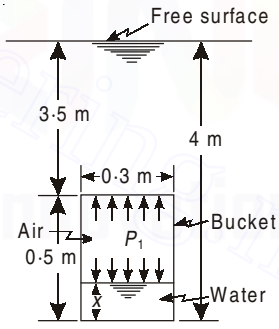


Figure Ex. 2.22

$$p_a \times \frac{\pi}{4}(0.3)^2 \times 0.5 = p_1 \times \frac{\pi}{4} \times (0.3)^2\,(0.5 - x)$$

or $$p_1 = \left(\frac{0.5}{0.5 - x}\right) p_a \qquad \text{...(i)}$$

Also $$p_1 = p_a + (4 - x) \times 1000 \qquad \text{...(ii)}$$

From Eqs. (i) and (ii), we get

$$\left(\frac{0.5}{0.5 - x}\right) p_a = p_a + (4 - x) \times 1000$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $p_a = 1.03 \times 10^4$ kg(f)/m^2, we get

$$\left(\frac{0.5}{0.5-x}\right) \times 1.03 \times 10^4 = 1.03 \times 10^4 + (4 - x) \times 1000$$

or $$\left(\frac{0.5}{0.5-x}\right) 1.03 \times 104 = (4 - x) \times 1000$$

$$\therefore \quad x = 0.136\,4 \text{ m}$$

Substituting the value of x in (i), we get

$$p_1 = \left(\frac{0.5}{0.5-0.1364}\right) \times 1.03 \times 10^4$$

$$= 1.4164 \times 10^4 \text{ kg(f)/m}^2$$

$$= 1.416\,4 \text{ kg(f)/cm}^2$$

The force tending to move the bucket in the upward direction

$$P_1 = p_1 \times \frac{\pi}{4}(0.3)^2$$

$$= 1.416\,4 \times 10^4 \times \frac{\pi}{4} \times (0.3)^2$$

$$= 1\,001.194 \text{ kg(f)}$$

The force acting on the bucket in the downward direction

$$P_2 = [1.03 \times 10^4 + 1000 \times 3.5] \frac{\pi}{4} \times (0.3)^2$$

$$= 13.8 \times 10^3 \times \frac{\pi}{4} \times (0.3)^2$$

$$= 975.464 \text{ kg(f)}$$

Thus the force required to maintain the bucket in this position

$$F = (P_1 - P_2)$$

$$= (1000.194 - 975.464)$$

$$= 24.73 \text{ kg(f)}$$

Example 2.23. *Petrol of specific gravity 0.8 flows upwards though a vertical pipe. A and B are two points in the pipe, B being 0.3 m higher than A. Connections are led from A and B to a U-tube containing mercury. If the difference of pressure between A and B is 0.18 kg(f)/cm^2, find the reading shown by the differential mercury manometer gage.*

Solution

Let p_A and p_B be the pressure intensities at the points A and B. If the reading of the gage is x, then as shown in Fig. Ex. 2.23.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_A}{w} + (x+y) \times 0.8 = \frac{p_B}{w} + (0.3+y)\,0.8 + (x \times 13.6)$$

$$\left(\frac{p_A}{w} - \frac{p_B}{w}\right) = x\,(13.6 - 0.8) + (0.3 \times 0.8)$$

But $$\left(\frac{p_A}{w} - \frac{p_B}{w}\right) = \frac{0.18 \times 10^4}{1000} = 1.8 \text{ m}$$

Hence $$1.8 = 12.8\,x + 0.24$$

$$\therefore \quad x = \frac{1.56}{12.8} = 0.122 \text{ m or } 12.2 \text{ cm.}$$

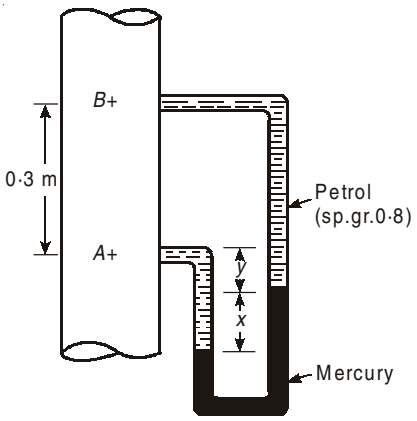


Figure Ex. 2.22

Example 2.24. *Assuming adiabatic conditions, find the temperature drop for an altitude of 2000 metres above the earth's surface. Assume the earth's surface temperature as 15°C, the gas constant R for air 289 m²/sec² per °C abs and n for standard polytropic atmosphere = 1.24. Also find the atmospheric pressure at that altitude, if it is given that it is 1.033 kg(f)/cm² at earth's surface.*

Solution

From Eq. 2.23 viz.,

$$\frac{T}{T_0} = \left[1 - \frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]$$

the temperature drop is given by

$$\frac{T_0 - T}{T_0} = \frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right); \; T_0 = (273+15) = 288° \text{ abs.}$$

By substituting the given values, we get

$$\frac{288 - T}{288} = \frac{9.81 \times 2000}{289 \times 288} \times \frac{1.24 - 1}{1.24}$$

$$\therefore \quad T = 274.86° \text{ abs.}$$

$\therefore$ Temperature drop = (288 – 274.86) = 13.14°C.

Further from Eq. 2.22 the variation of atmospheric pressure with altitude is given as

$$\frac{p}{p_0} = \left[1 - \frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]^{\frac{n}{n-1}}$$

By substituting the given values, we get

$$\frac{p}{1.033} = \left[1 - \frac{9.81 \times 2000}{289 \times 288} \times \left(\frac{1.24-1}{1.24}\right)\right]^{\left(\frac{1.24}{1.24-1}\right)}$$

$$p = 0.812 \text{ kg(f)/cm}^2.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 2.25. *At the top of a mountain the temperature is –5°C and a mercury barometer reads 56.6 cm, whereas the reading at the foot of the mountain is 74.9 cm. Assuming dry adiabatic conditions with R = 287 joule/[kg (m) deg C abs], calculate the height of the mountain.*

Solution

From Eq. 2.23, we have

$$\frac{T}{T_0} = \left[1 - \frac{g(z - z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]$$

R = 287 joule /[kg(m) deg C abs]

= 287 $m^2/(sec^2$ deg C abs)

T = (273 – 5) = 268°C abs.

For dry abiabatic conditions, n =1.4

Thus by substitution, we get

$$\frac{268}{T_0} = \left[1 - \frac{9.81(z - z_0)}{287 \times T_0} \times \left(\frac{1.4-1}{1.4}\right)\right]$$

$$\therefore \quad T_0 = \left[268 + \frac{9.81 \times (z - z_0) \times 0.4}{287 \times 1.4}\right]$$

Further from Eq. 2.22

$$\frac{p}{p_0} = \left[1 - \frac{g(z - z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]^{\frac{n}{n-1}}$$

By substitution, we get

$$\frac{56.6}{74.9} = \left[1 - \frac{9.81 \times (z - z_0) \times 0.4 \times 287 \times 1.4}{(287 \times 1.4)[(268 \times 287 \times 1.4) + 9.81 \times (z - z_0) \times 0.4]}\right]^{\frac{1.4}{0.4}}$$

or $$\left(\frac{56.6}{74.9}\right)^{\frac{0.4}{1.4}} = \left[1 - \frac{9.81 \times (z - z_0) \times 0.4}{(268 \times 287 \times 1.4) + 9.81 \times (z - z_0) \times 0.4}\right]$$

or $$0.923 = \frac{268 \times 287 \times 1.4}{(268 \times 287 \times 1.4) + 9.81 \times (z - z_0) \times 0.4}$$

$$\therefore \quad (z - z_0) = 2289 \text{ m.}$$

Example 2.26. *Figure Ex. 2.26 shows a device used to indicate the level inside a fuel tank. It is in the form of a U-tube with the crossing over. The U-tube is partially filled with a manometric fluid of specific weight w_2 which is heavier than the fuel whose specific weight is w_1. When the U-tube and indicating tube are vertical and the tank is full of fuel the reading on the scale is unity.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

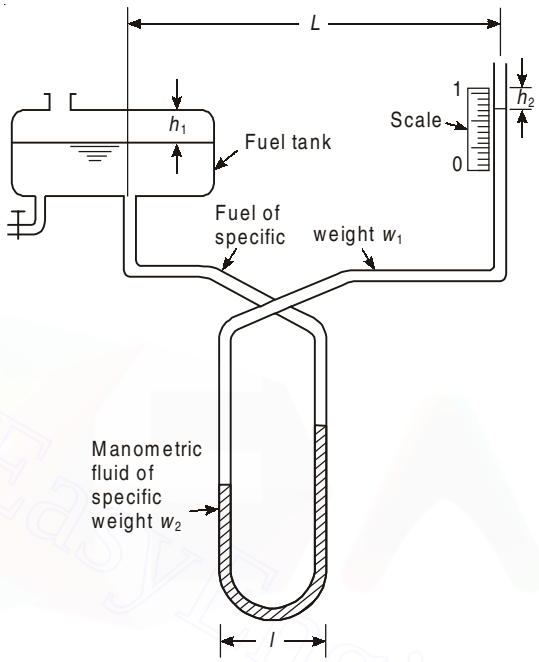


Figure Ex. 2.26

(*a*) *Determine the relation between* h_1, *the distance through which the fuel level is lower in the tank and* h_2, *the distance through which the fuel level falls in the indicating tube.*

(*b*) *For what value of the ratio* $\frac{l}{L}$ *will the reading of the device remains unaltered when the system is given small tilt in the vertical plane*?

Solution

(a) When the fuel tank is full, let the manometric liquid in the limb of the U-tube connected to the fuel tank be at *B* and that in the other limb of the U-tube be at *A* (see Fig. 2.26).

Equating the pressures at point *A* and *A′*, we get

$$w_1X = w_1Y + w_2Z \qquad \ldots(i)$$

When the level of fuel in the tank is reduced by h_1 and that in the indicating tube is reduced by h_2, the level of manometric liquid in the limb of the U-tube connected to the indicating tube will be raised from *A* to *C* by h_2, and that in the limb of the U-tube connected to the fuel tank will be reduced from *B* to *D* by h_2.

Thus equating the pressures at points *C* and *C′*, we get

$$w_1(X - 2h_2) = w_1(Y - h_1 + h_2) + w_2(Z - 2h_2) \qquad \ldots(ii)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

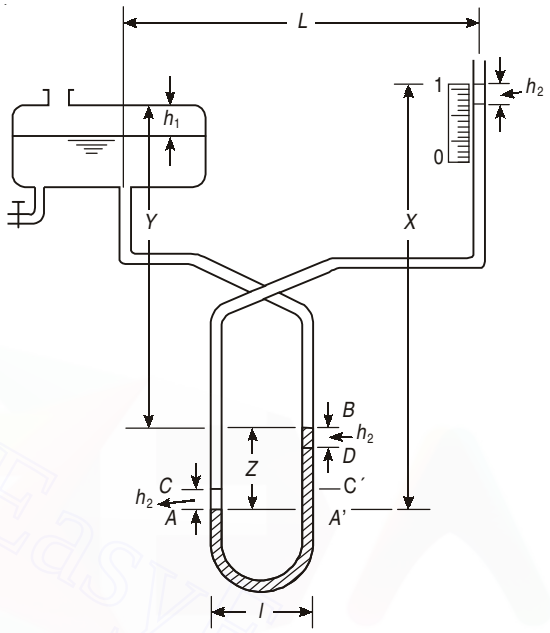


Figure Ex. 2.26

Subtracting Eq. (ii) from (i), we get

$$2w_1h_2 = w_1h_1 - w_1h_2 + 2w_2h_2$$

or
$$\frac{h_1}{h_2} = 1 - 2\left(\frac{w_2}{w_1} - 1\right)$$

which is the required relation between h_1 and h_2.

(b) For the readings of the device to remain unaltered when the system is given a small tilt θ in the vertical plane, sin θ or tan θ may be expressed as

$$\frac{Z}{l} = \frac{(X - Y)}{L}$$

or
$$\frac{l}{L} = \frac{Z}{X - Y}$$

From Eq. (i), we have

$$\frac{Z}{X - Y} = \frac{w_1}{w_2}$$

$\therefore$
$$\frac{l}{L} = \frac{w_1}{w_2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. Pressure or intensity of pressure is defined as the force per unit area.
2. The pressure p at any point in a static mass of fluid (or fluid at rest) varies only in z or vertical direction as indicated by the following equation

$$\frac{dp}{dz} = -w = -\rho g \qquad ...(i)$$

which represents the basic law of hydrostatics.

(i) A liquid may be considered as incompressible fluid for which w or ρ is constant and hence integration of the above equation gives pressure p at any point in a static mass of liquid (or liquid at rest) as

$$p = -wz + C$$

where C is the constant of integration.

(ii) For any point lying in a static mass of liquid (or liquid at rest) at a vertical depth h below the free surface of the liquid, $z = -h$, and at the free surface of the liquid i.e. , at $z = 0$, the pressure is equal to the atmospheric pressure p_a, the above equation gives

$$p = wh + p_a$$

and the pressure in excess of the atmospheric pressure is

$$p = wh$$

(iii) The vertical height of the free surface above any point in a liquid at rest is known as pressure head which may be expressed from the above equation as

$$h = \frac{p}{w}$$

3. For a compressible fluid (gas) since the density ρ or w varies with the pressure p, Eq. (i) can be integrated to determine the pressure p at any ponit only if a relation between p and ρ or w is known. The following two relations between p and ρ or w are considered.

(i) If a static mass of compressible fluid is assumed to undergo isothermal compressible, then $\frac{p}{\rho} = \frac{p_0}{\rho_0}$ = constant.

By introduing this equation in Eq. (i), it is integrated and the resulting equation is combined with the equation of state to obtain the pressure p at a height z in a static mass of compressible fluid undergoing isothermal compression as

$$p = p_0 \exp.\left(-\frac{gz}{RT_0}\right)$$

where

p_0 = absolute pressure of the fluid at initial condition at some reference level ;

z = height of the point above the reference level ;

R = gas constant ; and

T_0 = absolute temperature of the fluid at initial condition, which, however, remains constant during the isothermal compression.

(ii) If a static mass of compressible fluid is assumed to undergo adiabatic compression, then $\frac{p}{\rho^k} = \frac{p_0}{\rho_0^k}$ = constant. By introducing this equation in Eq. (i), it is integrated and the resulting equation is combined with the equation of state to obtain the pressure p at a height z in a static mass of compressible fluid undergoing adiabatic compression as

$$p = p_0\left[1 - \frac{gz}{RT_0}\left(\frac{k-1}{k}\right)\right]^{\frac{k}{k-1}}$$

where k is the adiabtic exponent or adiabatic index, and the other notations are same as indicated earlier.

Since in this case temperature is not constant it is given by

$$T = T_0\left[1 - \frac{gz}{RT_0}\left(\frac{k-1}{k}\right)\right]$$

The rate at which the temperature changes with elevation or height z is known as Temperature Lapse rate λ, which is given by

$$\lambda = \frac{\partial T}{\partial z} = -\frac{g}{R}\left(\frac{k-1}{k}\right)$$

(a) For $k = 1$, temperature lapse rate $\lambda = 0$. This condition pertains to isothermal compression during which the temperature remains constant and hence it does not vary with elevation or height z.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) For $k > 1$, temperature lapse rate λ is negative, which means that temperature decreases with the increase of height z.

4. The Pascal's law states that the pressure at any point in a fluid at rest has the same magnitude in all directions.
5. Fluid pressure may be measured with respect to (i) absolute zero pressure, and (ii) local atmospheric pressure.

   When pressure is measured above absolute zero or complete vacuum it is called absolute pressure, and when pressure is measured either above or below atmospheric pressure it is called gage pressure. If the pressure of a fluid is below atmospheric pressure it is designated as vacuum pressure or suction pressure or negative gage pressure.

   Thus the relation between absolute pressure, atmospheric pressure and gage pressure may be expressed as

$$p_{abs} = p_{atm} + p_{gage}$$
$$p_{abs} = p_{atm} - p_{gage}$$

6. Manometer is a device used for measuring pressure at a point in a fluid.

   Manometers are classified as (a) Simple manometers, and (b) Differential manometers.

(a) Simple manometers are used for measuring pressure at a point in a fluid contained in a pipe or vessel. (b) Differential manometers are used for measuring the difference of pressure between any two points in a fluid contained in a pipe or a vessel or in two different pipes or two different vessels.

7. Micromanometers are used for the measurement of very small pressure differences or for the measurment of pressure differences with very high precision.
8. Mechanical gages are those pressure measuring devices which embody an elastic element which deflect under the action of the applied pressure, and this movement mechanically magnified, operates a pointer moving against a graduated circumferential scale. These gages are generally used for measuring high pressures and where high precision is not required. Bourdon Tube Pressure gage is the most commonly used mechanical gage which may be used for measuring both pressures above atmospheric pressure as well as pressures below atmospheric pressure or vacuum pressures.

PROBLEMS

2.1 Explain the terms—intensity of pressure and pressure head.

2.2 Prove that the pressure is the same in all directions at a point in a static fluid.

2.3 State Pascal's Law and give some examples where this principle is applied.

2.4 Define the terms gage pressure, vacuum pressure and absolute pressure. Indicate their relative positions on a chart.

2.5 Briefly explain the principle employed in the manometers used for the measurement of pressure.

2.6 Differentiate between simple and differential type of manometers.

2.7 Describe with the help of neat sketches different types of manometers.

2.8 State the advantages of mechanical pressure gages over the manometers.

2.9 Explain how vacuum pressure can be measured with the help of a U-tube manometer.

2.10 Describe with a neat sketch a micromanometer used for very precise measurement of small pressure difference between two points.

2.11 Express a pressure intensity of 5 kg (f)/cm^2 in all possible units. Take the barometer reading as 76 cm of mercury.

[*Ans.* Gage units : 5 kg(f)/cm^2 ; 5×10^2 kg(f)/m^2; 50 m of water; 49.05×10^4 N/m^2; 3.68 m of mercury;

Absolute units: 6.034 kg(f)/cm^2; 6.034×10^4 kg(f)/m^2; 59.19×10^4 N/m^2 ; 60.34 m of water ; 4.44 m of mercury ; 5.842 atmospheres]

2.12 Find the depth of a point below free surface in a tank containing oil where the pressure intensity is 9 kg (f)/cm^2. Specific gravity of oil is 0.9. [*Ans.* 100 m]

2.13 Convert a pressure head of 15 m of water to (a) metres of oil of specific gravity 0.750 ; (b) metres of mercury of specific gravity 13.6.

[*Ans.* (a) 20 m; (b) 1.103 m]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

2.14 A U-tube containing mercury has its right limb open to atmosphere. The left limb is full of water and is connected to a pipe containing water under pressure, the centre of which is in level with the free surface of mercury. Find the pressure of the water in the pipe above atmosphere, if the difference of level of mercury in the limbs is 5.08 cm. [*Ans.* 0.064 kg(f)/cm^2]

2.15 The pressure between two points *M* and *N* in a pipe conveying oil of specific gravity 0.9 is measured by an inverted U-tube and the column connected to point *N* stands 1.5 m higher than that at point *M*. A commercial pressure gage attached directly to the pipe at *M* reads 2 kg(f)/cm^2; determine its reading when attached directly to the pipe at *N*.
[*Ans.* 2.135 kg(f)/cm^2]

2.16 A pipe containing water at 172 kN/m^2 pressure is connected by a differential gage to another pipe 1.5 m lower than the first pipe and containing water at high pressure. If the difference in heights of the two mercury columns of the gage is equal to 75 mm, what is the pressure in the lower pipe? Specific gravity of mercury is 13.6. [*Ans.* 196 kN/m^2]

2.17 Two pressure points in a water pipe are connected to a manometer which has the form of an inverted U-tube. The space above the water in the two limbs of the manometers is filled with toluene of specific gravity 0.875. If the difference of level of water columns in the two limbs is equal to 0.12 m what is the corresponding difference of pressure expressed in (a) kg(f)/cm^2; (b) N/m^2.
[*Ans.* (a) 0.0015 kg(f)/cm^2; (b) 147.15 N/m^2]

2.18 A U-tube differential gage is attached to two sections *A* and *B* in a horizontal pipe in which oil of specific gravity 0.8 is flowing. The deflection of the mercury in the gage is 60 cm, the level nearer to *A* being the lower one. Calculate the difference of pressure in kg(f)/cm^2 between the sections *A* and *B*.
[*Ans.* 0.768 kg(f)/cm^2]

2.19 In Fig. P 2.19 the areas of the plunger *A* and cylinder *B* are 40 and 4 000 cm^2 respectively. The weight of cylinder *B* is 4 100 kg(f). The vessel and the connecting passages are filled with oil of specific gravity 0.750. What force *F* is required for equilibrium, if the weight of *A* is neglected ? [*Ans.* 40.85 kg(f)]

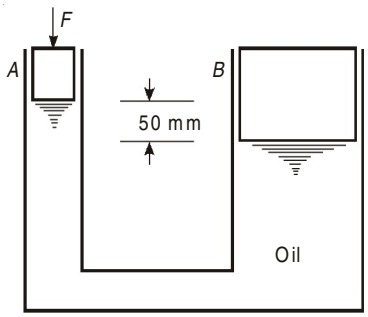


Fig. P. 2.19

2.20 Vessels *A* and *B* contain water under pressures of 274.68 kN/m^2 [2.8 kg(f)/cm^2] and 137.34 kN/m^2 [1.4 kg(f)/cm^2] respectively. What is the deflection of the mercury in the diffrential gage shown in Fig. P. 2.20 ? [*Ans.* 1.27 m]

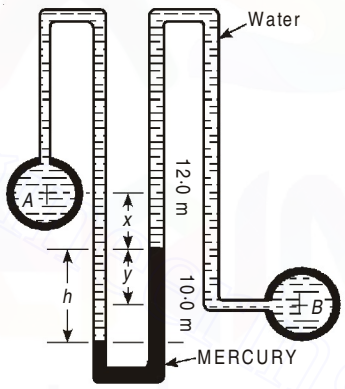


Fig. P. 2.20

2.21 The tank in the accompanying figure contains oil of specific gravity 0.750. Determine the

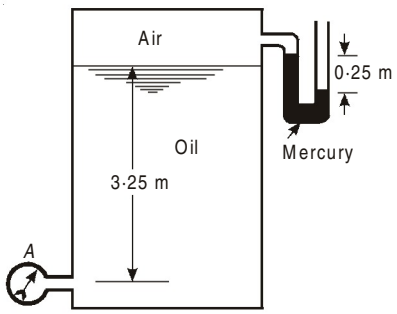


Fig. P. 2.21

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

reading of pressure gage A in (a) kg(f)/cm^2 ; (b) kN/m^2.

[*Ans.* (a) – 0.096 kg(f)/cm^2; (b) – 9.442 kN/m^2]

2.22 In the manometer shown Fig. P. 2.22 the liquid on the left side is carbon-tetrachloride of specific gravity 1.60 and liquid on the right side is mercury. If $(p_A - p_B)$ is 525 kg(f)/m^2 (5 150.25 N/m^2) , find the specific gravity of the liquid X. [*Ans.* 0.75]

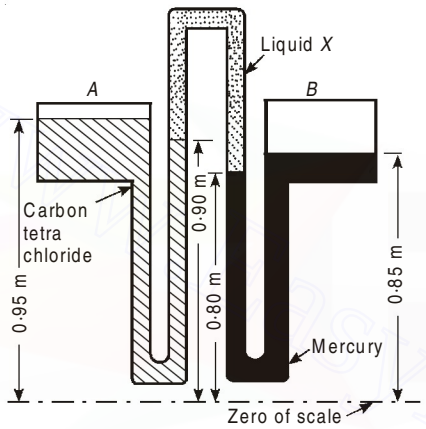


Fig. P. 2.22

2.23 In the left hand tank shown in the Fig. P. 2.23, the air pressure is –0.23 m of mercury. Determine the elevation of the gage liquid in the right hand column at A, if the liquid in the right hand tank is water. [*Ans.* 94.62 m]

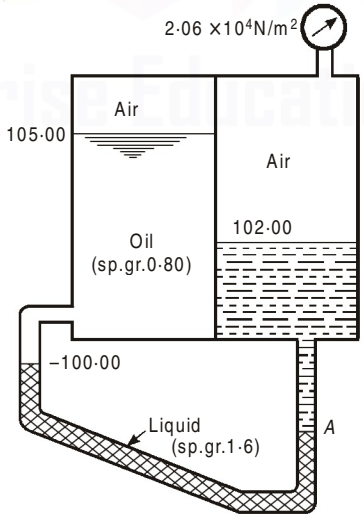


Fig. P. 2.23

2.24 As shown in Fig. P. 2.24, the pipe and connection B are full of oil of specific gravity 0.9 under pressure. If the U-tube contains mercury, find the elevation of point A in metres. [*Ans.* 112.76 m]

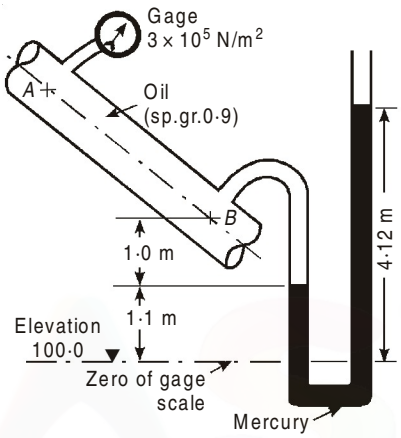


Fig. P. 2.24

2.25 Neglecting the friction between the piston A and the gas tank, find the gage reading at B in metres of water. Assuming gas and air to be of constant specific weight and equal to 5.501 N/m^2 and 12.267 N/m^3 respectively. [*Ans.* 0.101 m]

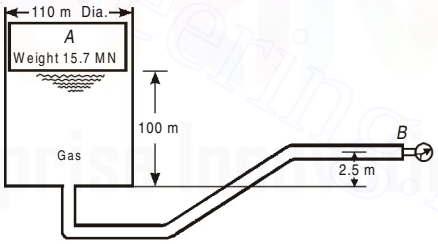


Fig. P. 2.25

2.26 A micro-manometer consists of two cylindrical bulbs A and B each 1000 sq.mm cross-sectional area which are connected by a U-tube with vertical limbs each of 25 sq.mm cross-sectional area. A liquid of specific gravity 1.2 is filled in A and another liquid of specific gravity 0.9 is filled in B, the surface of separation being in the limb attached to B. Find the displacement of the surface of separation when the pressure on the surface in B is greater than that in A by an amount equal to 15 mm head of water. [*Ans.* 42.6 mm]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hydrostatic Forces on Surfaces

Chapter 3

3.1 TOTAL PRESSURE AND CENTRE OF PRESSURE

When a static mass of fluid comes in contact with a surface, either plane or curved, a force is exerted by the fluid on the surface. This force is known as *total pressure.* Since for a fluid at rest no tangential force exists, the total pressure acts in the direction normal to the surface. The point of application of total pressure on the surface is known as *centre of pressure.* As indicated later an engineer is often required to compute the magnitude of total pressure and to locate its point of application in the design of several hydraulic structures.

3.2 TOTAL PRESSURE ON A PLANE SURFACE

3.2.1 Total Pressure on a Horizontal Plane Surface

Consider a plane surface immersed in a static mass of liquid of specific weight w, such that it is held in a horizontal position at a depth h below the free surface of the liquid, as shown in Fig. 3.1. Since every point on the surface is at the same depth below the free surface of the liquid, the pressure intensity is constant over the entire plane surface, being equal to $p = wh$. Thus if A is the total area of the surface then the total pressure on the horizontal surface is

$$P = pA = (wh)\,A = wAh \qquad \ldots(3.1)$$

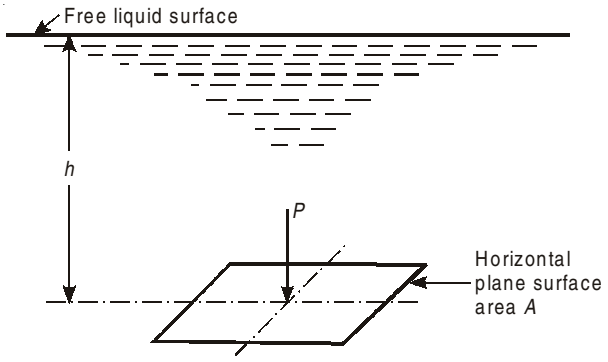


Figure 3.1 *Total pressure on a horizontal plane surface*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The direction of this force is normal to the surface, as such it is acting towards the surface in the vertical downward direction at the centroid of the surface.

3.2.2 Total Pressure on a Vertical Plane Surface

Figure 3.2 shows a plane surface of arbitrary shape and total area A, wholly submerged in a static mass of liquid of specific weight w. The surface is held in a vertical position, such that the centroid of the surface is at a vertical depth of $\bar{x}$ below the free surface of the liquid. It is required to determine the total pressure exerted by the liquid on one face of the plane surface.

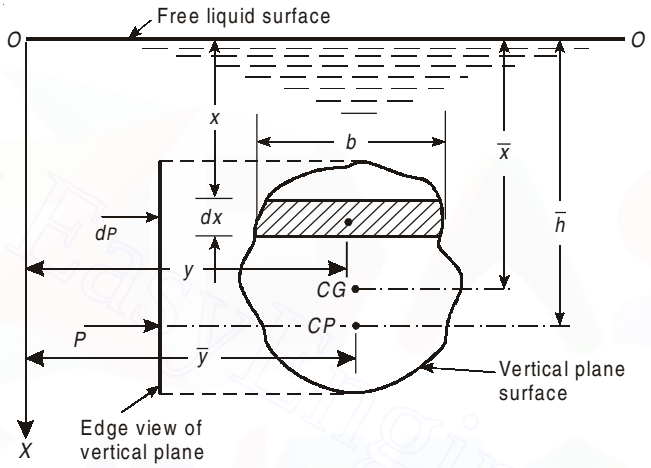


Figure 3.2 *Total pressure on a vertical plane surface*

In this case since the depth of liquid varies from point to point on the surface, the pressure intensity is not constant over the entire surface. As such the total pressure on the surface may be determined by dividing the entire surface into a number of small parallel strips and computing the total pressures on each of these strips. A summation of these total pressures on the small strips will give the total pressure on the entire plane surface.

Consider on the plane surface a horizontal strip of thickness dx and width b lying at a vertical depth x below the free surface of the liquid. Since the thickness of the strip is very small, for this strip the pressure intensity may be assumed to be constant equal to $p = wx$. The area of the strip being $dA = (b \times dx)$, the total pressure on the strip becomes

$$dP = pdA = wx(bdx) \qquad \ldots(3.2)$$

$\therefore$ Total pressure on the entire plane surface is

$$P = \int dP = w\int x(bdx).$$

But $\int x\,(bdx)$ represents the sum of the first moments of the areas of the strips about an axis OO, (which is obtained by the intersection of the free surface of the liquid with the vertical plane in which the plane surface is lying) which from the basic principle of mechanics is equal to the product of the area A and the distance $\bar{x}$ of the centroid of the surface area from the same axis OO. That is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\int x\ (bdx) = A\bar{x}$$

$$P = wA\bar{x} \qquad ...(3.3)$$

Equation (3.3) thus represents a general expression for total pressure exerted by a liquid on a plane surface. Since $w\bar{x}$ is the intensity of pressure at the centroid of the surface area, it can be stated that the total pressure on a plane surface is equal to the product of the area of the surface and the intensity of pressure at the centroid of the area.

Total pressure on a horizontal plane surface can also be determined by Eq. (3.3), since in this case $\bar{x} = h$.

3.2.3 Centre of Pressure for Vertical Plane Surface

As stated earlier, the point of application of the total pressure on a plane surface is known as centre of pressure. For a plane surface immersed horizontally since the pressure intensity is uniform the total pressure would pass through the centroid of the area i.e., in this case the centroid of the area and the centre of pressure coincide with each other. However, for a plane surface immersed vertically the centre of pressure does not coincide with the centroid of the area. Since the pressure intensity increases with the increase in the depth of liquid, the centre of pressure for a vertically immersed plane surface lies below the centroid of the surface area. The position of the centre of pressure for a vertically immersed plane surface may be determined as explained below.

As shown in Fig. 3.2 let $\bar{h}$ be the vertical depth of the centre of pressure for the plane surface immersed vertically. Then the moment of the total pressure P about axis OO is equal to $(P\bar{h})$.

From Eq. 3.2 the total pressure on the strip shown in Fig. 3.2 is, $dP = wx\ (bdx)$ and its moment about axis OO is

$$(dP)\ x = wx^2\ (bdx)$$

Likewise, by considering a number of small strips and summing the moments of the total pressure on these strips about axis OO, the sum of the moments of the total pressures on all the strips becomes

$$\int (dP)\ x = w \int x^2 (bdx).$$

By using the "Principle of Moments", which states that the moment of the resultant of a system of forces about an axis is equal to the sum of the moments of the components about the same axis, the moment of the total pressure about axis OO is

$$P\bar{h} = w \int x^2\ (bdx) \qquad \ldots(3.4)$$

In Eq. 3.4, $\int x^2\ (bdx)$ represents the sum of the second moment of the areas of the strip about axis OO, which is equal to the moment of inertia I_0 of the plane surface about axis OO. That is

$$I_0 = \int x^2\ (bdx) \qquad \ldots(3.5)$$

Introducing Eq. 3.5 in Eq. 3.4 and solving for $\bar{h}$,

$$\bar{h} = \frac{wI_0}{P}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting for the total pressure from Eq. 3.3, we obtain

$$\bar{h} = \frac{I_0}{A\bar{x}} \qquad \text{...[3.5 (a)]}$$

Further from the "parallel axes theorem" for the moment of inertia,

$$I_o = I_G + A\bar{x}^2 \qquad \text{...(3.6)}$$

where I_G is the moment of inertia of the area about an axis passing through the centroid of the area and parallel to axis *OO*.

Introducing Eq. 3.6 in Eq. 3.5 (a), it becomes

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}} \qquad \text{...(3.7)}$$

Equation 3.7 gives the position of the centre of pressure on a plane surface immersed vertically in a static mass of liquid. Since for any plane surface the factor $(I_G/A\,\bar{x})$ is always positive, Eq. 3.7 indicates that $\bar{h} > \bar{x}$, i.e., the centre of pressure is always below the centroid of the area.

Further it is seen that deeper the surface is submerged, i.e., the greater is the value of $\bar{x}$, the factor $(I_G/A\,\bar{x})$ becomes smaller and the centre of pressure comes closer to the centroid of the plane surface. This is so because, as the pressure becomes greater with increasing depth, its variation over a given area becomes smaller in proportion, thereby making the distribution of pressure more uniform. Thus where the variation of pressure is negligible the centre of pressure may be taken as approximately at the centroid. This is justifiable in liquids, only if the depth is very large and the area is small, and in gases because in them the pressure changes very little with depth.

The lateral location of the centre of pressure can also be readily determined by taking moments about any convenient axis in the vertical direction . Thus if *OX* is the reference axis (as shown in Fig. 3.2) in the vertical direction, lying in the same vertical plane in which the plane surface is lying, from which $\bar{y}$ is the distance of the centre of pressure of the plane surface and y is the distance of the centre of pressure of the small strip on the plane surface, then the distance $\bar{y}$ may be determined by taking the moments about axis *OX*. The moment of dP about axis *OX* is $(dP)\,y = wx\,(bdx)\,y$ and the sum of the moments of the total pressure on all such strips considered on the plane surface is

$$\int (dP)y = w\int xy\,(bdx)$$

which by the principle of moments is equal to the moment of P about axis *OX*, i.e., $(P\,\bar{y})$. Thus

$$(P\,\bar{y}) = w\int xy\,(bdx)$$

or

$$(wA\,\bar{x})\,\bar{y} = w\int xy\,(bdx)$$

$$\therefore \quad \bar{y} = \frac{\int xy(bdx)}{A\bar{x}} \qquad \text{...(3.8)}$$

The centre of pressure of the plane surface immersed vertically in a static mass of liquid is therefore, at a vertical depth $\bar{h}$ (given by Eq. 3.7) below the free surface of the liquid and at a distance $\bar{y}$ (given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Eq. 3.8) from an assumed vertical reference axis *OX*. If the plane surface has a vertical axis of symmetry passing through its centroid, then this axis may be taken as the reference axis *OX*, in which case $\int xy$ $(bdx) = 0$, and the centre of pressure lies on the axis of symmetry at a vertical depth $\bar{h}$ below the free surface of the liquid.

3.2.4 Total Pressure on Inclined Plane Surface

Consider a plane surface of arbitrary shape and total area *A*, wholly submerged in a static mass of liquid of specific weight *w*. The surface is held inclined such that the plane of the surface makes an angle θ with the horizontal as shown in Fig. 3.3. The intersection of this plane with the free surface of the liquid is represented by axis *OO*, which is normal to the plane of the paper.

Let $\bar{x}$ be the vertical depth of the centroid of the plane surface below the free surface of the liquid, and the inclined distance of the centroid from axis *OO* measured along the inclined plane be $\bar{y}$.

Consider on the plane surface, a small strip of area *dA* lying at a vertical depth of *x* and its distance from axis *OO* being *y*. For this strip the pressure intensity may be assumed to be constant equal to $p = wx$.

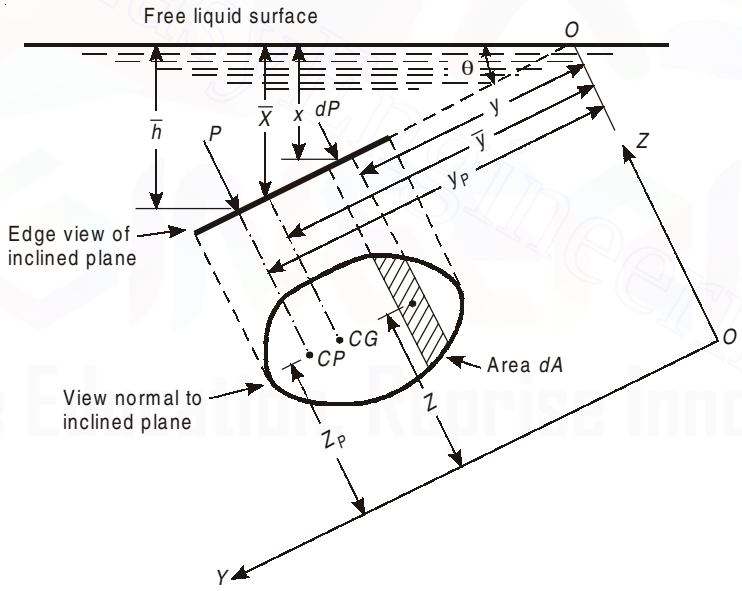


Figure 3.3 *Total pressure on inclined plane surface*

∴ Total pressure on the strip is

$$dP = wx\,(dA)$$

Since $x = y \sin\theta$

$$dP = w\,(y \sin\theta)\,(dA)$$

By integrating the above expression the total pressure on the entire surface is obtained as

$$P = (w \sin\theta) \int y\,(dA)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again $\int y\,dA$ represents the sum of the first moments of the areas of the strips about axis OO, which is equal to the product of the area A and the inclined distance of the centroid of the surface area from axis OO. That is

$$\int y\,dA = A\bar{y}$$

$$\therefore \quad P = wA(\bar{y}\sin\theta) \qquad ...(3.9)$$

But $\quad \bar{x} = \bar{y}\sin\theta$

$$\therefore \quad P = wA\bar{x} \qquad ...(3.10)$$

Equation 3.10 is same as Eq. 3.3, thereby indicating that for a plane surface wholly submerged in a static mass of liquid and held either vertical or inclined, the total pressure is equal to the product of the pressure intensity at the centroid of the area and the area of the plane surface.

3.2.5 Centre of Pressure for Inclined Plane Surface

As shown in Fig. 3.3, let $\bar{h}$ be the vertical depth of the centre of pressure for the inclined plane surface below the free surface of the liquid and its inclined distance from the axis OO be y_p.

Total pressure on the strip shown in Fig. 3.3 is $[w(y\sin\theta)\,dA\,]$ and its moment about axis OO is

$$(dP)\,y = (w\sin\theta\,y^2\,dA)$$

By summing the moments of the total pressures on such small strips about axis OO and using the "Principle of Moments" stated earlier in Sec. 3.2.3, we get

$$Py_p = w\sin\theta\int y^2\,dA \qquad ...(3.11)$$

Again in Eq. 3.11, $\int y^2\,dA$ represents the sum of the second moments of the areas of the strips about axis OO, which is equal to the moment of inertia I_0 of the plane surface about axis OO. That is,

$$I_0 = \int y^2\,dA \qquad ...(3.12)$$

Introducing Eq. 3.12 in Eq. 3.11 and solving for y_p, we obtain

$$y_p = \frac{w\sin\theta I_0}{P} \qquad ...(3.13)$$

Further from the "parallel axes theorem" for the moments of inertia,

$$I_0 = I_G + A\bar{y}^2 \qquad ...(3.14)$$

where I_G is the moment of inertia of the area about an axis passing through the centroid of the area and parallel to axis OO.

Introducing Eq. 3.14 in Eq. 3.13 and substituting for the total pressure from Eq. 3.9, we get

$$y_p = \frac{w\sin\theta(I_G + A\bar{y}^2)}{wA(\bar{y}\sin\theta)}$$

or $$y_p = \bar{y} + \frac{I_G}{A\bar{y}} \qquad ...(3.15)$$

But $$y_p = \frac{\bar{h}}{\sin\theta} \text{ and } \bar{y} = \frac{\bar{x}}{\sin\theta}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad \bar{h} = \bar{x} + \frac{I_G \sin^2 \theta}{A\bar{x}} \qquad ...(3.16)$$

Equation 3.15 gives the vertical depth of centre of pressure below free surface of liquid, for an inclined plane surface, wholly immersed in a static mass of liquid. Obviously for $\theta = 90°$, Eq. 3.16 becomes same as Eq. 3.7 which is applicable for vertically immersed plane surfaces.

The lateral location of the centre of pressure in this case may also be determined in the same manner as in the case of vertically immersed plane surface, by considering a reference axis *OY* perpendicular to the axis *OO* (or *OZ*) (as shown in Fig. 3.3) and lying in the same plane in which the inclined plane surface is lying. From the axis *OY* let z_p be the distance of the centre of pressure of the plane surface and z be the distance of the centre of pressure of the small strip on the plane surface. The moment of the total pressure dP on the small strip about axis *OY* is $(dP)\, y = wx(dA)\, z$ and the sum of the moments of the total pressure on all such strips considered on the plane surface is

$$\int (dP)\, z = w \int xz\, (dA)$$

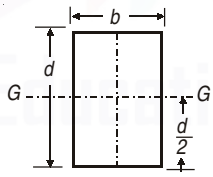
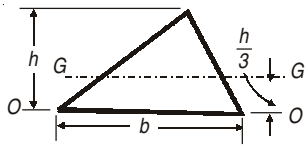
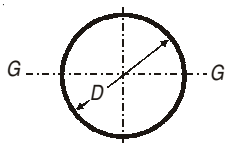
which by the principle of moments is equal to the moment of P about axis *OY*, i.e., (Pz_p). Thus

$$(Pz_p) = w \int xz\, (dA)$$

or
$$(wA\bar{x})\, z_p = w \int xz\, (dA)$$

$$\therefore \quad z_p = \frac{\int xz(dA)}{A\bar{x}} \qquad ...(3.17)$$

TABLE 3.1 *Moment of Inertia and other Geometric Properties of Plane Surfaces*

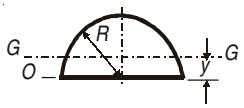
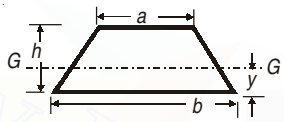
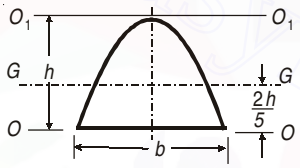
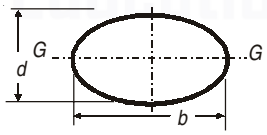
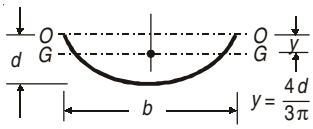
Plane	*Surface*	*Area*	*Moment of inertia of area about axis GG through centroid*
Rectangle		bd	$\frac{bd^3}{12}$
Triangle		$\frac{bh}{2}$	$\frac{bh^3}{36}\left[I_{o-o} = \frac{bh^3}{12}\right]$
Circle		$\frac{\pi D^2}{4}$	$\frac{\pi D^4}{64}$

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Contd.

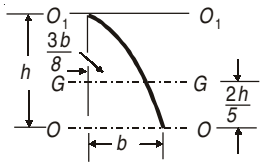
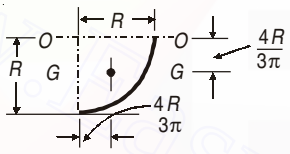
Plane	*Surface*	*Area*	*Moment of inertia of area about axis GG through centroid*
Semi-circle	 $y = \frac{4R}{3\pi} = 0.425R$	$\frac{\pi R^2}{2}$	$\left[\frac{\pi}{8} - \frac{8}{9\pi}\right]R^4 = 0.11R^4$ $\left[I_{o-o} = \frac{\pi R^4}{8}\right]$
Trapezium	 $y = \left(\frac{2a+b}{a+b} \times \frac{h}{3}\right)$	$\frac{1}{2}(a+b)h$	$\left[\frac{a^2+4ab+b^2}{36(a+b)}\right]h^3$
Parabola		$\frac{2}{3}bh$	$\frac{8}{175}bh^3$ $\left[I_{o_1-o_1} = \frac{2bh^3}{7}\right]$ $\left[I_{o-o} = \frac{16bh^3}{105}\right]$
Ellipse		$\frac{\pi}{4}bd$	$\frac{\pi bd^3}{64}$
Semi-ellipse		$\frac{\pi bd}{4}$	$\left[\frac{\pi}{16} - \frac{4}{9\pi}\right]bd^3 = 0.055bd^3$ $\left[I_{o-o} = \frac{\pi bd^3}{16}\right]$

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Contd.

Plane	*Surface*	*Area*	*Moment of inertia of area about axis GG through centroid*
Semi-parabola		$\frac{2}{3}bh$	$\frac{8}{175}bh^3$ $\left[I_{o-o} = \frac{2bh^3}{7}\right]$
Quarter circle		$\frac{\pi R^2}{4}$	$\left[\frac{\pi}{16} - \frac{4}{9\pi}\right]R^4 = 0.055R^4$ $\left[I_{o-o} = \frac{\pi R^4}{16}\right]$

The centre of pressure of the plane surface held immersed in an inclined position in a static mass of liquid is therefore, at a vertical depth $\bar{h}$ (given by Eq. 3.15) below the free surface of the liquid, at a distance y_p (given by Eq. 3.15) from axis *OO* (or *OZ*) and at a distance z_p (given by Eq. 3.17) from axis *OY*. Further in this case also if the plane surface has an axis of symmetry parallel to axis *OY* and passing through the centroid of the plane surface then this axis may be taken as the reference axis *OY*, in which case $\int xz\,(dA) = 0$, and the centre of pressure lies on the axis of symmetry at a vertical depth $\bar{h}$ (given by Eq. 3.15) below the free surface of the liquid.

Table 3.1 gives the moments of inertia and other geometric properties of different plane surfaces which are commonly met in actual practice.

It is obvious that for a plane surface shown in Fig. 3.2 or 3.3, Eq. 3.3 or 3.10 gives the total pressure on one face only. However for a plane surface of negligible thickness the total pressure on one face would exactly balance the total pressure on the other if both the faces were in contact with the liquid. But, as indicated later, in most cases of practical interest, either total pressures are required to be computed only on one face of the surface or the total pressures exerted on the two faces of the plane surface are not the same.

Moreover in the computation of total pressure, only gage pressure has been considered. This is so because the effect of atmospheric pressure at the free surface of liquid is to provide a uniform addition to the gage pressure throughout the liquid, and therefore to the force on any surface in contact with the liquid. Normally atmospheric pressure also provides a uniform force on the other face of the plane, and so it has no effect on either the magnitude or position of the net total pressure exerted on the surface.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

3.3 PRESSURE DIAGRAM

Total pressure as well as centre of pressure for a plane surface wholly submerged in a static mass of liquid, either vertically or inclined, may also be determined by drawing a pressure diagram. A pressure diagram is a graphical representation of the variation of the pressure intensity over a surface. Such a diagram may be prepared by plotting to some convenient scale the pressure intensities at various points on the surface. Since pressure at any point acts in the direction normal to the surface, the pressure intensities at various points on the surface are plotted normal to the surface. Figure 3.4 shows typical pressure diagrams for horizontal, vertical and inclined plane surfaces.

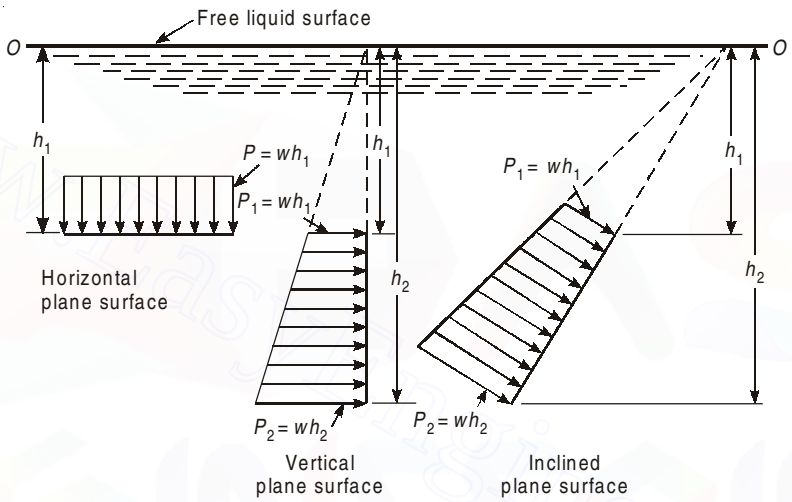


Figure 3.4 *Pressure diagrams for horizontal, vertical and inclined plane surfaces*

As an example consider a rectangular plane surface of width l and depth b, held vertically submerged in a static mass of liquid of specific weight w, as shown in Figure 3.5. Let the top and bottom edges of the plane surface be at vertical depths of h_1 and h_2 respectively below the free surface of the liquid. Thus for every point near the top edge of the plane surface the pressure intensity is

$$p_1 = wh_1$$

Similarly for every point near the bottom edge of the plane surface the pressure intensity is

$$p_2 = wh_2$$

Since the pressure intensity at any point varies linearly with the depth of the point below the free surface of the liquid, the pressure diagram may be drawn as shown in Fig. 3.5, which will be trapezium with the length of the top edge equal to wh_1, the length of the bottom edge equal to wh_2 and its height equal to b, the depth of the rectangular plane surface. In the same manner if the pressure diagrams are drawn for all the vertical sections of the plane surface, a trapezoidal prism will be developed as shown in Fig. 3.5. The volume of the prism gives the total pressure on the plane surface, which in the present case is

$$P = \left(\frac{wh_1 + wh_2}{2}\right) b \times l$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$P = \frac{w(h_1 + h_2)}{2}(b \times l) \quad \text{...(3.18)}$$

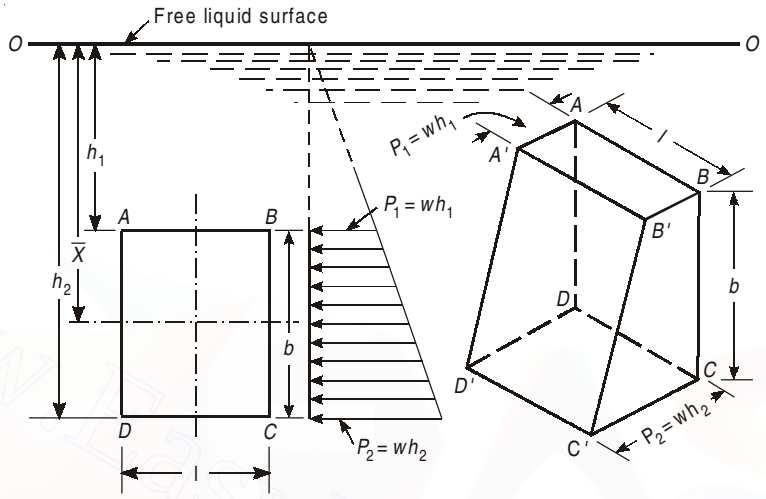


Figure 3.5 *Pressure diagram for a vertical rectangular plane surface*

Further the centre of pressure is located at the centroid of the prism so developed. In the present case on account of symmetry the centroid of the trapezoidal prism will lie along the vertical axis of the rectangular plane surface and its depth below the free surface of the liquid is equal to the depth of the centroid of end face *BCC′B′* of the trapezoidal prism. Accordingly the depth of the centre of pressure is

$$\bar{h} = h_1 + \frac{(2wh_2 + wh_1)}{(wh_2 + wh_1)}\frac{b}{3}$$

or $$\bar{h} = h_1 + \frac{(2h_2 + h_1)}{(h_2 + h_1)}\frac{b}{3} \quad \text{...(3.19)}$$

The results obtained above and represented by Eqs 3.18 and 3.19 may also be obtained by using Eqs 3.3 and 3.7 and the same may therefore be verified. This method is quite useful for the determination of the total pressure and the centre of pressure when a plane surface is partly in contact with one liquid and partly with another liquid, both of which are static and immiscible.

3.4 TOTAL PRESSURE ON CURVED SURFACE

Consider a curved surface wholly submerged in a static mass of liquid of specific weight w. As shown in Fig. 3.6, *ABC* is the trace of the curved surface which extends in the direction normal to the plane of the paper. At any point on the curved surface also the pressure acts normal to the surface. Thus if dA is the area of a small element of the curved surface lying at a vertical depth h below the free surface of the liquid, then the total pressure on the elementary area is

$$dP = (pdA) = (wh)\,dA \quad \text{...(3.20)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which is acting in the direction normal to the surface.

Further integration of equation 3.20 would provide the total pressure on the curved surface and hence

$$P = \int p\, dA \qquad ...(3.21)$$

However, in the case of a curved surface the direction of the total pressure on the elementary areas varies from point to point. As such the determination of the total pressure on a curved surface by evaluating the integral in equation 3.21 is rather impossible. The problem can however be simplified by resolving the total pressure P, on the curved surface into the horizontal and vertical components P_H and P_v. As indicated below, the determination of the components P_H and P_V is simple, from which the total pressure P can be determined.

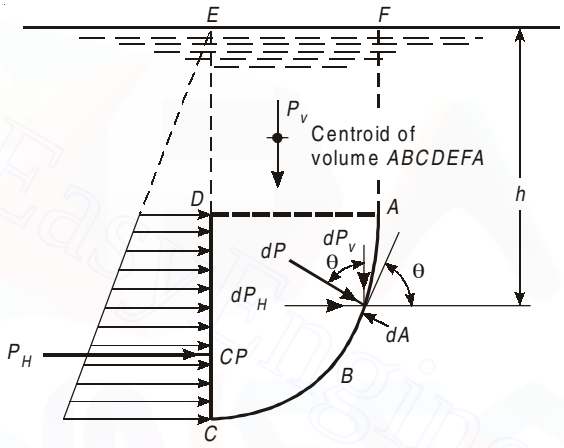


Figure 3.6 *Total pressure on a curved surface*

The total pressure dP acting on the elementary area dA can be resolved into its horizontal and vertical components

$$dP_H = dP \sin\theta = pdA \sin\theta$$

and $$dP_V = dP \cos\theta = pdA \cos\theta$$

where θ is the inclination of the elementary area with the horizontal. Substituting $p = wh$ in the above expressions and then integrating the same to evaluate P_H and P_v, we have

$$P_H = \int dP_H = w \int h\, dA \sin\theta$$

and $$P_V = \int dPV = w \int h\, dA \cos\theta$$

In the above expression ($dA \sin\theta$) is the vertical projection of the elementary area dA and ($dA \cos\theta$) is the horizontal projection of the elementary area dA. Thus [(wh) $dA \sin\theta$] represents the total pressure on the vertical projection of the elementary area dA, and therefore [$\int (wh)\, dA \sin\theta$] represents the total pressure on the projected area of the curved surface on a vertical plane. As such,

P_H = Total pressure on the projected area of the curved surface on the vertical plane, the trace of which is represented by CD in Fig. 3.6.

Since P_H is equal to the total pressure exerted by the liquid on an imaginary vertically immersed plane surface which is the vertical projection of the curved surface, it will act at the centre of pressure of the plane surface.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further [$(wh)\, dA \cos\theta$] represents the total pressure on the horizontal projection of the elementary area dA, and it is equal to the weight of the liquid contained in the portion extending above the elementary area upto the free surface. Therefore [$\int (wh)\, dA \cos\theta$] represents the weight of the liquid lying above the curved surface in the portion *ABCDEFA* shown in Fig. 3.6. As such,

P_V = The weight of the liquid contained in the portion extending vertically above the curved surface upto the free surface of the liquid.

Obviously P_V will act through the centre of gravity of the volume of liquid contained in the portion extending above the curved surface upto the free surface of the liquid (represented by the profile *ABCDEFA* in the present case).

In some cases it is the underside of a curved surface, which is subjected to the hydrostatic pressure, and the upper side of the curved surface is not subjected to the hydrostatic pressure. The vertical component of the total pressure on the curved surface then acts upwards and equals the weight of an imaginary volume of liquid extending vertically from the curved surface upto the level of the free surface. This is so because, if the imaginary liquid was in fact present, pressures at the two sides of the curved surface would be identical and the net force reduced to zero. However, the horizontal component of the total pressure in such cases may be determined in the same manner as indicated earlier.

If the two components of the total pressure P_H and P_V lie in the same plane, (which will be so in the case of a curved surface which is regular and symmetrical) then these may be combined into a single resultant force (total pressure) by parallelogram of forces as,

$$P = \sqrt{P_H^2 + P_V^2} \qquad ...(3.22)$$

The direction of the resultant force P is given by

$$\theta = \tan^{-1}\left(\frac{P_V}{P_H}\right) \qquad ...(3.23)$$

where θ is the angle made by the resultant force P with the horizontal. The point of application of the resultant force P on the curved surface may then be determined by extending the line of action of force P to meet the surface.

However, if the curved surface is irregular and not symmetrical then the two components of the total pressure may not lie in the same plane. In such a case there is no single resultant force on the surface.

Sometimes a free surface of fluid may not exist. Such a case may arise when any surface (plane or curved) is subjected to pressure force by a fluid under pressure in a closed chamber. In such a case an imaginary free surface may be assumed to be located at a height (p/w) above any point at which the pressure intensity is p which is known, and w is the specific weight of the imaginary fluid. The specific weight of the imaginary fluid must, of course, be the same as that of the actual fluid so that the variation of pressure over the surface is truly represented. The total pressure and the centre of pressure for the surface may then be determined by adopting the methods discussed earlier.

3.5 PRACTICAL APPLICATIONS OF TOTAL PRESSURE AND CENTRE OF PRESSURE

In practice there exist several hydraulic structures which are subjected to hydrostatic pressure forces. In the design of these structures it is therefore necessary to compute the magnitude of these forces and

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

to locate their points of application on the structures. Some of the common types of such structures are noted below:

(a) Dams (b) Gates (c) Tanks

(a) Dams. A dam is a concrete or masonry structure constructed across a river in order to check the flow of water and impound it in the reservoir formed on the upstream side, for the purpose of irrigation and power generation. Since the upstream face of a dam is always in contact with a static mass of water of great depth, it is subjected to a pressure force, which it should be able to resist safely. As shown in Fig. 3.7, if H is the depth of water stored on the upstream side of a dam, then the total pressure exerted on the upstream face for 1 m length of the dam may be computed by using Eq. 3.3 as

$$p = wA\bar{x} = w(H \times 1) \times \frac{H}{2} = \frac{wH^2}{2}$$

where w is the specific weight of water. It will act normal to the upstream face at the centre of pressure the depth of which below the surface of water in the reservoir may be determined by using Eq. 3.7 as

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}} = \frac{H}{2} + \frac{\frac{1}{12} \times 1 \times H^3}{(H \times 1) \times \frac{H}{2}} = \frac{2}{3}H$$

i.e., the total pressure exerted on upstream face of a dam acts at a depth of $\left(\frac{2}{3}H\right)$ below the water surface or at a height of $\left(\frac{H}{3}\right)$ above the base of the dam.

Obviously the pressure force exerted on a dam tends to overturn it, which is however resisted by the weight of the dam W, acting vertically downwards at the centroid of the dam section. The water

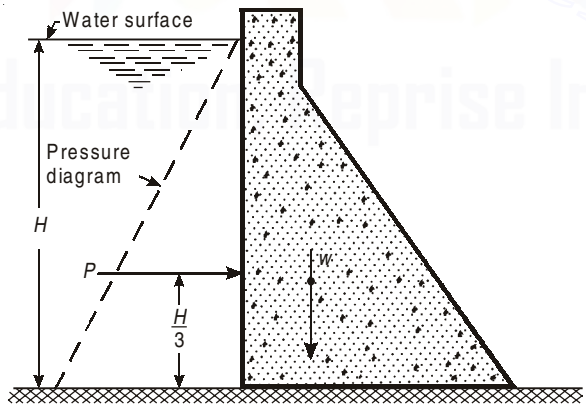


Figure 3.7 *Typical section of a concrete dam*

pressure and the self weight of the dam are the two main forces acting on a dam. But in addition to these two forces, there are some more forces such as uplift pressure, earthquake forces, wind, and

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

wave pressures, silt and ice pressures etc., which are to be considered in the design of dams. As indicated in the Illustrative Example 3.18, the maximum and minimum stresses in the base of the dam may be computed from the forces which act on the dam.

(b) Gates. In several hydraulic structures, openings are required to be provided in order to carry water from the place of its storage to the place of its utilisation for various purposes. The flow of water through such openings, called sluices, is controlled by means of gates which are known as sluice gates. A *sluice gate* may have water either on one side of the gate or on both sides of the gate. Accordingly it will be subjected to water pressure on one side of the gate only or on both sides of the gate.

Figure 3.8 shows a sluice gate of area A which is subjected to water pressure on both the sides. Let H_1 and H_2 be the depths of water above the lower edge of the gate on its either side. Now if h is the height of the gate, then using Eq. 3.3 the total pressures P_1 and P_2 exerted normal to the gate on its either side are obtained as

$$P_1 = wA\bar{x}_1 = wA\left(H_1 - \frac{h}{2}\right)$$

and

$$P_2 = wA\bar{x}_2 = wA\left(H_2 - \frac{h}{2}\right)$$

$\therefore$ Resultant pressure force on the gate is

$$P = (P_1 - P_2)$$

which will be parallel to P_1 and P_2.

The total pressures P_1 and P_2 act at the centres of pressure on either side of the gate, the depth of which below the water surface may be determined by using Eq. 3.7. The position of the point of application of the resultant pressure force P may then be obtained by taking the moments of the forces P, P_1 and P_2 about any point say top or bottom of the gate.

In some cases inclined sluice gates may have to be provided in which case the total pressure exerted on the gate may be determined by treating it as an inclined plane surface in contact with water.

Another type of gates which are used to change the water level in a canal or a river are known as *lock gates.* The water level is required to be raised or lowered in a canal or a river used for navigation, at a section where the bed of the canal or the river has a vertical fall. At such a section of a canal or a river, in order to facilitate the transfer of a boat from the upper water level to the lower one or *vice versa*, a chamber known as lock is constructed by providing two pairs of lock gates. If a boat is to be transferred from the upper water level to the lower water level, the lock is filled up through the openings provided in the upstream pair of lock gates and keeping the similar openings in the downstream pair of lock gates closed. When the level of water in the lock becomes equal to the upper water level, the upstream gates are opened and the boat is transferred to the lock. The upstream gates as well as the openings through these gates are then closed but

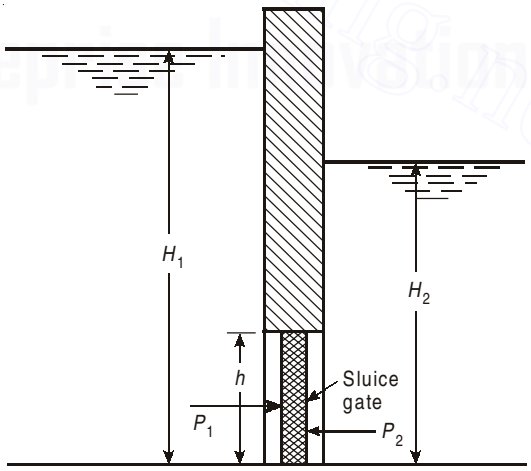


Figure 3.8 *Total pressure on a sluice gate*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the openings through the downstream gates are opened, so that the lock is gradually emptied. When the level of water in the lock becomes equal to the lower water level, the downstream gates are opened and the boat is then transferred from the lock to the downstream side for its onward movement downwards. A boat can also be transferred from the lower to higher side by adopting a procedure reverse of the above procedure.

Figure 3.9 shows plan and elevation of a single pair of lock gates. *LM* and *MN* are two gates symmetrically placed such that the two gates are closely butting with each other at *M*. Each of the gates is carried on two hinges fixed on their top and bottom at the ends *L* and *N*. Under the action of water pressure the gates remain tightly closed at *M*. It is required to determine the magnitudes of the forces on the hinges due to water pressure exerted on the gates.

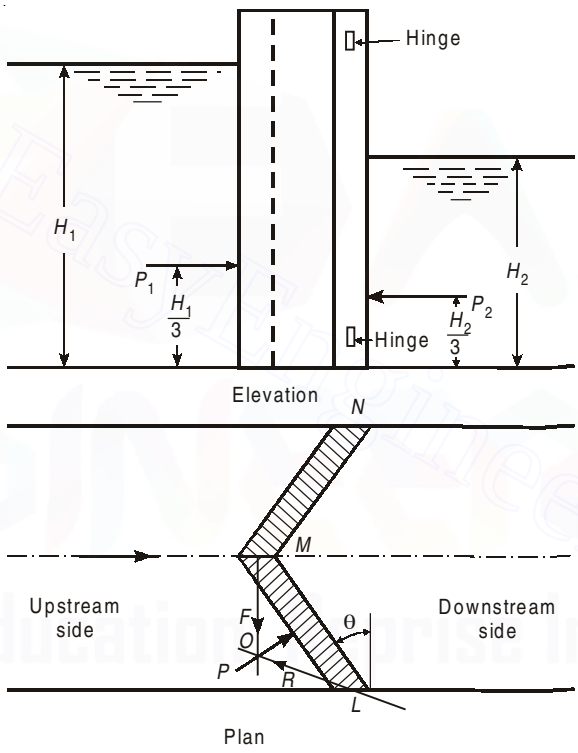


Figure 3.9 *Plan and elevation of a pair of lock gates*

Let H_1 and H_2 be the heights of water on upstream and downstream sides of the gates respectively as shown in Fig. 3.9. Considering one of the gates *LM*, if its width is *l*, the resultant water pressure acting on the gate is

$$P = (P_1 - P_2)$$

where P_1 is the total pressure on the upstream face of the gate given by

$$P_1 = w \times (l \times H_1) \times \frac{H_1}{2} = \frac{wlH_1^2}{2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

acting at centre of pressure which is at a depth of $\left(\frac{2}{3}H_1\right)$ below the water surface, and P_2 is the total pressure on the downstream face of the gate given by

$$P_2 = w \times (l \times H_2) \times \frac{H_2}{2} = \frac{wlH_2^2}{2}$$

acting at centre of pressure which is at a depth of $\left(\frac{2}{3}H_2\right)$ below the water surface.

On account of symmetry the pressures P_1, P_2 and the resultant pressure P will act along the vertical axis of the gate. The height of the point of application of the resultant pressure P may be determined by taking the moments of P, P_1 and P_2 about the bottom of the gate.

The other force acting on the gate LM is the reaction F, between the butting edges of the two gates at M, which acts normal to the surface of contact of the two gates.

Let the two forces P and F intersect at O.

In addition to forces P and F there is one more force acting on the gate LM, which is the resultant reaction R, of the hinges. Since the gate is in equilibrium under the action of the three forces F, P and R, all the three forces must intersect at the same point. Thus hinge reaction R will also pass through point O, at which two forces P and F intersect. It is however assumed that three forces P, F and R are coplanar, that is the forces F and R act at the same height above the bottom of the gate as the force P. The forces F and R may then be evaluated by applying the conditions of static equilibrium.

By symmetry ΔLMO is an isosceles triangle, so

$$\angle LMO = \angle MLO = \theta$$

Now resolving all the forces along the gate LM and equating the sum to zero, we obtain

$$F \cos\theta = R \cos\theta$$

or
$$F = R \qquad ...(3.24)$$

Also resolving all the forces along the direction normal to the gate LM and equating the sum to zero, we obtain

$$P = F \sin\theta + R \sin\theta$$

or
$$P = (F + R) \sin\theta \qquad ...(3.25)$$

Introducing Eq. 3.24 in Eq. 3.25, we obtain

$$R = \frac{P}{2\sin\theta} \qquad ...(3.26)$$

and
$$F = \frac{P}{2\sin\theta} \qquad ...(3.27)$$

In Eqs 3.26 and 3.27 the resultant water pressure P is already known and the value of θ can be determined if the angle between the two gates is known. Thus the two unknown forces R and F can be determined.

Further if R_T and R_B represent the individual hinge reactions at the top and the bottom hinges respectively, then

$$R_T + R_B = R \qquad ...(3.28)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The magnitudes of the individual hinge reactions R_T and R_B may be determined by developing one more equation by taking the moments of R, R_T and R_B about the bottom edge of the gate, and using the 'Principle of Moments'.

In addition to the various types of gates discussed above there are several other types of gates such as roller gates, tainter gates etc., used for regulating the flow of excess water from the reservoir formed on the upstream side of a dam, through a portion of the dam known as spillway. These gates are also subjected to pressure forces due to water which may be computed by using the equations derived earlier, and as indicated in some of the illustrative examples.

(c) Tanks. Tanks are constructed either overhead or on the ground surface in order to store water or other liquids. The sides of these tanks are subjected to forces due to liquid pressure. These forces as well as their points of application may also be obtained by using the expressions derived earlier.

ILLUSTRATIVE EXAMPLES

Example 3.1. *A 3.6 m by 1.5 m wide rectangular gate MN is vertical and is hinged at point 0.15 m below the center of gravity of the gate. The total depth of water is 6 m. What horizontal force must be applied at the bottom of the gate to keep the gate closed ?*

Solution

Total pressure acting on the plane surface of the gate is given by

$$P = wA\bar{x}$$

$$A = (3.6 \times 1.5)\ \text{m}^2$$

and $$\bar{x} = (6 - 1.8) = 4.2\ \text{m}$$

$\therefore$ By substitution

$$P = 1000 \times (3.6 \times 1.5) \times 4.2$$

$$= 22\,680\ \text{kg(f)}$$

The depth of centre of pressure is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 4.2 + \frac{\frac{1}{12} \times 1.5 \times (3.6)^3}{(1.5 \times 3.6) \times 4.2}$$

$$= (4.2 + 0.257) = 4.457\ \text{m}$$

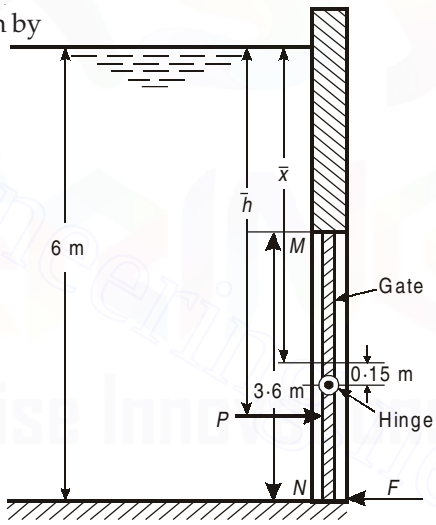


Figure Ex. 3.1.

Let F be the force required to be applied at the bottom of the gate to keep it closed.

By taking moments of all the forces about the hinge and equating to zero for equilibrium, we get

$$F\ (1.8 - 0.15) - 22680\ (0.257 - 0.15) = 0$$

$$\therefore \quad F = \frac{22680 \times 0.107}{1.65} = 1471\ \text{kg(f)}$$

Example 3.2. *A vertical gate closes a horizontal tunnel 5 m high and 3 m wide running full with water. The pressure at the bottom of the gate is 196.2 kN/m^2 [12 kg(f) cm^2]. Determine the total pressure on the gate and position of the centre of pressure.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

(*a*) *SI units*

The height of imaginary free surface of water above the bottom of the gate equivalent to a pressure intensity of 196.2 kN/m^2 is

$$h = \frac{p}{w} \frac{196.2 \times 10^3}{9810} = 20 \text{ m}$$

The total pressure on the gate is

$$P = wA\bar{x}$$

$$w = 9\,810 \text{ N/m}^3 ; A = (5 \times 3) = 15 \text{ m}^2 ;$$

$$\bar{x} = (20 - 2.5) = 17.5 \text{ m}$$

Thus by substitution, we get

$$P = (9\,810 \times 15 \times 17.5)$$

$$= 2.575 \times 106 \text{ N} = 2.575 \text{ MN}$$

(*b*) *Metric units*

The height of imaginary free surface of water above the bottom of the gate equivalent to a pressure intensity of 2 kg(f)/cm^2 is

$$h = \frac{p}{w} = \frac{2 \times 10^4}{1000} = 20 \text{ m}$$

The total pressure on the gate is

$$P = wA\bar{x}$$

$$w = 1000 \text{ kg(f)/m}^3; A = (5 \times 3) = 15 \text{ m}^2$$

$$\bar{x} = (20 - 2.5) = 17.5 \text{ m}$$

Thus by substitution, we get

$$P = (1000 \times 15 \times 17.5)$$

$$= 2.625 \times 10^5 \text{ kg(f)}$$

The position of the centre of pressure in both the systems of units is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

or

$$\bar{h} = 17.5 + \frac{\frac{1}{12} \times 3 \times (5)^3}{(3 \times 5) \times 17.5}$$

$$= (17.5 + 0.119) = 17.619 \text{ m}$$

i.e., the total pressure acts of 0.119 m below the centroid of the gate.

Example 3.3. *A triangular gate which has a base of 1.5 m and an altitude of 2 m lies in a vertical plane. The vertex of the gate is 1 m below the surface of a tank which contains oil of specific gravity 0.8. Find the force exerted by the oil on the gate and the position of the centre of pressure.*

Solution

(*a*) *SI units*

The force exerted on the gate is given by

$$P = wA\bar{x}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$w = (9\,810 \times 0.8)$$
$$= 7\,848 \text{ N/m}^3 = 7.848 \text{ kN/m}^3$$

$$A = \left(\frac{1}{2} \times 1.5 \times 2\right) = 1.5 \text{ m}^2$$

$$\bar{x} = \left(1 + \frac{2}{3} \times 2\right) = 2.33 \text{ m}$$

Thus by substitution, we get

$$P = (7.848 \times 1.5 \times 2.33)$$
$$= 27.43 \text{ kN}$$

(*b*) *Metric units*

The force exerted on the gate is given by

$$P = wA\bar{x}$$
$$w = (1000 \times 0.8) = 800 \text{ kg(f)/m}^3$$

$$A = \left(\frac{1}{2} \times 1.5 \times 2\right) = 1.5 \text{ m}^2$$

$$\bar{x} = \left(1 + \frac{2}{3} \times 2\right) = 2.33 \text{ m}$$

Thus by substitution, we get

$$P = (800 \times 1.5 \times 2.33)$$
$$= 2\,796 \text{ kg(f)}$$

The position of the centre of pressure in both the systems of units is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$I_G = \frac{1}{36} \times 1.5 \times (2)^3 = 0.33 \text{ m}^4$$

$$\therefore \quad \bar{h} = 2.33 + \frac{0.33}{1.5 \times 2.33} = 2.42 \text{ m}$$

Example 3.4. *The caisson for closing the entrance to a dry dock is of trapezoidal form 15 m wide at top and 12 m wide at bottom and 8 m deep. If the water on outside is just level with the top and the dock is empty, find the total water pressure on it and the depth of the centre of pressure. Take specific weight of sea water as 10.055 kN/m³ [1025 kg(f)/m³].*

Solution

(*a*) *SI units*

The total pressure on the surface is given by

$$P = wA\bar{x}$$
$$w = 10.055 \text{ kN/m}^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$A = \left(\frac{15+12}{2}\right) \times 8 = 108 \text{ m}^2$$

$$\bar{x} = \left(\frac{2\times12+15}{15+12}\right)\frac{8}{3} = 3.85 \text{ m}$$

Thus by substitution, we get

$$P = (10.055 \times 108 \times 3.85)$$
$$= 4\,181 \text{ kN} = 4.181 \text{ MN}$$

(*b*) *Metric units*

The total pressure on the surface is given by

$$P = wA\bar{x}$$
$$w = 1025 \text{ kg(f)/m}^3$$
$$A = 108 \text{ m}^2;\ \bar{x} = 3.85 \text{ m}$$

Thus by substitution, we get

$$P = (1025 \times 108 \times 3.85)$$
$$= 4.262 \times 10^5 \text{ kg(f)}$$

The depth of centre of pressure in both the systems of units is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$I_G = \frac{[(12)^2 + 4(15\times12) + (15)^2]}{36(15+12)} \times (8)^3$$
$$= 573.63 \text{ m}^4$$

Thus by substitution, we get

$$\bar{h} = 3.85 + \frac{573.63}{108\times3.85} = 5.23 \text{ m}$$

Example 3.5. *A rectangular door covering an opening 3 m wide and 2 m high in a vertical wall is hinged about its vertical edge by two pivots placed symmetrically 0.25 m from either end. The door is locked by a clamp placed at the centre of the vertical edge. Determine the reactions at the two hinges and the clamp, when the height of water is 1.5 m above the top edge of the opening.*

Solution

(*a*) *SI units*

The total pressure acting on the door is given by

$$P = wA\bar{x}$$
$$W = 9\,810 \text{ N/m}^3$$
$$A = (3\times2) = 6 \text{ m}^2;\ \bar{x} = (1.5+1) = 2.5 \text{ m}$$

$\therefore$
$$P = 9\,810 \times 6 \times 2.5$$
$$= 1\,47\,150 \text{ N} = 147.15 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By symmetry half of the total pressure will be resisted by the clamp and the other half is resisted by the two hinges.

Thus if R_c represents the clamp reaction, then

$$R_c = \frac{147.15}{2} = 73.575 \text{ kN}$$

The position of the centre of pressure is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 2.5 + \frac{\frac{1}{12} \times 3 \times (2)^3}{(3 \times 2) \times 2.5} = 2.633 \text{ m}$$

Let R_T and R_B be the hinge reactions at the top and the bottom hinges respectively. Then taking moments of all the forces about the top hinge and equating the algebraic sum of the moments of all the forces to zero for equilibrium, we have

$$P \times (2.633 - 1.5 - 0.25) - R_B \times 1.5 - R_C \times 0.75 = 0$$

or $$147.15 \times 0.883 - R_B \times 1.5 - 73.575 \times 0.75 = 0$$

or $$R_B = 49.835 \text{ kN}$$

Since $$R_T + R_B = 73.575 \text{ kN}$$

$\therefore$ $$R_T = (73.575 - 49.835) = 23.74 \text{ kN}$$

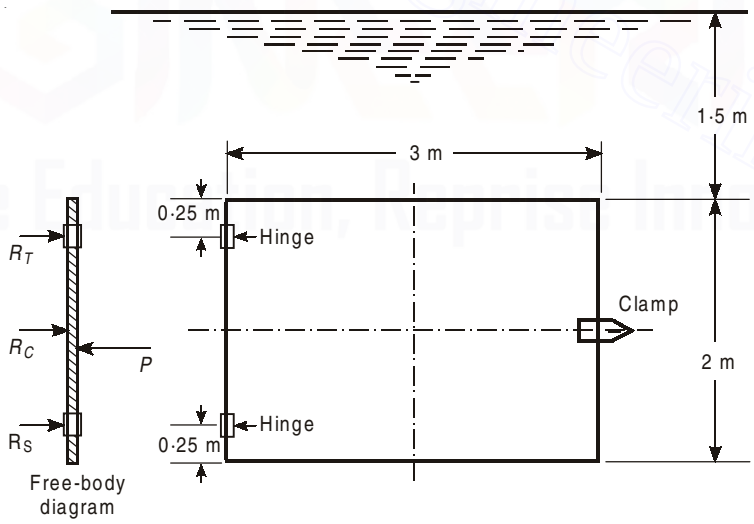


Figure Ex. 3.5

(*b*) *Metric units*

The total pressure acting on the door is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$P = wA\bar{x}$$
$$w = 1000 \text{ kg(f)/m}^3$$
$$A = (3 \times 2) = 6 \text{ m}^2;\ \bar{x} = (1.5 + 1) = 2.5 \text{ m}$$
$$\therefore \quad P = 1000 \times 6 \times 2.5 = 15\,000 \text{ kg(f)}$$

Clamp reaction $R_C = \dfrac{15000}{2} = 7500$ kg(f)

Depth of centre of pressure

$$\bar{h} = 2.633 \text{ m}$$

Taking moments of all the forces about the top hinge and equating the algebraic sum of the moments of all the forces to zero for equilibrium, we have

$$15\,000 \times (2.633 - 1.5 - 0.25) - R_B \times 1.5 - 7500 \times 0.75 = 0$$

or $R_B = 5\,080$ kg(f)

Since $R_T + R_B = 7500$

$\therefore \quad R_T = 2\,420$ kg(f)

Example 3.6. *A vertical gate 2 m × 2 m rests with its top edge 1 m below the water surface. Find the depth of a horizontal line which divides the gate such that (i) the pressure on the top portion is equal to the pressure on the bottom portion; (ii) the moments of the pressures on the top and bottom portions about this line are equal.*

Solution

(i) Let the required dividing line be located at a distance x below the top edge of the gate.

Thus total pressure P_1 on the top portion of the gate is given by

$$P_1 = w \times (2 \times x)\left(1 + \frac{x}{2}\right)$$

or $$P_1 = w\,[x(2 + x)] \quad \ldots(i)$$

Similarly total pressure P_2 on the bottom portion of the gate is given by

$$P_2 = w \times 2(2 - x) \times \left(1 + x + \frac{2 - x}{2}\right)$$

or $$P_2 = w\,[(2 - x)\,(4 + x)] \quad \ldots(ii)$$

Equating the total pressures on the two portions of the gate, we get

$$w[x(2 + x)] = w[(2 - x)\,(4 + x)]$$

or $$x^2 + 2x - 4 = 0$$

or $$x = 1.24 \text{ m}$$

i.e., the dividing line is at a depth of 1.24 m below the top of the gate.

(ii) Let the required dividing line be located at a distance x below the top edge of the gate.

Thus total pressures P_1 and P_2 on the top and bottom portions of the gate are given by Eqs (i) and (ii) as

$$P_1 = w[x(2 + x)] \quad \ldots(i)$$
$$P_2 = w[(2 - x)\,(4 + x)] \quad \ldots(ii)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The depth of the point of application of P_1 below the free surface is given by

$$\bar{h}_1 = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$\bar{x} = \left(1 + \frac{x}{2}\right) = \frac{1}{2}(2 + x); A = (2 \times x) = 2x$$

$$I_G = \left(\frac{1}{12} \times 2 \times x^3\right) = \frac{1}{6}x^3$$

Thus by substitution, we get

$$\bar{h}_1 = \frac{1}{2}(2 + x) + \frac{\frac{1}{6}x^3}{2x \times \frac{1}{2}(2 + x)}$$

or $$\bar{h}_1 = \frac{2\left(x^2 + 3x + 3\right)}{3(2 + x)}$$

If the point of application of P_1 is located at a distance y_1 from the dividing line, then

$$y_1 = (1 + x) - \bar{h}_1$$

or $$y_1 = (1 + x) - \frac{2\left(x^2 + 3x + 3\right)}{3(2 + x)}$$

or $$y_1 = \frac{x(3 + x)}{3(2 + x)}$$

Similarly the depth of the point of application of P_2 below the free surface is given by

$$\bar{h}_2 = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$\bar{x} = \left(1 + x + \frac{2 - x}{2}\right) = \left(\frac{4 + x}{2}\right)$$

$$A = 2(2 - x); \text{ and } I_G = \frac{1}{12} \times 2(2 - x)^3$$

Thus by substitution, we get

$$\bar{h}_2 = \left(\frac{4 + x}{2}\right) + \frac{\frac{1}{12} \times 2(2 - x)^3}{2(2 - x) \times \left(\frac{4 + x}{2}\right)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\bar{h}_2 = \frac{2x^2 + 10x + 26}{3(4+x)}$$

If the point of application of P_2 is located at a distance y_2 from the dividing line, then

$$y_2 = \bar{h}_2 - (1 + x)$$

or $$y_2 = \frac{2x^2 + 10x + 26}{3(4+x)} - (1 + x)$$

or $$y_2 = \frac{14 - 5x - x^2}{3(4+x)} = \frac{(2-x)(7+x)}{3(4+x)}$$

Equating the moments of P_1 and P_2 about the dividing line, we get

$$P_1 \times y_1 = P_2 \times y_2$$

By substitution, we get

$$w[x(2+x)] \times \frac{x(3+x)}{3(2+x)} = w[(2-x)(4+x)] \times \frac{(2-x)(7+x)}{3(4+x)}$$

or $$x^2(3 + x) = (2 - x)^2 (7 + x)$$

or $$x = 1.167 \text{ m}$$

i.e., the dividing line is at a depth of 1.167 m below the top of the gate.

Example 3.7. *The vertical side of a reservoir has a rectangular opening 2.75 m long and 1.2 m high. It is closed by a plate using 4 bolts placed at the corners of the opening. What would be the tension in the bolts if water stands to a height of 1.8 m above the top edge of the opening which is horizontal?*

Solution

The total pressure on the plate is given by

$$P = wA\,\bar{x}$$

$w = 9.81 \text{ kN/m}^3$; $A = (1.2 \times 2.75) = 3.3 \text{ m}^2$; and

$\bar{x} = (1.8 + 0.6) = 2.4 \text{ m}$

Thus by substitution, we get

$$P = (9.81 \times 3.3 \times 2.4)$$
$$= 77.6952 \text{ kN}$$

The depth of the centre of pressure below the water surface is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$\bar{x} = 2.4$ m; $A = 3.3 \text{ m}^2$; and $I_G = \frac{1}{12} \times 2.75 \times (1.2)^3$

Thus by substitution, we get

$$\bar{h} = 2.4 + \frac{\frac{1}{12} \times 2.75 \times (1.2)^3}{(2.75 \times 1.2) \times 2.4}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $\bar{h} = 2.45$ m

Let R_T and R_B be the tensions in each of the top and the bottom bolts respectively.

Thus

$$2(R_T + R_B) = 77.6952$$

or $$R_T + R_B = 38.8476 \quad \ldots(i)$$

Further by taking the moments of all the forces about the top bolt line, we get

$$2 \times 1.2 \times R_B = 77.6952 \times (2.45 - 1.8)$$

$$\therefore \quad R_B = 21.0425 \text{ kN}$$

and by introducing the value of R_B in Eq. (i) we get

$$R_T = 17.8051 \text{ kN}.$$

Example 3.8. *A circular plate 2.5 m diameter is immersed in water, its greatest and least depth below the free surface being 3 m and 1 m respectively. Find (a) the total pressure on one face of the plate, and (b) the position of the centre of pressure.*

Solution

(*a*) *SI units*

The total pressure on one face of the plate is given by

$$P = wA\bar{x}$$

$w = 9\,810$ N/m^3;

$$A = \frac{\pi}{4}(2.5)^2 = 4.909 \text{ m}^2; \text{ and}$$

as shown in Fig. Ex. 3.8

$$\bar{x} = \left(\frac{3+1}{2}\right) = 2 \text{ m}$$

$$\therefore \quad P = (9810 \times 4.909 \times 2)$$

$$= 96\,315 \text{ N} = 96.315 \text{ kN}$$

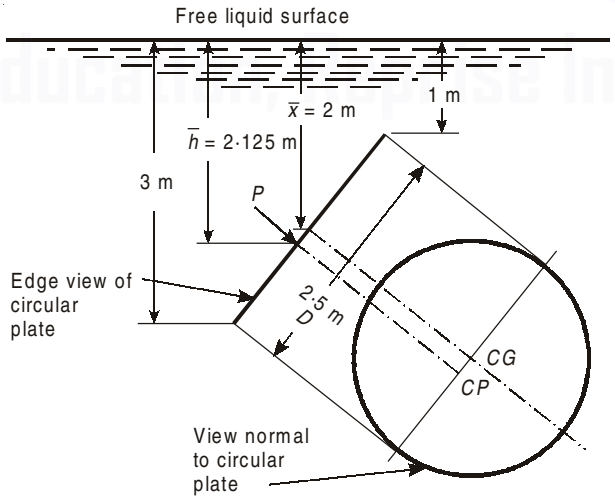


Figure Ex. 3.8

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*b*) *Metric units*

The total pressure on one face of the plate is given by

$$P = wA\bar{x}$$
$$w = 1000 \text{ kg(f)/m}^3$$
$$A = 4.909 \text{ m}^2$$
$$\bar{x} = 2 \text{ m}$$
$$\therefore \quad P = 1000 \times 4.909 \times 2$$
$$= 9\,818 \text{ kg(f)}$$

The depth of the centre of pressure is given by

$$\bar{h} = \bar{x} + \frac{I_G \sin^2 \theta}{A\bar{x}}$$

$$I_G = \frac{\pi}{64}(2.5)^4 = 1.917 \text{ m}^4; \sin\theta = \frac{2}{2.5} = 0.8$$

$$\therefore \quad \bar{h} = 2 + \frac{(1.917)\times(0.8)^2}{(4.909\times 2)} = 2.125 \text{ m.}$$

Example 3.9. *An opening in a dam is closed by a plate 1 m square which is hinged at the upper horizontal edge as shown in Fig. Ex. 3.9. The plate is inclined at an angle of 60° to the horizontal and its top edge is 2 m below the water surface in the reservoir. If this plate is pulled by means of a chain attached to the centre of the lower edge, find the necessary pull T in the chain. The line of action of the chain makes an angle of 45° with the plate. Weight of the plate is 1.962 kN.*

Solution

Area of plate $A = (1\times 1) = 1 \text{ m}^2$

Depth of CG below the free surface of water

$$= \left(2 + \frac{1}{2}\sin 60°\right)$$
$$= 2.433$$

Total pressure acting on the plate is

$$P = wA\bar{x}$$
$$= (9810 \times 1 \times 2.433)$$
$$= 23\,868 \text{ N} = 23.868 \text{ kN}$$

The depth of the centre of pressure below the water surface is given by

$$\bar{h} = \bar{x} + \frac{I_G \sin^2 \theta}{A\bar{x}}$$

$$= 2.433 + \frac{\frac{1}{12}\times\left(\frac{\sqrt{3}}{2}\right)^2}{1\times 2.433} = 2.459 \text{ m}$$

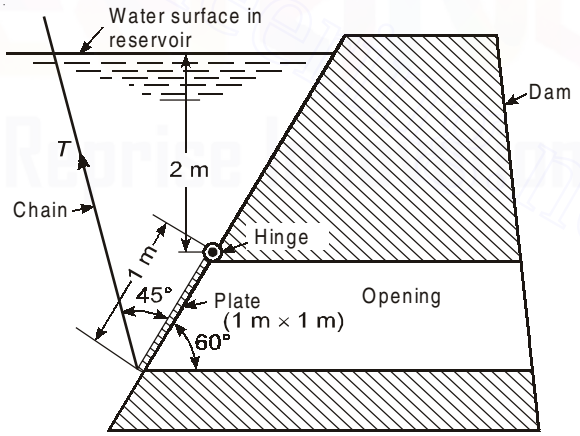


Figure Ex. 3.9

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore$ The distance of the total pressure P from the hinge along the plate is equal to $(2.459 - 22)\dfrac{1}{\sin 60°} =$ 0.53 m.

Let T be the necessary pull in the chain. Then taking the moments of all the forces about the hinge and equating the algebraic sum of the moments to zero, we get

$$T \times (1 \times \sin 45°) = (23.868 \times 0.53) + 1.962 \times \left(\frac{1}{2}\cos\ 60°\right)$$

or $$\frac{T}{\sqrt{2}} = 13.141$$

$\therefore$ $$T = 18.584 \text{ kN}$$

Example 3.10. *A cubical box, 2 m on each edge, has its base horizontal and is half filled with a liquid of specific gravity 1.5. The remainder of the box is filled with an oil of specific gravity 0.90. One of the sides is held in position by means of four screws, one at each corner. Find the tension in each screw due to hydrostatic pressure.*

Solution

Since there are two liquids in contact with the vertical sides of the tank, the total pressure on each of the vertical sides may be determined by drawing a pressure diagram as shown in the Fig. Ex. 3.10.

The total pressure on one of the vertical sides is equal to the area of the pressure diagram multiplied by the length of the side.

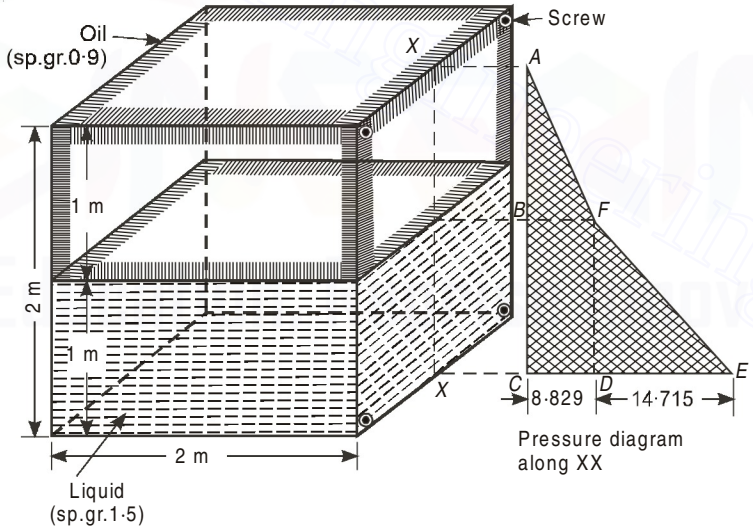


Figure Ex.3.10

The area of the pressure diagram *ABCDEFA* is equal to

$$\left[\frac{1}{2} \times (0.9 \times 9810 \times 1) \times 1 + (0.9 \times 9810 \times 1 \times 1) + \frac{1}{2}(1.5 \times 9810 \times 1) \times 1\right]$$

$$= 20\,601 \text{ N/m} = 20.601 \text{ kN/m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

∴ Total pressure on the vertical side is

$$P = (20.601 \times 2) = 41.202 \text{ kN}$$

The total pressure will act at the centroid of the pressure diagram *ABCDEFA*, which may be determined by applying the principle of moments.

Let the centroid of the pressure diagram *ABCDEFA* be at a distance x above the base.

By taking the moments about the base, we get

$$(20.601 \times x) = \frac{1}{2} \times 8.829 \times 1 \times \left(2 - \frac{2}{3}\right) + 8.829 \times 1 \times \left(\frac{1}{2}\right) + \frac{1}{2} \times 14.715 \times 1 \times \left(\frac{1}{3}\right)$$

$$\therefore \quad x = 0.62 \text{ m}$$

Alternatively the total pressure on the vertical side of the tank as well as its point of application may be computed by employing imaginary liquid as indicated below.

The total pressure P on one of the vertical sides of the tank is equal to $(P_1 + P_2)$, where P_1 and P_2 are the total pressures on the portions of the side in contact with the top and the bottom liquids respectively which may be computed as follows :

$$P_1 = 0.9 \times 9\,810 \times (2 \times 1) \times 0.5 = 8\,829 \text{ N} = 8.829 \text{ kN}$$

For computing P_2, 1 m depth of the superimposed liquid of specific gravity 0.9 may be converted into an equivalent depth of liquid of specific gravity 1.5 to obtain an imaginary liquid surface (ILS). Since 1 m depth of liquid of specific gravity 0.9 is equivalent to $(1 \times 0.9/1.5) = 0.6$ m depth of liquid of specific gravity 1.5, the ILS is located 1.6 m above the base of the tank. Then

$P_2 = 1.5 \times 9\,810 \times (2 \times 1) \times (0.6 + 0.5) = 32\,373 \text{ N} = 32.373 \text{ kN}$ acting at a depth $\bar{h}$ below ILS given by

$$\bar{h} = (0.6 + 0.5) + \frac{\frac{1}{12} \times (2 \times 1^3)}{(2 \times 1) \times (0.6 + 0.5)} = 1.176 \text{ m}$$

Thus the total pressure $P = (8.829 + 32.373) = 41.202$ kN and its point of application may be obtained by the principle of moments as follows:

Let x be the distance of the point of application of P above the base of the tank, then by taking the moments of P_1, P_2 and P about the base, we get

$$41.202 \times x = 8.829 \times \left(1 + \frac{1}{3}\right) + 32.373\ (1.6 - 1.176)$$

$$\therefore \quad x = 0.62 \text{ m}$$

All the four screws will resist combinedly the force equal to the total pressure. Further by symmetry half of the total pressure will be resisted by the two screws on one side of the vertical axis and the remaining half by the two screws on the other side.

Thus if R_T and R_B are the tensile forces in the top and the bottom screws on one side of the vertial axis, then

$$R_T + R_B = \frac{P}{2} = \frac{41.202}{2} = 20.601 \text{ kN} \qquad \text{...(i)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Also the resultant of R_T and R_B will act at the same height above the base as the total pressure. Then by taking the moments about the point at which the resultant acts, we get

$$R_T \times (2 - 0.62) = R_B \times 0.62$$

or $$1.38\, R_T = 0.62\, R_B \quad \text{...(ii)}$$

By solving Eqs (i) and (ii), we get

$$R_T = 6.386 \text{ kN}; \; R_B = 14.215 \text{ kN}$$

That is each of the top screws will have a tensile force of 6.386 kN and each of the bottom screws will have a tensile force of 14.215 kN.

Example 3.11. *Water rises to level M in the pipe attached to the tank ABCD shown in Fig. E.x. 3.11. (a) Compute and locate the total pressure acting on the area AB which is 2.5 m wide. (b) Compute the total pressure on the bottom of the tank having area 6.5 m × 2.5 m, and (c) Compare the total weight of the water with the result in (b). Explain why there is a difference between the two. Neglect the weight of the tank and the riser pipe.*

Solution

(a) The total pressure on the surface AB is given by

$$P = wA\bar{x}$$

$$A = (2 \times 2.5) = 5 \text{ m}^2$$

$$\bar{x} = (4 + 1) = 5 \text{ m}$$

$$\therefore \quad P = (9\,810 \times 5 \times 5)$$

$$= 245\,250 \text{ N} = 245.25 \text{ kN}$$

This will be acting at a depth given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 5 + \frac{\frac{1}{12} \times 2.5 \times (2)^3}{(2.5 \times 2) \times 5} = 5.07 \text{ m from O}'$$

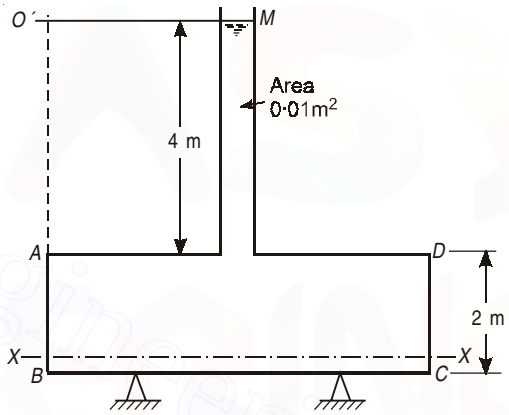


Figure Ex. 3.11

(b) The pressure intensity on the bottom BC is uniform and equal to

$$p = 9\,810 \times (4 + 2)$$

$$= 58\,860 \text{ N/m}^2 = 58.86 \text{ kN/m}^2$$

∴ Total pressure on the bottom of the tank is

$$P = (p \times A)$$

$$= 58.86 \times (6.5 \times 2.5)$$

$$= 956.475 \text{ kN}$$

(c) The total weight of the water is

$$W = 9\,810\,(6.5 \times 2.5 \times 2 + 4 \times 0.01)$$

$$= 319\,217.4 \text{ N} = 319.217\,4 \text{ kN}$$

Thus it is observed that the total weight of the water in the tank is much less than the total pressure on the bottom of the tank. This is known as Pascal's paradox or merely the paradox in Hydraulics. The reason for such an apparent paradox is as given below.

A free-body of the lower part of the tank (cut by horizontal plane XX above level BC) will indicate a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

downward force on area *BC* of 956.475 kN and upward forces due to vertical tension in the walls of the tank, and the reaction of the supporting plane. The reaction of the supporting plane must equal the total weight of water i.e., 319.217 4 kN. The tension in the walls of the tank is caused by the upward force on the top *AD* of the tank which is

$$\begin{aligned} P_{AD} &= wA\bar{x} \\ &= 9\,810\,(6.5 \times 2.5 - 0.01) \times 4 \\ &= 637\,257.6\ \text{N} = 637.257\,6\ \text{kN} \end{aligned}$$

Since for the equilibrium of the free-body diagram considered the sum of the vertical forces is equal to zero, i.e.,

$$956.475 - 319.217\,4 - 637.257\,6 = 0$$

which clarifies the paradox.

Example 3.12. *Gate PQ shown in Fig. Ex. 3.12 is 1.25 m wide and 2 m high and it is hinged at P. Gage G reads – 14.715 kN/m^2. The left hand tank contains water and the right hand tank oil of specific gravity 0.75 upto the heights shown in the figure. What horizontal force must be applied at Q to keep the gate closed?*

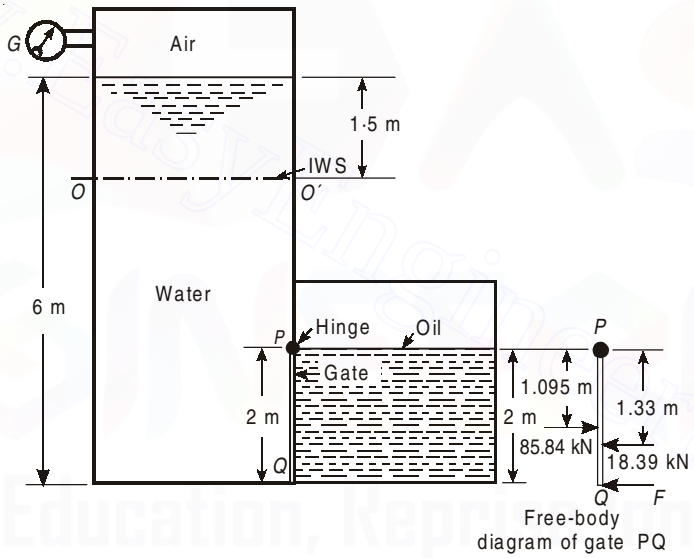


Figure Ex. 3.12

Solution

On R.H.S. of gate the total pressure due to oil is given by

$$\begin{aligned} P_{\text{oil}} &= wA\bar{x} \\ &= (9\,810 \times 0.75) \times (1.25 \times 2) \times 1 \\ &= 18\,390\ \text{N} = 18.39\ \text{kN} \end{aligned}$$

It is acting normal to the gate at a depth of

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 1 + \frac{\frac{1}{12} \times (1.25 \times 2^3)}{(1.25 \times 2) \times 1} = 1.33\ \text{m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For L.H.S. of gate, it is necessary to convert the negative pressure due to the air to its equivalent in metres of water. Thus

$$\bar{h} = \frac{p}{w} = \frac{14.715}{9.81} = -1.5 \text{ m}$$

This negative pressure head is equivalent to having the water level in the tank reduced by 1.5 m. It is therefore convenient and useful to employ an imaginary water surface (IWS) 1.5 m below the real surface and solve the problem by direct use of the basic equations.

Thus total pressure on the gate due to water is given by

$$P_{\text{water}} = 9\,810 \times (2 \times 1.25) \times (4.5 - 1)$$
$$= 85\,840 \text{ N} = 85.84 \text{ kN}$$

It is acting normal to the gate at a depth of

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 3.5 + \frac{\frac{1}{12} \times (1.25 \times 2^3)}{(1.25 \times 2) \times 3.5}$$

$$= 3.595 \text{ m below the imaginary water surface } OO'$$

Consider the free-body diagram of the gate shown in the Fig. Ex. 3.12, in which the forces acting on the gate are shown. Taking the moments of all the forces about the hinge and equating the sum of the moments to zero for the equilibrium of the gate, we get

$$F \times 2 + (18.39 \times 1.33) - 85.84 \times (3.595 - 2.5) = 0$$

$\therefore$ $F = 34.768$ kN acting at Q to the left.

Example 3.13. *Calculate the total pressure and the x and y coordinates of the centre of pressure of the vertical right angled triangular plane held immersed in water as shown in Fig. Ex. 3.13.*

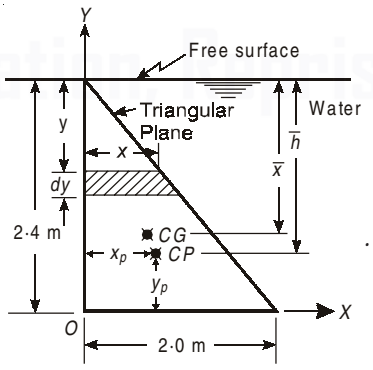


Figure Ex. 3.13

Solution

The total pressure on one face of the plane is given by

$$P = wA\bar{x}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$A = \left(\frac{1}{2}\times 2.4\times 2.0\right) = 2.4 \text{ m}^2$$

$$\bar{x} = \frac{2}{3}(2.4) = 1.6 \text{ m}$$

$$\therefore \quad P = (9\,810 \times 2.4 \times 1.6)$$
$$= 37\,670 \text{ N} = 37.67 \text{ kN}$$

Let x_p and y_p be the coordinates of the centre of pressure.

Consider on the plane surface a horizontal strip of thickness dy and width x lying at a vertical depth y below the surface. The total pressure on the strip is

$$dP = wy(xdy)$$

Since the thickness of the strip is very small, it may be considered as rectangular in shape and hence its centre of pressure may be considered to be located at a distance $(x/2)$ from the axis OY and at a depth y below the free surface.

The moment of P about axis OY is $(dP)\,\frac{x}{2} = \frac{1}{2}(wy\,x^2\,dy)$, and the sum of the moments of the total pressures on all such strips considered on the plane surface is

$$\int (dP)\frac{x}{2} = \frac{1}{2}w\int_{y=0}^{y=2.4} yx^2dy$$

which by the principle of moments is equal to the moment of P about axis OY, i.e., (Px_p). Thus

$$(Px_p) = \frac{1}{2}w\int_{y=0}^{y=2.4} yx^2dy$$

or
$$(wA\bar{x})\,x_p = \frac{1}{2}w\int_{y=0}^{y=2.4} yx^2dy$$

or
$$x_p = \frac{1}{2A\bar{x}}\int_{y=0}^{y=2.4} yx^2dy$$

From the similar triangles, we have

$$\frac{x}{2} = \frac{y}{2.4}\text{; or } x = \left(\frac{2}{2.4}\right)y$$

Thus by substitution, we get

$$x_p = \frac{1}{(2\times 2.4\times 1.6)}\times\left(\frac{2}{2.4}\right)^2\int_{y=0}^{y=2.4} y^3dy$$

or
$$x_p = \frac{1}{(2\times 2.4\times 1.6)}\times\left(\frac{2}{2.4}\right)^2\times\frac{1}{4}\times(2.4)^4$$

or
$$x_p = 0.75 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The moment of dP about OX is

$$(dP)\,(2.4-y) = [(wy\,(x\,dy)\,(2.4-y)]$$

and the sum of the moments of the total pressures on all such strips considered on the plane surface is

$$\int (dP)\,(2.4-y) = w\int_{y=0}^{y=2.4} xy(2.4-y)\,dy$$

which by the principle of moments is equal to the moment of P about axis OX, i.e., (Py_p). Thus

$$Py_p = w\int_{y=0}^{y=2.4} xy(2.4-y)\,dy$$

or
$$(wA\,\bar{x})\,y_p = w\int_{y=0}^{y=2.4} xy(2.4-y)\,dy$$

or
$$y_p = \frac{1}{A\bar{x}}\,w\int_{y=0}^{y=2.4} xy(2.4-y)\,dy$$

Again by substitution, we get

$$y_p = \frac{1}{(2.4\times 1.6)}\times\frac{2}{2.4}\int_{y=0}^{y=2.4} y^2(2.4-y)dy$$

or
$$y_p = \frac{1}{(2.4\times 1.6)}\times\left(\frac{2}{2.4}\right)\times\left[\frac{2.4\,y^3}{3}-\frac{y^4}{4}\right]_0^{2.4}$$

or
$$y_p = \frac{1}{(2.4\times 1.6)}\times\frac{2}{2.4}\times\frac{(2.4)^4}{12}$$

or
$$y_p = 0.6\text{ m}$$

Note : The value of y_p may also be obtained by an alternative method as follows :

Since
$$y_p = (2.4-\bar{h}\,)$$

where
$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

and
$$I_G = \frac{bh^3}{36} = \frac{2\times(2.4)^3}{36} = 0.768\text{ m}^4$$

So
$$\bar{h} = 1.6 + \frac{0.768}{2.4\times 1.6} = 1.8\text{ m}$$

$\therefore$
$$y_p = (2.4-1.8) = 0.6\text{ m}$$

Therefore total pressure on the plane surface = 37.67 kN and the coordinates of the centre of pressure are $x_p = 0.75$ m and $y_p = 0.6$ m.

Example 3.14. *An annular plate 3 m external diameter and 1.5 m internal diameter is immersed in water with its greatest and least depths below water surface as 3.6 m and 1.2 m respectively. Determine the total pressure and the position of the centre of pressure on one face of the plate.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

As shown in Fig. Ex. 3.14, the external area of the plate $= \frac{\pi}{4}(3)^2 = (2.25\,\pi)\ \text{m}^2$

The internal area of the plate $= \frac{\pi}{4}(1.5)^2 = (0.56\,\pi)\ \text{m}^2$

$\therefore$ Net area of the plate $= (2.25\,\pi - 0.56\,\pi) = 1.69\,\pi = 5.31\ \text{m}^2$

The depth of the centroid of the plate below the water surface

$$\bar{x} = \frac{3.6 + 1.2}{2} = 2.4\ \text{m}$$

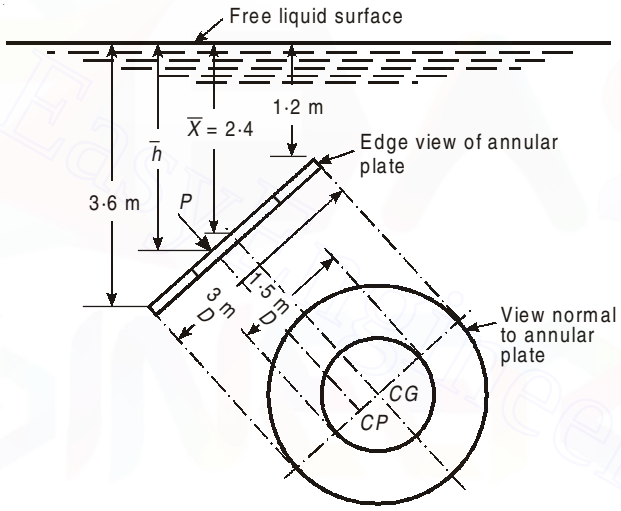


Figure Ex. 3.14

The total pressure on one face of the plate is given by

$$P = wA\bar{x}$$
$$= (9\,810 \times 5.31 \times 2.4)$$
$$= 1\,25\,019\ \text{N} = 125.019\ \text{kN}$$

The depth of the centre of pressure below the free surface of water is given by

$$\bar{h} = \bar{x} + \frac{I_G \sin^2\theta}{A\bar{x}}$$

$$I_G = \frac{\pi}{64}(D^4 - d^4) = \frac{\pi}{64}[(3)^4 - (1.5)^4]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\sin\theta = \frac{2.4}{3} = 0.8 \text{ m}$$

$$\therefore \quad \bar{h} = 2.4 + \frac{\frac{\pi}{64}[(3)^4 - (1.5)^4] \times (0.8)^2}{\frac{\pi}{4}[(3)^2 - (1.5)^2] \times 2.4}$$

$$= 2.59 \text{ m}$$

Example 3.15. *A square disc of side 1 m is immersed vertically in water so that an edge of the square makes an angle of 35 with the horizontal. If the highest corner of the disc is at a depth of 1.5 m below the free surface, find the total pressure on one face of the disc and the depth of centre of pressure.*

Solution

The total pressure on the plane surface is given by

$$P = wA\bar{x}$$

From Fig. Ex. 3.15 the value of $\bar{x}$ may be determined as follows.

$$\angle AOP = 10° \text{ ; and } OP = AO \cos 10°$$

But $$AO = \frac{1}{\sqrt{2}} \text{ m}$$

$$\therefore \quad OP = \left(\frac{1}{\sqrt{2}} \times \cos 10°\right) = 0.696 \text{ m}$$

Thus $$\bar{x} = (1.5 + 0.696) = 2.196 \text{ m}$$

$$A = (1 \times 1) = 1 \text{ m}^2$$

By substitution, we get

$$P = (9\,810 \times 1 \times 2.196)$$

$$= 21\,543 \text{ N} = 21.543 \text{ kN}$$

The depth of centre of pressure is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

For computing the moment of inertia (M.I.) I_G of the plane surface, its three portions may be considered which are (as shown Fig. Ex. 3.15) triangle *AXD*, parallelogram *XDYB* and triangle *BYC*. Then

I_G = [(M.I. of ΔAXD about axis *GG*) + (M.I. of parallelogram *XDYB* about axis *GG*) + (M.I. of ΔBYC about axis *GG*)]

From the figure, the height of each of the triangles *AXD* and *BCY* is equal to

$$AD \sin 35° \text{ (or } BC \sin 35°)$$

$$= (1 \times 0.5736) = 0.5736 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

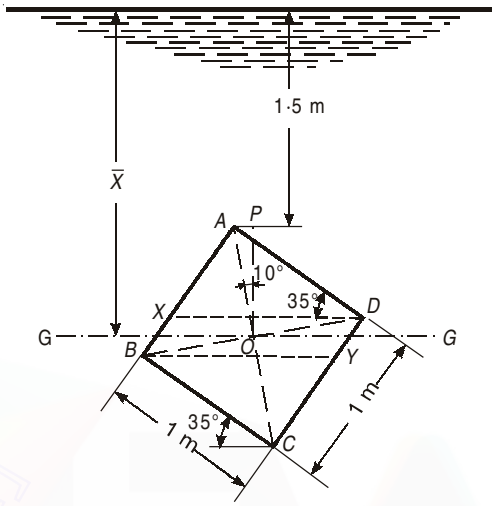


FIgure Ex. 3.15

Also the base of each of the triangles *AXD* and *BCY* is equal to

$$DX = BY = (1\times \sec 35°) = 1.2208$$

The height of the parallelogram *XDYB*

$$= (1 \times \cos 35°) - (1\times \sin 35°)$$
$$= (0.8192 - 0.5736) = 0.2456 \text{ m}$$

The area of $\Delta AXD = \left(\frac{1}{2}\times 1.2208 \times 0.5736\right) = 0.3501 \text{ m}^2$

$\therefore$ The moment of inertia of ΔAXD about axis *GG*

$$= \frac{1}{36} \times 1.2208 \times (0.5736)^3 + 0.3501 \left(\frac{0.5736}{3} + \frac{0.2456}{2}\right)^2$$
$$= (0.0064 + 0.0345) = 0.0409 \text{ m}^4$$

Similarly the moment of inertia of ΔBCY about axis *GG*

$$= 0.0409 \text{ m}^4$$

The moment of inertia of the parallelogram *XDYB* about axis *GG*

$$= \frac{1}{12} \times 1.2208 \times (0.2456)^3 = 0.0015 \text{ m}^4$$

$\therefore$ The moment of inertia of the whole plane about axis *GG* is

$$I_G = (0.0409 + 0.0015 + 0.0409)$$
$$= 0.0833 \text{ m}^4$$

Alternatively in the case of a square lamina, irrespective of its orientation, the value of I_G is given by

$$I_G = \frac{a^4}{12}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in which a is the side of the square lamina.

Thus in this case since $a = 1$ m

$$I_G = \frac{(1)^4}{12} = 0.0833 \text{ m}^4$$

$$\therefore \quad \bar{h} = 2.196 + \frac{0.0833}{1 \times 2.196} = 2.234 \text{ m.}$$

Note: The alternative method indicated above for computing I_G is, however, not applicable in the case of a rectangular lamina placed in a similar condition.

Example 3.16. *A vertical gate of width B and height H retains water on one side upto its top edge. Find the depths of two horizontal lines which divide the gate into three portions such that the total pressure on each portion is equal. Also locate the position of the centre of pressure for each of the three portions. Hence show that if a gate is divided into N such portions each of which is subjected to equal total pressure then the height of each portion is given by*

$$h_r = H\sqrt{\frac{r}{N}}$$

and the depth of centre of pressure for each portion below the top of the gate is given by

$$\bar{h}_r = \frac{2}{3} H \frac{\left[r^{3/2} - (r-1)^{3/2}\right]}{\sqrt{N}}$$

where r = 1, 2, 3……, N.

Solution

Let h_1 and h_2 be the depths of the two horizontal lines below the top of the gate, which divide the gate into three portions.

The total pressures on the three portions of the gate are thus obtained as

$$P_1 = w \times (B \times h_1) \times \frac{h_1}{2} = \frac{wBh_1^2}{2} \quad \text{... (i)}$$

$$P_2 = w \times [B \times (h_2 - h_1)] \times \frac{h_1 + h_2}{2}$$

or

$$P_2 = \frac{wB\left(h_2^2 - h_1^2\right)}{2} \quad \text{... (ii)}$$

$$P_3 = w \times [B \times (H - h_2)] \times \frac{h_2 + H}{2}$$

or

$$P_3 = \frac{wB\left(H^2 - h_2^2\right)}{2} \quad \text{... (iii)}$$

Since $P_1 = P_2 = P_3$; we obtain from Eqs (i) and (ii)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{wBh_1^2}{2} = \frac{wB\left(h_2^2 - h_1^2\right)}{2}$$

or $$2h_1^2 = h_2^2 \quad \text{... (iv)}$$

and from Eqs (ii) and (iii)

$$\frac{wB\left(h_2^2 - h_1^2\right)}{2} = \frac{wB\left(H^2 - h_2^2\right)}{2}$$

or $$2h_2^2 = h_1^2 + H^2 \quad \text{... (v)}$$

From Eqs (iv) and (v) we obtain

$$h_1 = H\sqrt{\frac{1}{3}} \quad \text{... (vi)}$$

and $$h_2 = H\sqrt{\frac{2}{3}} \quad \text{... (vii)}$$

Hence from these equations the following general equation may be obtained

$$h_r = H\sqrt{\frac{r}{N}}$$

Let $\bar{h}_1$, $\bar{h}_2$ and $\bar{h}_3$ be the depths of the centres of pressures below the top of the gate for the three portions of the gate. Thus

$$\bar{h}_1 = \frac{h_1}{2} + \frac{\frac{1}{12}B \times h_1^3}{\left(B \times h_1\right) \times \frac{h_1}{2}} = \frac{2}{3}h_1 \quad \text{... (viii)}$$

Substituting the value of h_1 from Eq. (vi), we get

$$\bar{h}_1 = \frac{2}{3}H\sqrt{\frac{1}{3}} \quad \text{... (ix)}$$

Similarly

$$\bar{h}_2 = \left(\frac{h_1 + h_2}{2}\right) + \frac{\frac{1}{12} \times B \times \left(h_2 - h_1\right)^3}{B \times \left(h_2 - h_1\right) \times \left(\frac{h_1 + h_2}{2}\right)}$$

or $$\bar{h}_2 = \frac{3\left(h_1 + h_2\right)^2 + \left(h_2 - h_1\right)^2}{6\left(h_1 + h_2\right)} \quad \text{... (x)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the values of h_1 and h_2 from Eqs (vi) and (vii), we get

$$\bar{h}_2 = \frac{2}{3}H\frac{\left(2^{3/2}-1\right)}{\sqrt{3}} \quad \text{... (xii)}$$

and

$$\bar{h}_3 = \left(\frac{h_2+H}{2}\right) + \frac{\frac{1}{12}\times B\times\left(H-h_2\right)^3}{B\times\left(H-h_2\right)\times\left(\frac{h_2+H}{2}\right)}$$

or

$$\bar{h}_3 = \frac{3\left(h_2+H\right)^2+\left(H-h_2\right)^2}{6\left(h_2+H\right)}$$

Substituting the value of h_2 from Eq. (vii), we get

$$\bar{h}_3 = \frac{2}{3}H\frac{\left(3^{3/2}-2^{3/2}\right)}{\sqrt{3}} \quad \text{... (xii)}$$

Hence from these equations the following general equation may be obtained

$$\bar{h}_r = \frac{2}{3}H\frac{\left(r^{3/2}-(r-1)^{3/2}\right)}{\sqrt{N}}$$

Example 3.17. *The end gates of a lock are 5 m high and when closed include an angle of 120°. The width of the lock is 6.25 m. Each gate is carried on two hinges placed at the top and the bottom of the gate. If the water levels are 4 m and 2 m on the upstream and downstream sides respectively, determine the magnitudes of the forces on the hinges due to the water pressure.*

Solution

Figure Ex. 3.17 shows a pair of lock gates.

The width of the lock is 6.25 m.

$$\therefore \text{ Width of each gate} = \frac{6.25}{2\cos 30°} = 3.61 \text{ m}$$

The total pressure on the upstream face of the gate is

$$P_u = wA\bar{x}$$

$$A = (4 \times 3.61) = 14.44 \text{ m}^2;\ \bar{x} = 2 \text{ m}$$

$$\therefore \quad P_u = (9819 \times 14.44 \times 2)$$

$$= 283\,313 \text{ N} = 283.313 \text{ kN}$$

The depth of the centre of pressure on the upstream face is given by

$$\bar{h}_u = \bar{x} + \frac{I_G}{A\bar{x}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\bar{h}_u = 2 + \frac{\frac{1}{12} \times 3.61 \times (4)^3}{(3.61 \times 4) \times 2} = 2.67 \text{ m}$$

The total pressure on the downstream face of the gate is

$$P_d = wA\bar{x}$$

$$A = (2 \times 3.61) = 7.22 \text{ m}^2 \; ; \bar{x} = 1 \text{ m}$$

$\therefore$ $$P_d = (9\,810 \times 7.22 \times 1)$$

$$= 70.828 \times 10^3 \text{ N} = 70.828 \text{ kN}$$

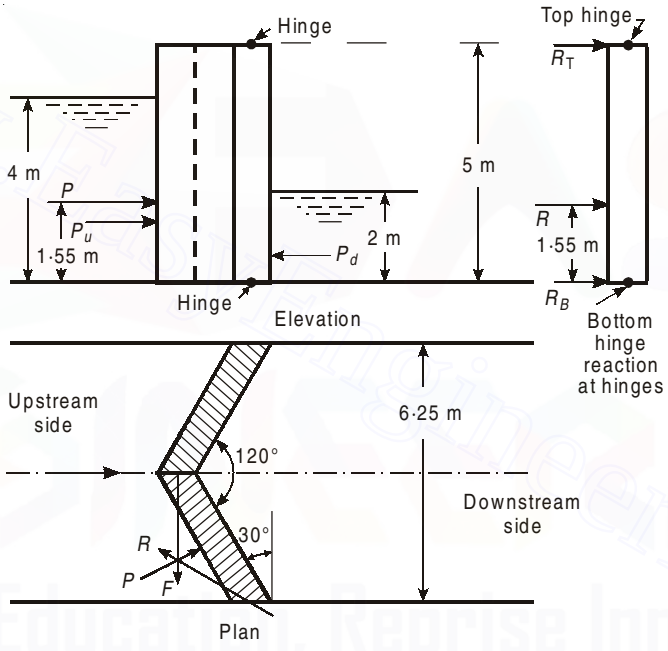


Figure Ex. 3.17

Similarly the depth of the centre of pressure on the downstream face is given by

$$\bar{h}_d = \bar{x} + \frac{I_G}{A\bar{x}}$$

or $$\bar{h}_d = 1 + \frac{\frac{1}{12} \times 3.61 \times (2)^3}{(3.61 \times 2) \times 1} = 1.33 \text{ m}$$

Thus the resultant water pressure on each gate is

$$P = (P_u - P_d)$$

$$= (283.313 - 70.828) = 212.485 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If x is the height of the point of application of the resultant water pressure on the gate, then by taking the moments, we get

$$P \times x = P_u \times (4 - \bar{h}_u) - P_d (2 - \bar{h}_d)$$

or $$212.485\ x = 283.313 \times (4 - 2.67) - 70.828\ (2 - 1.33)$$

$$\therefore \quad x = 1.55 \text{ m}$$

From Eqs 3.22 and 3.24, we have

$$F = R$$

and $$R = \frac{P}{2 \sin 30°} = P$$

i.e., $$R = F = P = 212.485 \text{ kN}$$

Now if R_T and R_B are the individual hinge reaction at the top and bottom hinges respectively, then

$$R_T + R_B = R = 212.485$$

Also by taking the moments of the hinge reactions about the bottom hinge, we get

$$R_T \times 5 = R \times 1.55$$

since the resultant hinge reaction is assumed to act at the same height as the resultant pressure.

Thus $$R_T \times 5 = 212.485 \times 1.55$$

$$\therefore \quad R_T = \frac{212.485 \times 1.55}{5} = 65.87 \text{ kN}$$

and $$R_B = (212.485 - 65.87) = 146.615 \text{ kN}$$

Example 3.18. (*a*) *A concrete dam retains water to a depth of 18 m. The face of the dam in contact with water is vertical for the first 6 m and there after it is inclined at 12° to the vertical in order to increase the thickness of the dam towards the base. Determine the magnitude and direction of the resultant water pressure per metre length of the dam and the depth of its point of application on the face of the dam.*

(*b*) *If the concrete weighs 23.55 kN/m*^{3}*, determine (i) the point of application of the resultant of the water pressure and the weight of the dam, on the base of the dam ; and (ii) normal stresses at the toe and heel of the dam.*

Solution

The horizontal water pressure P_H acting on the vertical projection of the dam MN is given by

$$P_H = WA\bar{x}$$
$$= 9\,810 \times (18 \times 1) \times 9$$
$$= 1\,589\,220 \text{ N}$$
$$= 1\,589.220 \text{ kN}$$

It will act at a height of 6 m above the base of the dam.

The vertical component P_V of the water pressure is equal to the weight of the prism of water 1 m long and having end area $LMNO$.

Area $$LMNO = \left(\frac{6+18}{2}\right)(12 \tan 12°) = 30.61 \text{ m}^2$$

$$\therefore \quad P_V = (9\,810 \times 1 \times 30.61)$$
$$= 300\,284 \text{ N} = 300.284 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

It will act at the centroid of the area *LMNO*, the distance of which from *MN* is given by

$$y = \left[\frac{(2\times 6)+18}{6+18}\right] \times \frac{12\tan 12°}{3}$$

$$= 1.06 \text{ m}$$

The resultant water pressure acting on the dam is

$$R = \sqrt{P_H^2 + P_V^2}$$

$$= \sqrt{(1589.220)^2 + (300.284)^2}$$

$$= 1\,617.341 \text{ kN}$$

Let the resultant water pressure meet the face of the dam at a height x above the base of the dam. Taking the moments about this point, we get

$$P_H (6 - x) = P_V \times (x \tan 12° - 1.06)$$

or $\quad 1\,598.220\,(6 - x) = 300.284\,(x \times 0.2126 - 1.06)$

$\therefore \quad x = 5.96$ m

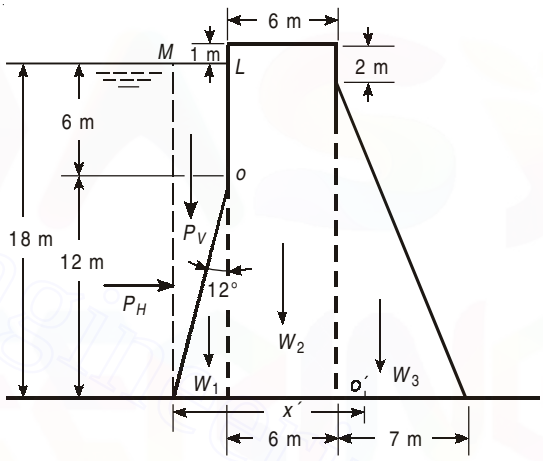


Figure Ex. 3.18

As shown in Fig. Ex. 3.18 the total weight of the dam W consists of the weights of the three portions, i.e.,

$$W = W_1 + W_2 + W_3$$

in which

$$W_1 = \left(\frac{1}{2}\times 12 \times 12\tan 12°\right) \times 1 \times 23.55$$

$$= 360.412 \text{ kN}$$

acting at a distance of $\left(\frac{2}{3}\times 12\tan 12°\right)$

$$= 1.70 \text{ m from N}$$

$$W_2 = (19 \times 6 \times 1) \times 1 \times 23.55$$

$$= 2\,684.700 \text{ kN}$$

acting at a distance of $(12 \tan 12° + 3)$

$$= 5.55 \text{ m from N}$$

$$W_3 = \left(\frac{1}{2}\times 7 \times 17\right) \times 1 \times 23.55$$

$$= 1\,401.225 \text{ kN}$$

acting at a distance of $\left(12\tan 12° + 6 + \frac{1}{3}\times 7\right)$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= 10.88$ m from N

$\therefore \quad W = 4\,446.337$ kN

Let x be the distance from N of the point of application of W, then by taking moments, we get

$(4\,446.337 \times x) = [(360.412 \times 1.70) + (2\,684.700 \times 5.55) + (1401.225 \times 10.88)]$

$\therefore \quad x = 6.92$ m

Now let the resultant of the water pressure and the weight of the dam meet the base at point O', which is assumed to be at a distance x' from the point N.

By taking moments of all the forces about O', we get

$$P_H \times 6 - Pv\,(x' - 1.06) - W\,(x' - 6.92) = 0$$

or $1\,589.220 \times 6 - 300.284\,(x' - 1.06) - 4\,446.337(x' - 6.92) = 0$

$\therefore \quad x' = 8.56$ m

The normal stress at the toe of the dam is given by

$$f_t = \frac{\sum V}{b}\left(1 + \frac{6e}{b}\right)$$

and that at the heel of the dam is given by

$$f_h = \frac{\sum V}{b}\left(1 + \frac{6e}{b}\right)$$

where $\sum V$ = sum of the vertical forces acting on the dam, b = base width of the dam and e = distance of point O' from the mid-point of the base (also called eccentricity)

In this case

$$\sum V = (P_V + W)$$
$$= (300.284 + 4\,446.337)$$
$$= 4\,746.621 \text{ kN}$$
$$b = (12 \tan 12° + 13) = 15.55 \text{ m}$$
$$e = \left(x' - \frac{b}{2}\right) = (8.56 - 7.78) = 0.78 \text{ m}$$

Thus
$$f_t = \frac{4746.621}{15.55}\left(1 + \frac{6 \times 0.78}{15.55}\right)$$
$$= 397.12 \text{ kN/m}^2 \text{ (compressive)}$$

and
$$f_h = \frac{4746.621}{15.55}\left(1 - \frac{6 \times 0.78}{15.55}\right)$$
$$= 213.38 \text{ kN/m}^2 \text{ (compressive)}$$

Example 3.19. *The sector gate shown in Fig. Ex. 3.19 consists of a cylindrical surface of which PN is the trace, supported by a structural frame hinged at M. The length of the gate perpendicular to paper is 9 m.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Determine the amount and location of the horizontal and vertical components of the total hydrostatic pressure on the gate.

Solution

The horizontal component of the total pressure on the curved gate is equal to the total pressure which would be exerted on a vertical projection of the curved surface.

The depth of this projection is (6 sin 60°) = 5.196 m and its width is 9 m. Its centroid is $\left(\frac{1}{2}\times 5.196\right)$ = 2.598 m below the free surface.

Therefore the horizontal component is

$$P_H = wA\bar{x}$$

$$= 9\,810 \times (9 \times 5.196) \times \left(\frac{1}{2}\times 5.196\right)$$

$$= 1\,191\,845 \text{ N} = 1\,191.845 \text{ kN}$$

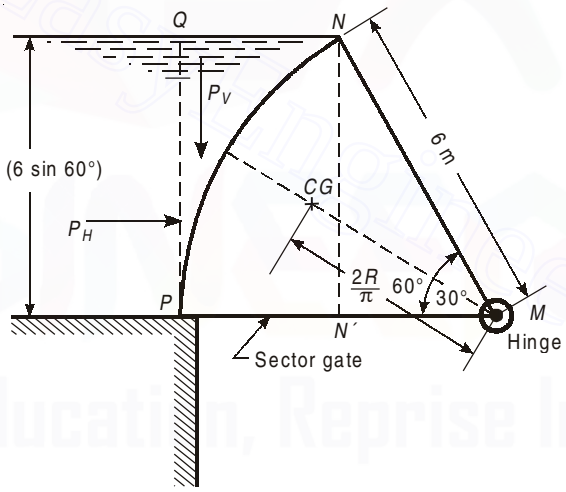


Figure Ex. 3.19

Its line of action passes through the centre of pressure of the vertical projection and hence its depth below the free surface is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 2.598 + \frac{\frac{1}{2}\times 9\times (5.196)^3}{(9\times 5.196)\times 2.598}$$

$$= 3.46 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The vertical component of the total pressure is equal to the weight of the prism of water 9 m long and having an end area *PQN*.

Area *PQN* = area of rectangle *PQNN''* + area of triangle *NN'M* – area of sector *PNM*

$$\therefore \quad \text{Area } PQN = (5.196 \times 3) + \left(\frac{1}{2} \times 3 \times 5.196\right) - \left(\frac{\pi}{6} \times 6 \times 6\right)$$

$$= (15.588 + 7.794 - 18.85) = 4.532 \text{ m}^2$$

$$\therefore \quad P_V = (9\,810 \times 9 \times 4.532)$$

$$= 400\,130 \text{ N} = 400.130 \text{ kN}$$

Its line of action passes through the centroid of the area *PQN*, which may be located by taking the moments about line *PQ*. Thus is *x* is the distance of the centroid of area *PQN* from *PQ*, then by taking the moment about *PQ*, as the centroid of a 60 circular sector is at a radial distance of $(2R/\pi)$ from the centre along the central radius, we get

$$4.532 \times x + \left(\frac{\pi}{6} \times 6 \times 6\right)\left(6 - \frac{2 \times 6}{\pi}\cos 30°\right)$$

$$= 5.196 \times 3 \times \frac{3}{2} + \frac{1}{2} \times 5.196 \times 3 \times \left(3 + \frac{3}{3}\right)$$

$$\therefore \quad x = \frac{3.815}{4.532} = 0.842 \text{ m}$$

The horizontal and vertical components are co-planar and therefore combine to give a single resultant force of magnitude

$$P = \sqrt{(1191.845)^2 + (400.13)^2}$$

$$= 1\,257.217 \text{ kN}$$

at an angle $\theta = \tan^{-1}\left(\frac{400.13}{1191.845}\right) = 18° 33'$ with the horizontal.

Since the total pressures on each of the elementary portions of the surface are normal to the surface, their lines of action pass through *M*. Therefore the line of action of the resultant force *P* also passes through *M*.

Note. On the basis of this consideration the location of P_V may also be determined graphically, since P_V also passes through the point of intersection of P_H and *P*.

Example 3.20. *The length of the tainter gate (shown in Fig. Ex. 3.20) perpendicular to the paper is 0.5 m. Find (a) the total horizontal push of water on the gate, (b) the total vertical component of water pressure against the gate; (c) the resultant water pressure on the gate and its location.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

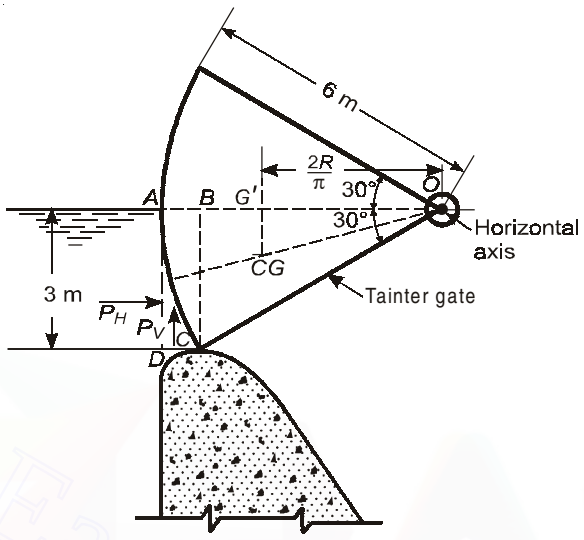


Figure Ex. 3.20

Solution

(a) The total horizontal push P_H, of water on the gate is equal to the total pressure on the vertical projection AD.

Thus

$$P_H = wA\bar{x}$$
$$A = (3 \times 0.5) = 1.5 \text{ m}^2;\ \bar{x} = 1.5 \text{ m}$$
$$P_H = (9\,810 \times 1.5 \times 1.5)$$
$$= 22\,073 \text{ N} = 22.073 \text{ kN}$$

Its line of action passes through the centre of pressure of the vertical projection and hence its depth below the free surface is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

$$= 1.5 + \frac{\frac{1}{12} \times 0.5 \times (3)^3}{(0.5 \times 3) \times 1.5} = 2.0 \text{ m}$$

(b) The total vertical component P_V of the water pressure against the gate (which is acting upwards in this case) is equal to the imaginary volume of water ABC

$$\text{Area } ABC = \text{area of sector } AOC - \text{area of triangle } BOC$$

$$\therefore \quad \text{Area } ABC = \left(\frac{\pi}{12} \times 6 \times 6\right) - \left(\frac{1}{2} \times 3\sqrt{3} \times 3\right)$$

$$= 1.63 \text{ m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $P_V = (9\,810 \times 1.63 \times 0.5)$

$= 7\,995 \text{ N} = 7.995 \text{ kN}$

Its line of action passes through the centroid of area *ABC*, which may be determined by taking moments about *O*. Thus if the distance of the centroid of area *ABC* is *x* from *O*, then since the distance *OG'* for the centroid of sector *OAC* is $\left(\frac{2\times 6}{\pi}\right)$, by taking the moments about *O*, we get

$$1.63\, x = \left(\frac{\pi}{12}\times 6\times 6\right)\left(\frac{2\times 6}{\pi}\right) - \left(\frac{1}{2}\times 3\sqrt{3}\times 3\right)\times\left(\frac{2}{3}\times 3\sqrt{3}\right)$$

$\therefore \quad x = 5.521$ m

So the distance of the line of action of P_V from *BC*

$$= \left(5.5213\sqrt{3}\right) = 0.325 \text{ m}$$

(c) As the components P_H and P_V are coplanar, these may be combined to give a single resultant force of magnitude

$$P = \sqrt{(22.073)^2 + (7.995)^2} = 23.476 \text{ kN}$$

at an angle $\theta = \tan^{-1}\left(\frac{7.995}{22.073}\right) = 19° 55'$ with the horizontal.

Again the line of action of *P* passes through point *O*, which may be obtained by joining the point of intersection of P_H and P_V and the point *O*.

Example 3.21. *A tank has a base 3 m square, from which four sides slope outward at 60° to the horizontal for a vertical height of 3 m, they then turn vertically upwards for another 3 m. The tank is filled with water of full depth of 6 m. Find the total pressure and the centre of pressure on one of the slopping sides of the tank.*

Solution

As shown in Fig. Ex. 3.24 each slopping face will be a trapezium.

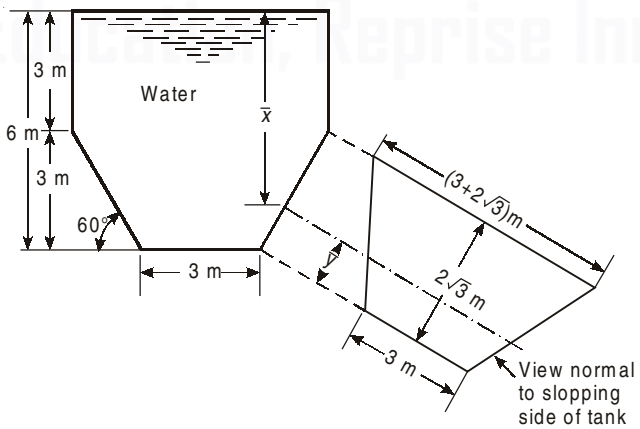


Figure Ex. 3.21

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The altitude of the trapezium

$$= \frac{3}{\sin 60°} = 3 \times \frac{2}{\sqrt{3}} = \left(2 \times \sqrt{3}\right) = 3.46 \text{ m}$$

Top width of the trapezium

$$= [3 + 2 \times (3 \cot 60°)] = \left(3 + 2\sqrt{3}\right) = 6.46 \text{ m}$$

The total pressure on the slopping face is given by

$$P = wA\bar{x}$$

$$A = \frac{1}{2}\left[3 + \left(3 + 2\sqrt{3}\right)\right] 2\sqrt{3} = 16.39 \text{ m}^{2j}$$

The inclined distance of the centroid of the trapezium from the base is given by

$$\bar{y} = \frac{\left[2\left(3 + 2\sqrt{3}\right) + 3\right]}{\left[\left(3 + 2\sqrt{3}\right) + 3\right]} \times \frac{2\sqrt{3}}{3} = 1.94 \text{ m}$$

$$\therefore \quad \bar{x} = \left[6 - \left(\bar{y} \sin 60°\right)\right]$$

$$= \left[6 - \left(1.94 \times \frac{\sqrt{3}}{2}\right)\right] = 4.32 \text{ m}$$

$$\therefore \quad P = (9\,810 \times 16.39 \times 4.32)$$

$$= 694\,595 \text{ N} = 694.595 \text{ kN}$$

The depth of the centre of pressure is given by

$$\bar{h} = \bar{x} + \frac{I_G \sin^2 \theta}{A\bar{x}}$$

$$I_G = \frac{[(6.46)^2 + 4(6.46 \times 3) + (3)^2]}{(6.46 + 3)} \times \frac{(3.46)^3}{36}$$

$$= 15.60 \text{ m}^4$$

and $\theta = 60°$

$$\therefore \quad \bar{h} = 4.32 + \frac{15.60 \times \left(\frac{\sqrt{3}}{2}\right)^2}{(16.39 \times 4.32)} = 4.49 \text{ m}$$

Example 3.22. *Calculate the minimum force F applied vertically, shown in Fig. Ex. 3.22, to keep the cover of the box, containing water, closed. The cover is 1 m wide perpendicular to the plane of the paper. The pressure gage at the bottom of the box reads 39.24 kN/m^2. Neglect the weight of the cover.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

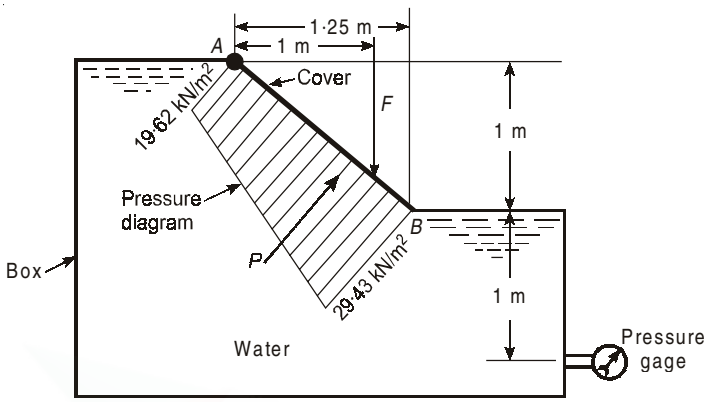


Figure Ex. 3.22

Solution

The pressure head in the tank in level with the pressure gage is equal to

$$h = \frac{p}{w} = \frac{39.24 \times 10^3}{9810} = 4 \text{ m}$$

The pressure heads at points A and B are 2 m and 3 m respectively.

Therefore, the pressure intensities at points A and B are

$$\begin{aligned} P_A &= (9\,810 \times 2) \\ &= 19\,620 \text{ N/m}^2 = 19.62 \text{ kN/m}^2 \\ p_B &= (9\,810 \times 3) \\ &= 29\,430 \text{ N/m}^2 = 29.43 \text{ kN/m}^2 \end{aligned}$$

The length of the cover $AB = \left[\sqrt{(1.25)^2 + (1)^2}\right] = 1.60$ m

The pressure diagram on the cover may be drawn as shown in the figure and the total pressure on the cover may be obtained as

$$P = \left(\frac{19.62 + 29.43}{2}\right)(1.60 \times 1) = 39.24 \text{ kN}$$

It will be acting normal to the cover at the centroid of the pressure diagram, the distance of which from the point A is

$$\bar{y} = \frac{[(2 \times 29.43) + 19.62]}{[19.62 + 29.43]} \times \frac{1}{3} = 0.853 \text{ m}$$

Thus by taking the moments of the forces about A and equating the sum of the moments to zero, we get

$$F \times 1 - P \times 0.853 = 0$$

or $$F - 39.24 \times 0.853 = 0$$

$\therefore$ $$F = 33.472 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 3.23. *A cylinder 2.4 m diameter weighs 2 kN and rests on the bottom of a tank which is 1 m long. As shown in Fig. Ex. 3.23 water and oil are poured into the left and right hand portions of the tank to depths 0.6 m and 1.2 m respectively. Find the magnitudes of the horizontal and vertical components of the force which will keep the cylinder touching the tank at B.*

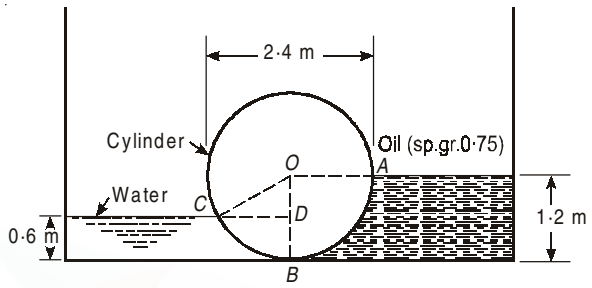


Figure Ex. 3.23

Solution

Net P_H = Component on *AB* to left–Component on *CB* to right

$$= \left[0.75 \times 9810 \times (1 \times 1.2) \times \frac{1.2}{2}\right] - 9810 \times (1 \times 0.6) \times \frac{0.6}{2}$$

$$= (5\,297 - 1\,766)$$

$$= 3\,531 \text{ N} = 3.531 \text{ kN to left}$$

Net P_V = Component upward on *AB* + component upward on *CB*

= Weight of quadrant of oil + weight (sector – triangle) of water

$$= 0.75 \times 9\,810 \times 1 \times \frac{1}{4}\left(\frac{\pi}{4} \times 2.4^2\right)$$

$$+ 9\,810 \times 1 \left[\frac{\pi}{6} \times 1.2^2 - \frac{1}{2} \times 0.6 \times \sqrt{(1.2)^2 - (0.6)^2}\right] = (8\,321 + 4\,338)$$

$$= 12\,659 \text{ N} = 12.659 \text{ kN upward}$$

The weight of the cylinder acting downward is equal to 2 kN

∴ Net downward force to hold the cylinder in place

= (12.659 – 2) = 10.659 kN

Thus the horizontal and vertical components of the force to hold the cylinder in place are 3.531 kN to the right and 10.659 kN downward.

Example 3.24. (*a*) *A spherical vessel is filled with water of weight Z kN. Show that the resultant fluid pressure on each of the halves into which it is divided by a vertical diametral plane is* $\left(Z\sqrt{13}/4\right)$ kN.

(*b*) *If the diametral plane is horizontal, show that the resultant fluid pressure on one half is 5 times that on the other.*

Solution

(a) Let *d* be the diameter of the sphere.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The horizontal component of the total pressure exerted on the half portion of the sphere is given by

$$P_H = wA\bar{x} = w\left(\frac{\pi d^2}{4}\right)\times\frac{d}{2} = \frac{w\pi d^3}{8}\text{ kN}$$

But $Z = w(\pi d^3/6)$

$\therefore$ $P_H = (3/4)Z$ kN

acting at the centre of pressure of the projected area.

The vertical component of the total pressure exerted on the half portion of the sphere is given by

P_V =Weight of the liquid contained in the half portion of the sphere acting vertically downwards through its centre of gravity.

Thus $P_V = (Z/2)$ kN

$\therefore$ $P = \sqrt{P_H^2 + P_V^2}$

$$= \sqrt{(3Z/4)^2 + (Z/2)^2}$$

$$= \left(Z\sqrt{13}/4\right)\text{ kN}$$

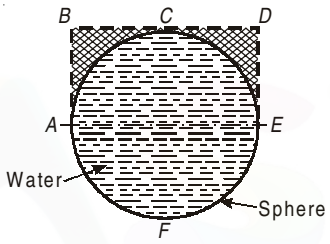


Figure Ex. 3.24

(b) When the diametral plane is horizontal then,

Resultant fluid pressure for the top half portion of the sphere

= Weight of water in cross hatched portion *ABCDECA*

$$= w\left(\frac{\pi d^2}{4}\times\frac{d}{2} - \frac{\pi d^3}{12}\right) = w\left(\frac{\pi d^3}{24}\right)\text{ kN}$$

Similarly, the resultant fluid pressure for the lower half portion of the sphere

= Weight of water in the portion *ABCDEFA*

$$= w\left(\frac{\pi d^2}{4}\times\frac{d}{2} + \frac{\pi d^3}{12}\right)$$

$$= 5w\left(\frac{\pi d^3}{24}\right)\text{ kN}$$

Hence it is proved that the resultant pressure on the lower half portion of the sphere is 5 times that on the upper half portion.

Example 3.25. *The cylindrical gate of a canal headworks, having a diameter of 3 m and a length of 6 m, is subjected to water pressure upto its top as shown in Fig. Ex. 3.25. The gate resisting on the concrete floor of the headworks, is laterally supported at A where the coefficient of friction* $\mu = 0.15$. *Assuming watertight condition at B and no rotation of the cylinder, find the minimum weight of the gate so that it may have no upward motion resulting from the water pressure.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

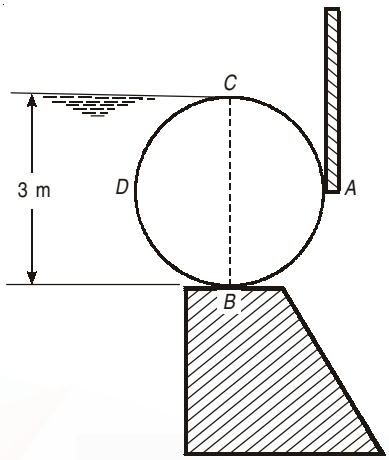


Figure Ex. 3.25

Solution

The reaction R at A is due to the horizontal component P_H of the water pressure acting on the cylinder

or $$R = P_H = \left(9810 \times 6 \times 3 \times \frac{3}{2}\right) = 264\,870 \text{ N} = 264.87 \text{ kN}$$

which will be acting towards left since P_H is acting towards right.

As the gate tends to have upward motion, the frictional resistance F acting downwards will be developed at A which may be obtained as

$$F = \mu R = (0.15 \times 264.87)$$
$$= 39.73 \text{ kN}$$

The upward motion of the gate will be caused by the vertical component P_V of the water pressure acting on the cylinder in the upward direction which is given as

$$P_V = \text{Weight of water in the portion } BDC$$
$$= 9\,810 \times 6 \times \frac{1}{2}\left(\frac{\pi}{4} \times 3 \times 3\right)$$
$$= 208\,028 \text{ N} = 208.028 \text{ kN}$$

The upward motion of the gate will be opposed by the weight W of the gate and the frictional resistance F at A, both these forces acting in the downward direction. Thus considering the limiting condition of the equilibrium of the gate in the vertical direction, we get

$$W + 39.73 = 208.028$$
$$\therefore \quad W = 168.298 \text{ kN}$$

Example 3.26. *Figure Ex. 3.26 shows the cross-section of a tank full of water under pressure. The length of the tank is 2 m. An empty cylinder lies along the length of the tank on one corner as shown in the figure. Find the magnitude and location of the horizontal and vertical components of the force acting on the curved surface ABC*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the cylinder. Also compute the magnitude of this force and locate its line of action as well as point of application.

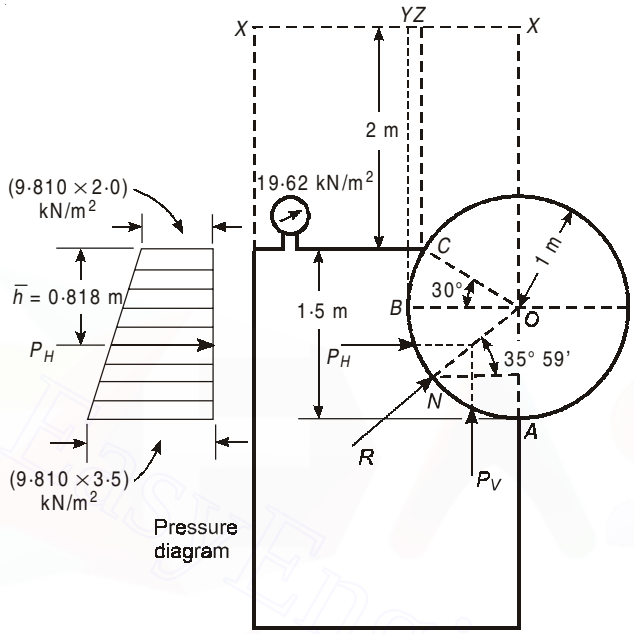


Figure Ex. 3.26

Solution

The water in the tank is under pressure, and the pressure at the top of the tank is 19.62 kN/m^2, which is equivalent to imaginary free surface of water in the tank being upto *XX*; where *XX* is at 2 m $\left(= \frac{19.62 \times 10^3}{9810}\right)$ above the top of the tank.

The horizontal component of the total pressure on the curved surface *ABC* is equal to the total pressure which would be exerted on the vertical projection of the curved surface, which is given by the area of the pressure diagram drawn on the left side of the figure. Thus horizontal component is

$$P_H = \left[9.810 \times \left(\frac{2+3.5}{2}\right) \times 1.5\right] \times 2 = 80.933 \text{ kN}$$

It is acting at the centroid of the pressure diagram. Thus if $\bar{h}$ is the depth of the centroid of the pressure diagram, then

$$\bar{h} = \left[\left(\frac{2 \times 3.5 + 2}{2 + 3.5}\right) \times \frac{1.5}{3}\right] = 0.818 \text{ m}$$

The vertical component of the total pressure on the curved surface *ABC* consists of the upward

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

pressure acting on concave upward part *AB* and the downward pressure on the concave downward part *CB*. The net vertical component is the algebraic sum of the upward force and the downward force.

Upward force P_{V_1} = Weight of liquid (real or imaginary) above curve *AB*

= 9 810 × 2 × (area of sector *AOB* + area of rectangle *BOXY*)

Downward force P_{V_2} = Weight of liquid (real or imaginary) above curve *BC*

= 9 810 × 2 × (area *BCZY*)

Thus net vertical component of the total pressure on the curved surface *ABC* which is acting upward is

$$P_V = 9\,810 \times 2 \times (\text{area } ZCBAX)$$

$$= 9\,810 \times 2 \times (\text{area of trapezium } ZCOX + \text{area of sector } ABCO)$$

$$= 9\,810 \times 2 \times \left[\left(\frac{2+2.5}{2}\right) \times \frac{\sqrt{3}}{2} + \frac{1}{2} \times \frac{2\pi}{3} \times (1)^2\right]$$

$$= 9\,810 \times 2 \times (1.949 + 1.047)$$

$$= 58\,782 \text{ N}$$

$$= 58.782 \text{ kN}$$

Its line of action passes through the centroid of the area *ZCBAX* which may be located as follows.

It $\bar{h}_1$ is the distance from *OX* of the centroid of the trapezium *ZCOX*, then

$$\bar{h}_1 = \frac{1}{3} \times \frac{\sqrt{3}}{2}\left[\frac{2 \times 2 + 2.5}{2 + 2.5}\right] = 0.417 \text{ m}$$

Similarly if $\bar{h}_2$ is the distance of the centroid of the sector *ABCO* from *AO* then

$$\bar{h}_2 = \frac{3r}{2\pi} = \frac{3 \times 1}{2\pi} = 0.477 \text{ m}$$

(which may be obtained by using the method of locating the centroid of an area by integration).

Now if $\bar{h}_y$ is the distance from *AX* of the centroid of the area *ZCBAX* then by taking moments about *AX*, we get

$$(1.949 + 1.047)\ \bar{h}_y = (1.949 \times 0.417) + (1.047 \times 0.477)$$

$$\therefore \quad \bar{h}_y = 0.438 \text{ m}$$

The magnitude of the resultant force is obtained as

$$R = \sqrt{(80.933)^2 + (58.782)^2} = 100.027 \text{ kN}$$

which is acting at an angle θ with the horizontal, where

$$\theta = \tan^{-1}\left(\frac{58.782}{80.933}\right) = 35° 59'$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since the total pressure on each of the elementary portion of the curved surface is normal to the surface, its line of action passes through the centre *O*. Therefore the force *R* should also pass through the centre *O*. In order to confirm this statement, take the moments of the horizontal and vertical components about *O*, which gives

$$\Sigma MO = [80.933(0.818 - 0.5) - 58.782(0.438)] = 0$$

The point of application *N* of the force *R* is located on the surface of the cylinder which is at a distance [1× cos 35°59'] = 0.8092 m from *OA* towards left and [1 × sin 35° 59'] = 0.5876 m below *OB*.

Example 3.27. *A gate closing an opening is triangular in section, as shown in Fig. Ex. 3.27. The gate is 1m long (in the direction perpendicular to the plane of the paper) and it is made of concrete weighing 24 kN/m³. If the gate is hinged at the top and freely supported at one of the bottom ends, find the height of water h on the upstream side when the gate will just be lifted.*

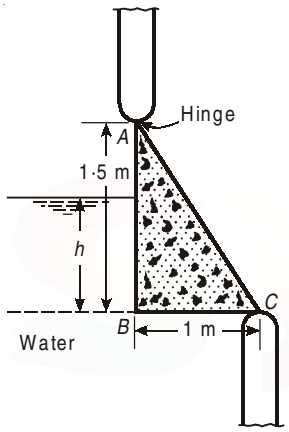


Figure Ex. 3.27

Solution

The weight of the gate is $W = \left(\frac{1}{2} \times 1.5 \times 1 \times 1 \times 24\right) = 18$ kN which is acting at $\frac{1}{3}$ m from *AB*.

The total pressure acting on the vertical surface *AB* of the gate is

$$P_H = 810 \times 1 \times h \times \frac{h}{2}$$
$$= 4\,905\, h^2 \text{ N} = 4.905\, h^2 \text{ kN}$$

which is acting at $\frac{h}{3}$ above *BC*.

The total pressure on the horizontal surface *BC* of the gate is

$$P_V = 9\,810 \times 1 \times 1 \times h$$
$$= 9\,810\, h\, N = 9.81h \text{ kN}$$

which is acting at $\frac{1}{2}$ m from *AB*.

When the gate is just to be lifted the algebraic sum of the moments of all the forces about the hinge *A* is equal to zero

$$\text{Thus, } 18 \times \frac{1}{3} - 4.905\, h^2 \left(1.5 - \frac{h}{3}\right) - 9.81h \times \frac{1}{2} = 0$$

or
$$h^3 - 4.5\, h^2 - 3h + 3.67 = 0$$

Solving it by trial $h = 0.662$ m.

Example 3.28. *A cylinder of radius 0.3 m is located in water as shown in Fig. Ex. 3.28 (a). The cylinder and the wall are smooth. For a 1.5 m length of cylinder, find*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) its weight,

(ii) the resultant force exerted by the wall on the cylinder,

(iii) the resultant moment around the centre of the cylinder due to water forces on the cylinder.

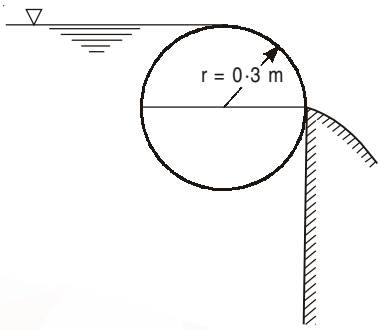


Figure Ex. 3.28 (a)

Solution

Since the cylinder and the wall are smooth the weight of the cylinder is equal to the vertical component of the water pressure exerted on the cylinder which consists of the following:

(a) the vertical component of the water pressure exerted on the portion *ABC*, P_{V_1}, which is equal to the weight of water in the portion *ABC* acting vertically upwards at the centroid of the area *ABC*; and

(b) the vertical component of the water pressure exerted on the portion *CD*, P_{V_2}, which is equal to the weight of water in the portion *ACDE* acting vertically upwards at the centroid of the area *ACDE*.

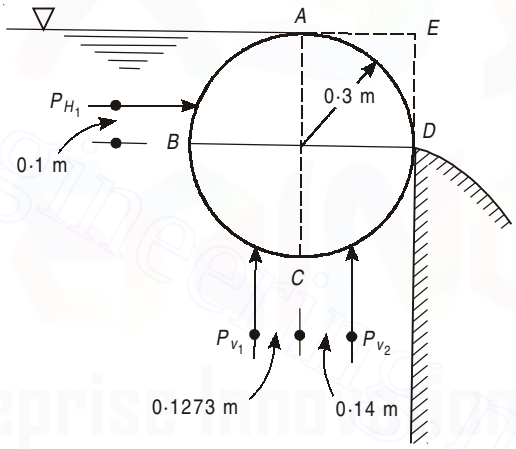


Figure Ex. 3.28

Thus

$$P_{V_1} = \left[9.81 \times \frac{1}{2}\pi(0.3)^2 \times 1.5\right] \text{kN}$$

$$= 2.0803 \text{ kN}$$

which acts vertically upwards at the centroid of the area *ABC*.

The distance of the centroid of the area *ABC* from *AC*

$$= \frac{4r}{3\pi} = \frac{4 \times 0.3}{3\pi} = 0.1273 \text{ m}$$

$$P_{V_2} = \left[9.81 \times \frac{1}{4}\pi(0.3)^2 \times 1.5 + 9.81 \times 0.3 \times 0.3 \times 1.5\right] \text{kN}$$

$$= (1.0401 + 1.3244) \text{ kN}$$

$$= 2.3645 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which acts vertically upwards at the centroid of the area *ACDE*.

If x is the distance of the centroid of the area *ACDE* from *AC*, then

$$2.3645x = 1.0401 \times \frac{4 \times 0.3}{3\pi} + 1.3244 \times \frac{0.3}{2}$$

or $$x = 0.14 \text{ m}$$

∴ Weight of the cylinder

$$= P_{V_1} + P_{V2}$$
$$= (2.0803 + 2.3645) \text{ kN}$$
$$= 4.4448 \text{ kN}$$

The horizontal component of the water pressure exerted on the cylinder consists of the following:

(a) total pressure P_{H_1} acting horizontally from left to right on the vertical projection of the curved surface *AB*,

(b) total pressure P_{H_2} acting horizontally from left to right on the vertical projection of the curved surface *BC*; and

(c) total pressure P_{H_3} acting horizontally from right to left on the vertical projection of the curved surface *CD*.

Since P_{H_2} and P_{H_3} are equal and are acting in opposite directions, the net horizontal component of the water pressure exerted on the cylinder is equal to P_{H_1} which is given by

$$P_{H_1} = \left(9.81 \times 0.3 \times 1.5 \times \frac{0.3}{2}\right) \text{ kN}$$
$$= 0.6622 \text{ kN}$$

which is acting at a depth of 0.2 m below the water surface, or at a height of 0.1 m above *BD*.

Again since the cylinder and the wall are smooth the resultant force exerted by the wall on the cylinder is equal to the resultant water pressure exerted on the cylinder which is equal to

$$P = \sqrt{(0.6622)^2 + (4.4448)^2}$$
$$= 4.4939 \text{ kN}$$

The resultant moment around the centre of the cylinder due to water forces on the cylinder

$$= 0.6622 \times 0.1 + 2.0803 \times 0.1273 - 2.3645 \times 0.14$$
$$= 0$$

Note: Since the water pressure exerted on each of the elementary portion of the curved surface of the cylinder is normal to the surface, its line of action passes through the centre of the cylinder, and hence the moment of the resultant water pressure around the centre of the cylinder is equal to zero.

Example 3.29. *A vertical gate 5 m × 2.5 m size weighing 0.5 tonnes slides along guides (coefficient of friction 0.25) fitted on the side walls of an overflow spillway and its crest. What force will have to be exerted at the hoisting mechanism to lift the gate when the head of water over the crest is 2 m?*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

The reaction R at the guides is equal to the total pressure P acting on the gate.

Thus

$$R = P = \left[1000 \times (5 \times 2.0) \times \frac{2.0}{2}\right] \text{ kg}$$

$$= 10000 \text{ kg} = 10 \text{ tonnes}$$

When the gate is lifted up, frictional resistance F acting vertically downwards along the guides will be developed which may be obtained as

$$F = \mu R$$

$$= (0.25 \times 10) = 2.5 \text{ tonnes}$$

The upward motion of the gate will be opposed by the weight of the gate and the frictional resistance, and hence the force to be exerted at the hoisting mechanism to lift the gate

$$= (0.5 + 2.5) = 3.0 \text{ tonnes}$$

SUMMARY OF MAIN POINTS

1. When a static mass of fluid comes in contact with a surface, either plane or curved, a force is exerted by the fluid on the surface. This force is known as 'total pressure'.
2. Since for a fluid at rest no tangential force exists, the total pressure acts in the direction normal to the surface.
3. The point of application of the total pressure on the surface is known as 'centre of pressure'.
4. The total pressure P acting on a plane surface of area A held wholly submerged in a static mass of liquid of specific weight w is given by

$$P = wA\bar{x}$$

where

$\bar{x}$ = vertical depth of the centroid of the surface area from the free surface of the liquid.

The above equation gives the total pressure acting on a plane surface wholly submerged in a static mass of liquid and held in either horizontal position or vertical position or inclined position.

5. The depth of the centre of pressure for a plane surface wholly submerged in a static mass of liquid, and held in vertical position is given by

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$$

where

$\bar{h}$ = vertical depth of the centre of pressure below the free surface of the liquid;

I_G = moment of inertia of the area about a horizontal axis passing through the centroid of the area; and A and $\bar{x}$ are same as defined earlier.

6. The depth of the centre of pressure for a plane surface wholly submerged in a static mass of liquid, and held in inclined position is given by

$$\bar{h} = \bar{x} + \frac{I_G \sin^2 \theta}{A\bar{x}}$$

where

$\bar{h}$ = vertical depth of the centre of pressure below the free surface of the liquid;

I_G = moment of inertia of the area about a horizontal axis passing through the centroid of the area;

θ = angle of an inclination of the plane of the plane of surface with the horizontal and;

A and $\bar{x}$ are same as defined earlier.

7. The total pressure p acting on a curved surface wholly submerged in a static mass of liquid is given by

$$P = \sqrt{(P_H)^2 + (P_V)^2}$$

where

P_H = total pressure on the projected area of the curved surface on the vertical plane; and

P_V = the weight of the liquid actually or imaginary contained in the portion

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

extending vertically above the curved surface up to the free surface of the liquid.

The direction of the total pressure P acting on the curved surface is given by

$$\tan\theta = \frac{P_V}{P_H}$$

where

θ = the angle made by the total pressure P with the horizontal.

8. The total pressure acting on a sluice gate is given by

$$P = P_1 - P_2$$

where

P_1 = total pressure acting on the upstream side of the sluice gate ; and

P_2 = total pressure acting on the downstream side of the sluice gate.

9. For lock gates the reaction F between the two gates is equal to the reaction R at the hings ; i.e. $F = R$. Further the reaction F between the two gates is also given by

$$F = \frac{P}{2\sin\theta}$$

where

P = total pressure acting on each gate, (= $P_1 - P_2$) ; and

θ = inclination of each gate with the normal to the side of the lock.

PROBLEMS

3.1 Explain the terms — total pressure and centre of pressure.

3.2 Show that the centre of pressure of any lamina immersed under liquid is always below its centroid.

3.3 What is the position of the centre of pressure for a vertical semi-circular plane submerged in a homogeneous liquid with its diameter d at the free surface?

$$\left[\textit{Ans.}\ \text{On centreline at depth}\ \frac{3\pi d}{32}\right]$$

3.4 A trapezoidal plate having its parallel sides equal to $2a$ and at a distance h apart is immersed vertically in a liquid with $2a$ side uppermost and at a distance h below the surface of the liquid. Find the thrust on the surface and the depth of the centre of pressure.

$$\left[\textit{Ans.}\ \frac{13wah^2}{6}\ ;\ \frac{3h}{2}\right]$$

3.5 A triangular gate which has a base of 1.5 m and an altitude of 2 m lies in a vertical plane. The vertex of the gate is 1m below the surface in a tank which contains oil of specific gravity 0.8. Find the force exerted by the oil on the gate and the position of the centre of pressure.

[*Ans.* 27.468 kN or 2 800 kg(f); 2.43 m]

3.6 A circular gate in a vertical wall has a diameter of 4 m. The water surface on the upstream side is 8 m above the top of the gate and on the downstream side 1 m above the top of the gate. Find the forces acting on the two sides of the gate and the resultant force acting on the gate and its location.

[*Ans.* 1.257×10^5 kg (f) (1.23 MN); 3.77×10^4 kg (f) (0.37MN); 8.80×10^4 kg (f) (0.86 MN) ; 2 m below the top of the gate]

3.7 A vertical shutter revolving about a horizontal axis sustains a pressure of 4 m of water on one side. At what depth should the axis be placed in order that the pressure on the portions of the shutter above and below the axis may be equal?

[*Ans.* 2.828 m]

3.8 A rectangular sluice gate 2 m wide and 2 m deep, having its upper edges at a depth of 1.5 m is inclined at 45° to the vertical. The sluice gate is lifted by a force applied parallel to its plane. Determine the magnitude of the lifting force with a coefficient of friction $\mu = 0.15$ between the gate and its grooves. Neglect the weight of the gate.

[**Hint:** Determine the total pressure P on the gate and the required force is $F = \mu P$]

[*Ans.* 1 324.2 kg(f) (12.99 kN)]

3.9 The lower corner of a water tank has the shape of a quadrant of a circle of radius 1 m. The

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

water surface is 2 m above the centre of curvature. Consider a section 3.5 m long and calculate magnitude, direction and location of the resultant force exerted on the curved surface.

[*Ans.* 13.10×10^3 kg(f) (128.52 kN) passing through centre of curvature and inclined at 48°6' with horizontal]

3.10 A cubical vessel whose capacity is 19.683 m^3 is filled with water; a vertical tube, also filled with water, having internal diameter x mm and length 1.5 m is inserted in the top. Find the respective total pressures against the bottom and one of the sides of the vessel.

[*Ans.* 300.363 kN {30,618 kg(f)} ; 203.817 kN {20,777 kg(f)}]

3.11 How thick is the layer of liquid mud of specific gravity 1.6, at the bottom of a tank with water 7.5 m deep above it, if there is a pressure of 490.5 kN/m^2 [5 kg(f)/cm^2] against the bottom of the tank ? [*Ans.* 26.56 m]

3.12 A tank with vertical sides is square in plan with side 3.5 m long . It contains oil of specific gravity 0.9 to a depth of 1 m floating on 0.75 m depth of water. Calculate the pressure on one side of the tank and also determine the height of the centre of pressure above the base of the tank.

[*Ans.* 4921.875 kg(f) (48.284 kN) ; 0.577 m]

3.13 A lock gate 15 m high and 7.5 m wide is hinged horizontally at the bottom and maintained in the vertical position by horizontal chains at top. Sea water stands at a depth of 10 m on one side and 7.5 m on the other. Find the total tension in chains. Specific gravity of sea water is 1.03.

[*Ans.* 4.961×10^4 kg(f) (4.867×10^5 N)]

3.14 A rectangular opening in a vertical water face of a dam impounding water is closed by a gate mounted on horizontal trunnions parallel to the longest edges of the gate and passing through the centre of the shorter vertical edges. If the water level is above the top edge of the gate, show that the torque required to keep the gate closed is independent of the water level. Determine the magnitude of the torque when the gate is 1.5 m high and 1 m deep.

[*Ans.* 1 226.25 N m {125 kg(f)-m}]

3.15 Water stands to a depth of 30 m at the face of a concrete dam which is vertical for the first 10 m below the water level, slopes at an angle of $\tan^{-1}\frac{3}{10}$ to the vertical for the next 10 m and slopes at angle of $\tan^{-1}\frac{1}{5}$ to the vertical for the remaining 10 m. Determine completely the resultant pressure per metre length of the dam.

[*Ans.* 481.115×10^4 N{49.04×10^4 kg(f)} acting at a depth of 20.74 m below water surface at $\angle\theta = \tan^{-1}(13/30) = 23°\,26'$ with horizontal]

3.16 A canal lock is 6 m wide and has two vertical gates which make an angle of 120° with each other. The depth of water on the two sides of the gates are 10 m and 3 m respectively. Each gate is supported on two hinges, the lower one being 0.6 cm above the bottom of the lock. Neglecting the weight of the gates themselves, calculate the thrust between the gates and the height of the upper hinges if the forces on them are to be half of those on the lower hinges.

[*Ans.* 1.546 2 MN {157 612 kg(f)} ; 9.495 m]

3.17 The profile of the upstream face of a sea-wall in contact with water is parabolic with the equation $2y = x^2$, where y m is the height above the base and x m is the horizontal distance of the face from the vertical reference line. The water level is 4.5 m above the base. Determine the total thrust per metre length of the wall due to water pressure, its inclination to the vertical and the point where the line of action of this force intersects the free water surface. Take specific gravity of sea water as 1.025.

[*Ans.* 13.89×10^3 kg(f) (136.217 kN) inclined at $\angle\theta = \tan^{-1}(9/8) = 48°22'$ with vertical and intersects the water surface at 5.25 m from the upstream face of the sea-wall]

3.18 A sector gate of radius 4 m and length 5 m controls the flow of water in a horizontal channel. For the conditions shown in Fig. P. 3.18 determine the total thrust on the gate.

[*Ans.* 20.97×10^3 kg(f) (205.713 kN) passing through the centre and inlined at $\angle\theta = \tan^{-1}(0.3152) = 17°30'$ with horizontal]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

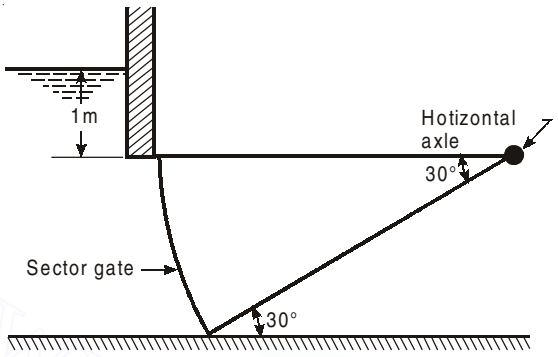


Figure P. 3.18

3.19 For a 2.5 m length of gate *AB* shown in Fig. P. 3.19 below, find the compression in strut *CD* due to water pressure if points *B*,*C* and *D* are hinged. [*Ans.* 6.92 × 10^3 kg(f) (67.885 kN)]

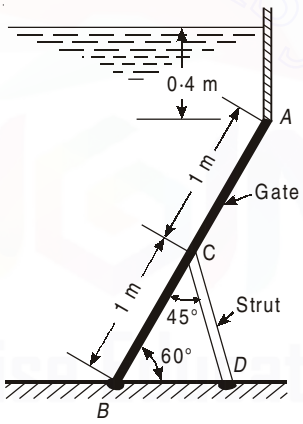


Figure P. 3.19

3.20 Find the dimension *Z* so that the total compression in rod *BD* shown in Fig. P. 3.20 will not be more than 88.29 kN [9´10^3 kg(f)]. The length of the board perpendicular to the paper may be taken as 1.25 m. [*Ans.* 1.94 m]

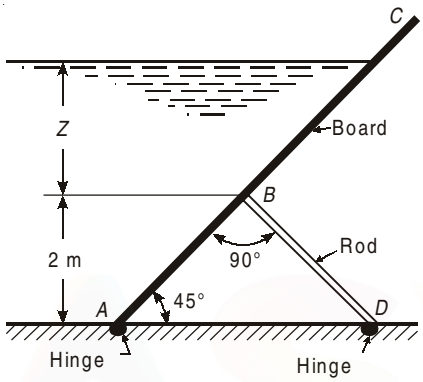


Figure P. 3.20

3.21 Calculate the depth of centre of pressure below the free surface of water of a plane lamina having a shape of a circular ring with outer and inner diameters 2*a* and a respectively. The lamina is immersed vertically with its top edge below the free surface at depth *a*.

[*Ans.* 2.156 *a*]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Buoyancy and Floatation

Chapter 4

4.1 BUOYANCY, BUOYANT FORCE AND CENTRE OF BUOYANCY

When a body is immersed in a fluid either wholly or partially it is subjected to an upward force which tends to lift (or buoy) it up. This tendency for an immersed body to be lifted up in the fluid, due to an upward force opposite to the action of gravity, is known as *buoyancy.* The force tending to lift up the body under such conditions is known as *buoyant force* (*or force of buoyancy or upthrust*). The point of application of the force of buoyancy on the body is known as *centre of buoyancy.*

The magnitude of the buoyant force can be determined by the well-known *Archimedes' principle,* which states that *when a body is immersed in a fluid either wholly or partially, it is buoyed or lifted up by a force which is equal to the weight of the fluid displaced by the body.* It is due to this upward force acting on a body immersed in a fluid, either wholly or partially, that there occurs an apparent loss in the weight

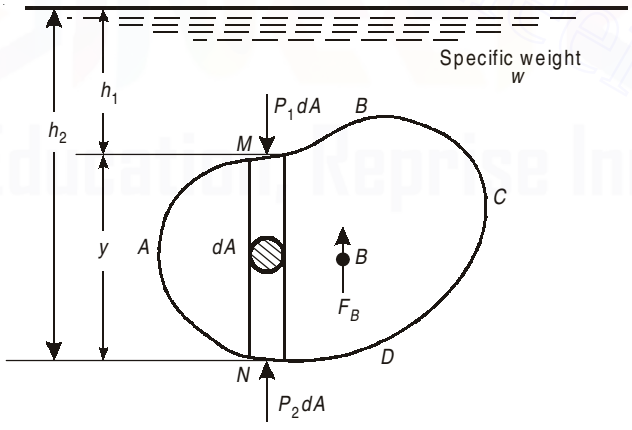


Figure 4.1 *Buoyant force on a wholly submerged body*

of the body. According to Archimedes' principle it is therefore known that the buoyant force is equal to the weight of the fluid displaced by the body. The buoyant force is exerted on a body immersed in a fluid, either wholly or partially, on account of the various portions of the boundary surface of the body being exposed to varying pressure intensities, which may be demonstrated by considering the pressure exerted by the surrounding fluid on the boundary surface of a submerged body as discussed below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider a wholly submerged body *ABCD* as shown in Fig. 4.1. The resultant horizontal pressure on the body is equal to zero because the horizontal pressure on the vertical projection from both sides are equal in the magnitude and opposite in direction. In order to obtain the resultant force acting on the body in the vertical direction, consider an elementary vertical prism *MN* of cross-section *dA* out of the whole body. On the top end of the prism *MN* a pressure force ($p_1\, dA$) is exerted which is acting in the vertical downward direction. Similarly on its bottom end a pressure force ($p_2 dA$) is exerted which is acting in the vertical upward direction. As shown in Fig. 4.1, $p_1 = wh_1$, $p_2 = wh_2$ and $(h_2 - h_1) = y$, where w is the specific weight of the fluid. Further since $p_2 > p_1$, the difference between the upward and downward pressure forces is a net upward force which is equal to the buoyant force dF_B on the vertical prism *MN*. Thus

$$dF_B = (p_2\, dA - p_1\, dA) = w\,(h_2 - h_1)\, dA = wy dA$$

If $d\bar{V}$ represents the volume of the vertical prism *MN* then

$$d\bar{V} = (y dA)$$

and

$$dF_B = w\, d\bar{V}$$

The buoyant force F_B on the entire submerged body *ABCD* is obtained by integrating dF_B. Therefore

$$F_B = \int dF_B = \int w d\bar{V} = (w\bar{V}) \qquad ...(4.1)$$

where $\bar{V}$ is the volume of the submerged body. Equation 4.1 thus indicates that the buoyant force exerted on a submerged body is equal to the weight of the fluid displaced by the submerged body. The buoyant force acts vertically upwards through the centre of buoyancy which evidently coincides with the centroid of the volume of the fluid displaced. Moreover for a wholly submerged body of homogeneous composition the centre of buoyancy will coincide with the centre of gravity of the body.

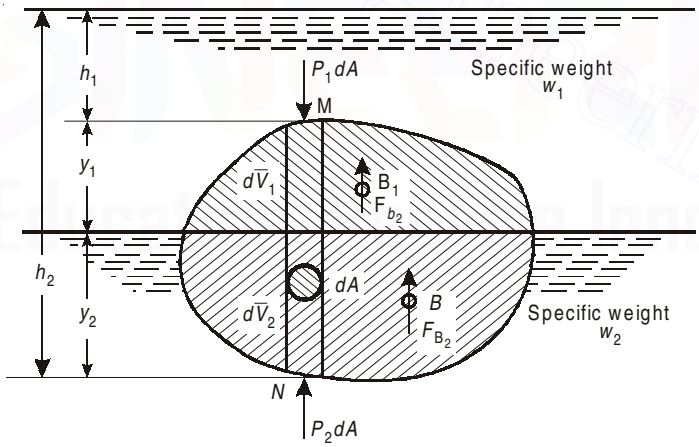


Figure 4.2 *Buoyant forces on a body floating at the surface of separation between two fluids*

When a body floats at a surface of separation (or interface) between two immiscible fluids of specific weights w_1 and w_2 as shown in Fig. 4.2, the buoyant force dF_B on a vertical prism MN of cross-section dA is

$$dF_B = (p_2 dA - p_1 dA)$$
$$= [w_1\,(h_1 + y_1) + w_2\, y_2 - (w_1\, h_1)]dA$$

or

$$dF_B = (w_1\, y_1 + w_2\, y_2)dA$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$dF_B = (w_1\, d\bar{V}_1 + w_2\, d\bar{V}_2)$$

in which $d\bar{V}_1$ and $d\bar{V}_2$ represent volumes of the elementary prism submerged in fluids of specific weights w_1 and w_2 respectively. The total buoyant force is

$$F_B = \int dF_B = \int (w_1\, d\bar{V}_1 + w_2 d\bar{V}_2)$$
$$= (w_1 \bar{V}_1 + w_2 \bar{V}_2)$$

which is the total weight of the displaced fluids. In this case it is therefore observed that the total buoyant force consists of the sum of the two components which correspond to the weight of the displaced fluids. The centre of buoyancy for each of these components obviously coincides with the centroid of the corresponding volume of fluid displaced, and may be located independently.

When a body is floating at the free surface of a liquid, it remains partially submerged in the liquid (as shown in Fig. 4.3), with the top portion of the body remaining in contact with air and its bottom portion being submerged in the liquid. In this case since the specific weight of air is negligible as compared with the specific weight of liquid, the weight of the air displaced by the top portion of the body may be neglected. Therefore same considerations as above show that the buoyant force exerted on a partially submerged body is, equal to the weight of the liquid displaced by the body, acting at the centre of buoyancy which obviously coincides with the centroid of the volume of the liquid displaced.

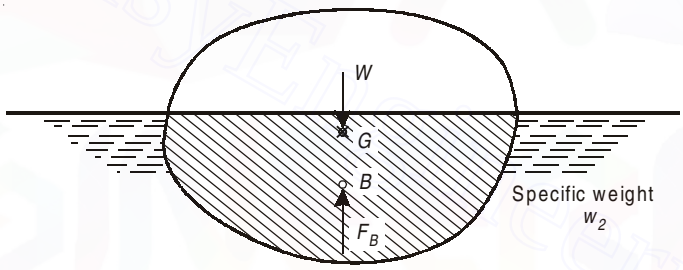


Figure 4.3 *Buoyant force on a body floating at the free surface of the liquid*

For a body immersed (either wholly or partially) in a fluid, the self-weight of the body always acts in the vertical downward direction. As such if a body floating in fluid is to be in equilibrium the buoyant force must be equal to the weight of the body i.e.,

$$F_B = W \qquad ...(4.2)$$

Further the lines of action of both, the buoyant force and the weight of the body must lie along the same vertical line, so that their moment about any axis is zero.

Equation 4.2 represents the *principle of floatation* which states that *the weight of a body floating in a fluid is equal to the buoyant force which in turn is equal to the weight of the fluid displaced by the body.* Thus for a body immersed in a fluid if the buoyant force exceeds the weight of the body, the body will rise until its weight equals the buoyant force. On the other hand if the weight of an immersed body exceeds the buoyant force, the body will tend to move downward and it may finally sink.

4.2 METACENTRE AND METACENTRIC HEIGHT

Consider a body floating in a liquid. If it is statically in equilibrium, it is acted upon by two forces viz., the weight of the body W acting at the centre of gravity G of the body and the buoyant force F_B

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

acting at the centre of buoyancy B. The forces F_B and W are equal and opposite and as shown in Fig. 4.4 the points G and B lie along the same vertical line which is the vertical axis of the body.

Let this body be tilted slightly or it undergoes a small angular displacement (or angle of heel) θ. It is assumed that the position of the centre of gravity G remains unchanged relative to the body. The centre of buoyancy B, however, does not remain fixed relative to the body. This is so because as the body is tilted the portion of the body immersed on the right hand side increases while that on the left hand side decreases and hence the centre of buoyancy moves to a new position B_1. In the tilted position of the body the buoyant force acts in a vertical upward direction at B_1. Now if a vertical line is drawn through the new centre of buoyancy B_1 (or in other words the line of action of the buoyant force is extended) it intersects the axis of the body BG at point M, which is known as *metacentre.* Thus metacentre

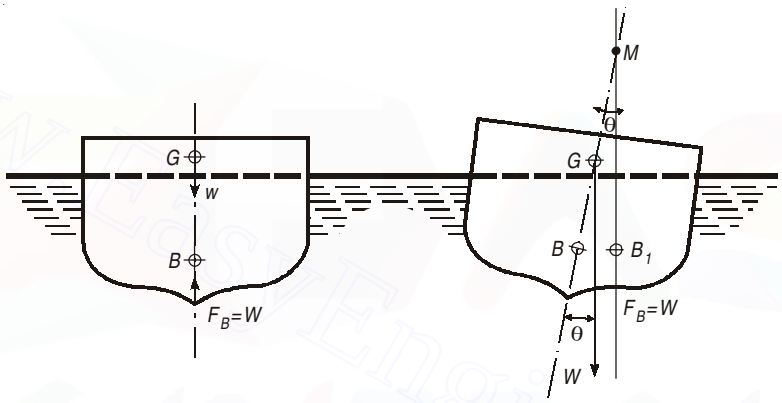


Figure 4.4 *Metacentre for a floating body*

may be defined as the point of intersection between the axis of the floating body passing through the points B and G and a vertical line passing through the new centre of buoyancy B_1. For small values of the angle of heel θ, the position of the metacentre M is practically constant. Further the distance between the centre of gravity G and the metacentre M of a floating body (i.e., GM) as $\theta \to 0$, is known as *metacentric height.* As indicated in the next section the position of the metacentre relative to the position of the centre of gravity of a floating body determines the stability condition of the floating body.

4.3 STABILITY OF SUBMERGED AND FLOATING BODIES

By stability of a submerged or a floating body is meant the tendency for the body to return to the original upright position after it has been displaced slightly. A submerged or a floating body may be given a small linear displacement in horizontal or vertical direction or it may be given small angular displacement under the action of external forces. If a submerged or a floating body which is in equilibrium, is slightly displaced in the vertical or horizontal direction, then after the removal of the outside force causing such displacement, certain unbalanced force is developed on the body which tends to return the body to its original position. For example if a small upward displacement is given to a floating body, it results in the reduction of the buoyant force acting on the body thereby developing an unbalanced downward force which tends to return the body to its original position. However a slight horizontal displacement does not change either the magnitude or the location of the buoyant force, the body is still in equilibrium.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When a submerged or a floating body is given a slight angular displacement, it may have either of the following three conditions of equilibrium developed viz.,

(i) Stable equilibrium.

(ii) Unstable equilibrium.

(iii) Neutral equilibrium.

(i) Stable equilibrium. A body is said to be in a state of stable equilibrium if a small angular displacement of the body sets up a couple that tends to oppose the angular displacement of the body, thereby tending to bring the body back to its original position.

(ii) Unstable equilibrium. A body is said to be in a state of unstable equilibrium if a small angular displacement of the body sets up a couple that tends to further increase the angular displacement of the body, thereby not allowing the body to restore its original position.

(*iii*) Neutral equilibrium. A body is said to be in a state of neutral equilibrium if a small angular displacement of the body does not set up couple of any kind, and therefore the body adopts the new position given to it by the angular displacement, without either returning to its original position, or increasing the angular displacement.

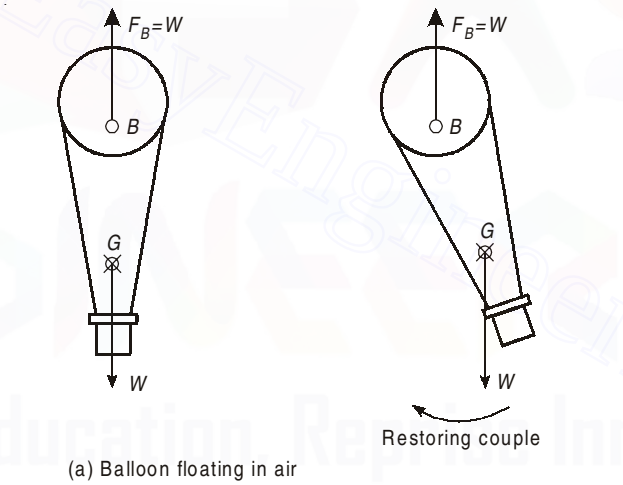


Figure 4.5 *Stability of a wholly submerged body*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For a body such as a balloon or a submarine, which is wholly submerged in a fluid, the positions of its centre of gravity and centre of buoyancy remain fixed. Therefore for a body wholly submerged in a fluid the conditions for stability of equilibrium are simple. In general a wholly submerged body is considered to be in a stable state of equilibrium if its centre of gravity is below the centre of buoyancy. This is so because as shown in Fig. 4.5, if such a submerged body is tilted slightly in a counter clockwise direction, the buoyant force and the weight produce a couple in the clockwise direction which restores the body to its original position. On the other hand a wholly submerged body will be in an unstable state of equilibrium if its centre of buoyancy is below its centre of gravity, because the couple produced by a slight angular displacement of the body tends to rotate the body away from its original position. However, if the centre of gravity and centre of buoyancy of a wholly submerged body coincide with each other, it is rendered in a neutral state of equilibrium.

A body floating in a liquid (or a partially immersed body) which is initially in equilibrium (i.e., $F_B = W$) when undergoes a small angular displacement, the shape of the immersed volume in general changes and hence centre of buoyancy moves relative to the body. It is on account of this shifting of the centre of buoyancy in the case of a partially immersed body that stable equilibrium can be achieved even when the centre of buoyancy is below the centre of gravity. Moreover, for a floating body the stability is not determined simply by the relative positions of the centre of gravity and the centre of buoyancy, but it depends on the position of the metacentre relative to the position of the centre of gravity as described below.

Consider a floating body which has undergone a small angular displacement in the clockwise direction as shown in Fig. 4.6. If the new centre of buoyancy B_1 is such that the metacentre M lies above the centre of gravity G of the body, as shown in Fig. 4.6 (a), then the buoyant force F_B and the weight W produce a couple of magnitude ($W\overline{GM}\sin\theta$) acting on the body in the anticlockwise direction, which is thus a restoring couple, tending to restore the body to its original position. The body is therefore in stable equilibrium. Hence it may be stated that for a floating body if the metacentre lies above its centre of gravity, i.e., $\overline{BM} > \overline{BG}$, then the body is in a stable state of equilibrium.

As shown in Fig. 4.6 (b), if for a floating body slightly tilted in clockwise direction, the metacentre M lies below the centre of gravity G of the body, then the buoyant force and the weight produce a couple acting on the body in the clockwise direction, which is thus an overturning couple, tending to increase the angular displacement of the body still further. The body is then considered to be in unstable equilibrium. Thus it may be stated that for a floating body if the metacentre lies below its centre of gravity, i.e., $\overline{BM} < \overline{BG}$, then the body is said to be in an unstable state of equilibrium.

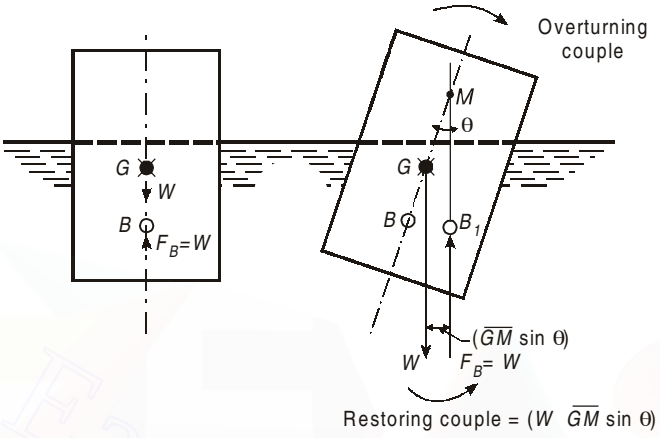
However, if for a floating body the metacentre coincides with the centre of gravity of the body i.e., $BM = BG$, then the body will be in a neutral state of equilibrium. This is so because there will be neither a restoring couple nor an overturning couple developed when the body is tilted slightly. As such the body will neither return to its original position nor increase its angular displacement, but it will simply adopt its new position.

Stability of floating objects such as boats, ships etc., is required to be studied on account of the fact that these objects are always acted upon by certain external forces which may temporarily cause angular displacement or heeling of these objects. Amongst these forces which are common are : wind and wave action, pressure due to tidal or river currents, pressure due to manoeuvring a boat or ship in a curved path by rudder or propeller action, anchor line pulls etc. The heeling may also be caused by any probable shifting of the cargo or of the passengers, or by any probable variation in the weight and

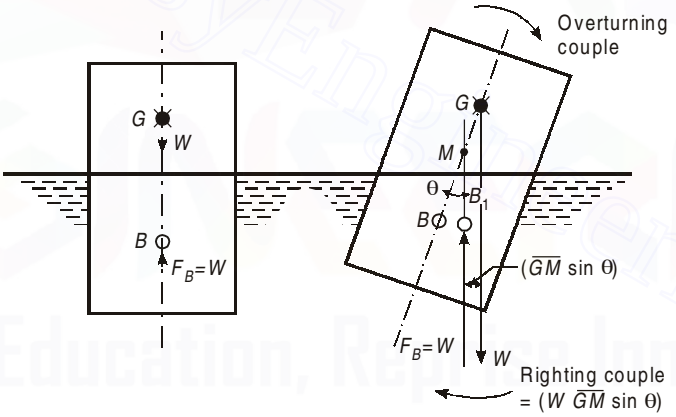
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

distribution of the cargo (for example due to consumption of fuel and other supplies) etc. Therefore in the design of the floating objects such as boats, ships etc., care has to be taken to keep the metacentre well above the centre of gravity of the object so that even under the worst conditions the floating object



(a) Floating body in stable equilibrium



(b) Floating body in unstable equilibrium

Figure 4.6 *Stability of a partially immersed (or floating) body*

shall be in a stable state of equilibrium. This may however be achieved by lowering the position of the centre of gravity of the body by loading it permanently by adding some heavier material known as ballast. Furthermore the location of the metacentre for the objects may be known by determining the metacentric height as discussed in Section 4.4.

4.4 DETERMINATION OF METACENTRIC HEIGHT

There are two methods which may be used to determine the metacentric height of a floating body. These are

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) Experimental Method.

(ii) Theoretical Method.

(i) **Experimental Method for the Determination of the Metacentric Height.** As shown in Fig. 4.7, consider a ship (or a boat) floating in water. Let w be a movable weight placed centrally on the deck of the ship and W be the total weight of the ship including w. It is assumed that the ship is initially in equilibrium, so that the deck is horizontal. Now the weight w is moved transversely through a distance x across the deck, so that the ship tilts through a small angle θ, and comes to rest in a new position of equilibrium. The angle θ may be measured by noting the horizontal distance moved by a long plumb line or a pendulum, hanging inside the ship from the centre point of the deck, on a horizontal scale, as shown in Fig. 4.7.

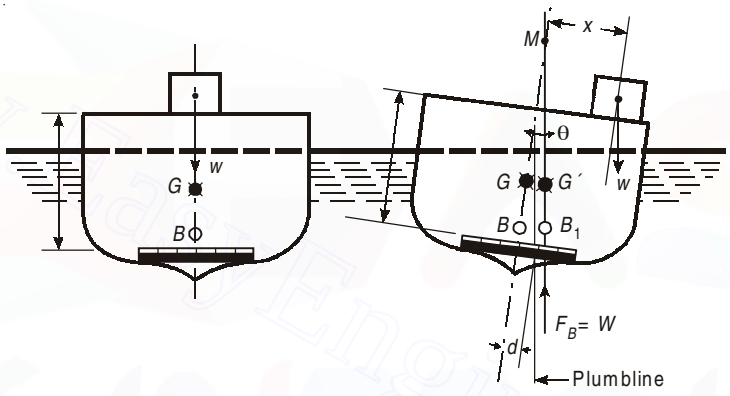


Figure 4.7 *Experimental determination of the metacentric height*

In the new (i.e., tilted) position of equilibrium of the ship, the centres of gravity and buoyancy will again be vertically in line. The movement of the weight w through a distance x causes a parallel shift of the total centre of gravity (that is the centre of gravity of the whole ship including w) from G to G' such that

$$wx = W\left(\overline{GG'}\right)$$

But $\overline{GG'} = (\overline{GM})\tan\theta$ and so

$$W(\overline{GM})\tan\theta = wx$$

$$\therefore \quad (\overline{GM}) = \frac{wx}{W\tan\theta} \qquad ...(4.3)$$

Now if l is the length of the plumb line or pendulum and d is the distance moved by it on the horizontal scale then

$$\tan\theta = \frac{d}{l}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and therefore $\overline{GM} = \frac{wx}{W} \times \frac{l}{d}$ ...(4.3 a)

Since the point *M* corresponds to the metacentre for small angles of heel only, the true metacentric height is the limiting value of $\overline{GM}$ as $\theta \to 0$. This may be determined from a graph plotted between the nominal values of $\overline{GM}$ calculated from Eq. 4.3 for various values of θ (both positive and negative) and the angle θ.

The metacentric height for a ship or a boat may be determined by the method discussed above only after it has been constructed. But in many cases it is, however, desirable to be able to determine the metacentric height before a ship or a boat has been constructed. In such cases the theoretical method for the determination of the metacentric height, as discussed below, may be adopted.

(ii) Theoretical Method for the Determination of the Metacentric Height. By this method the metacentric height for a ship or a boat may be determined simply by considering its shape. Fig. 4.8 shows the different views of a ship. At (a) is shown the initial equilibrium position of the ship and at (b) is shown the position of the ship after it has been tilted about the longitudinal axis through a small angle θ. In the tilted position of the ship the portion on the left *AOA'* has emerged from the liquid,

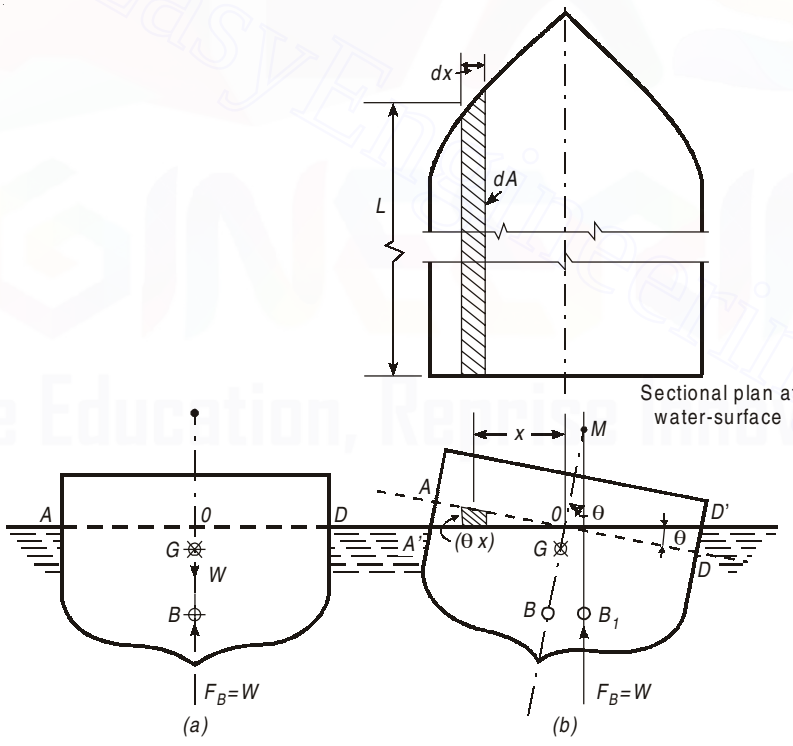


Figure 4.8 *Determination of metacentric height by the theoretical method*

whereas the portion *DOD'* on the right has moved down into the liquid. It is assumed that there is no overall vertical movement, thus the vertical equilibrium is not disturbed. Further as the total weight of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the body is not altered, the volume immersed remains unaltered, and therefore the volumes, corresponding to *AOA'* and *DOD'* wedges on either sides are equal. This is so if the planes of floatation for the equilibrium and the tilted positions intersect along the centroidal axes of the planes.

As the ship is tilted its centre of buoyancy shifts from *B* to B_1 which causes a change in the moment of the buoyant force amounting to $\left(F_B \times \overline{BM} \times \theta\right)$. This shift of the centre of buoyancy is entirely due to change in the geometrical form of the displaced volume by the submergence of the right wedge *DOD'* into the liquid and the emergence of the left wedge *AOA'* from the liquid. These two wedges may represent respectively a positive and negative buoyant force which form a couple. The effect of this couple must be the same as the change in moment of the buoyant force that is $\left(F_B \times \overline{BM} \times \theta\right)$ produced by the shift of the total buoyant force F_B.

To evaluate the couple due to the wedges, consider two small prisms of the wedges at a distance *x* from *O* on either side. The volume of each prism is ($L\,\theta x dx$), where *L* is the length of the ship. If *w* is the specific weight of the liquid, then the weight of the volume of the liquid equal to that of the prism is ($wL\,\theta x dx$). The moment of the couple due to this pair of prisms is ($2x \times wL\theta x dx$). The summation of the moments of such prisms considered in the wedges must be equal to $\left(F_B \times \overline{BM} \times \theta\right)$. Thus

$$F_B \times \overline{BM} \times \theta = 2w\,\theta \int x^2 L dx = 2w\,\theta \int x^2 dA$$

in which dA ($= Ldx$) is an elementary area of the cross-sectional area of the ship at the liquid surface, as shown in Fig. 4.8. The quantity ($2\int x^2 dA$) is the moment of inertia *I* of the cross-sectional area of the ship at the liquid surface about its longitudinal axis. Therefore

$$\overline{BM} = \frac{wI}{F_B} = \frac{wI}{wV} = \frac{I}{V} \qquad ...(4.4)$$

where *V* is the volume of the liquid displaced by the ship. Equation 4.4 shows that the location of metacentre for any ship tilting through a small angle of heel θ depends on the geometry and the weight. Further the length $\overline{BM}$ is sometimes known as the metacentric radius. The metacentric height *GM* may then be determined as

$$\overline{GM} = \left(\overline{BM} - \overline{BG}\right) = \left(\frac{I}{V} - \overline{BG}\right)$$

if the metacentre *M* lies above the centre of gravity *G*, and

$$\overline{GM} = \left(\overline{BG} - \overline{BM}\right) = \left(\overline{BG} - \frac{I}{V}\right)$$

if the metacentre *M* lies below the centre of gravity *G*. Both the above expressions for $\overline{GM}$ may be combined as follows:

$$\overline{GM} = \pm \left(\overline{BM} - \overline{BG}\right) = \pm \left(\frac{I}{V} - \overline{BG}\right). \qquad ..(4.5)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

4.5 METACENTRIC HEIGHT FOR FLOATING BODIES CONTAINING LIQUID

If a floating body contains liquid with a free surface then as shown below, the metacentric height of the body is reduced and consequently the stability of the body is reduced.

Consider a rectangular body of width b and length l floating in a liquid of specific weight w and containing liquid of specific weight w_1 as shown in Fig . 4.9 (a). When the body floats with its axis vertical, let CD be the free surface of the liquid contained by it, G be the centre of gravity of the floating body including the inside liquid and B be the centre of buoyancy. Now if the body is tilted through an angle θ, the centre of buoyancy would shift from B to B_1 and since the surface of the liquid in the body will remain horizontal, the new surface of this liquid is represented by $H K$ as shown in Fig. 4.9 (*b*).

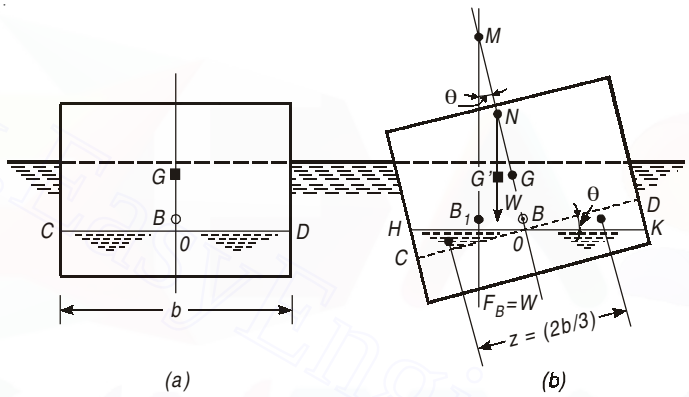


Figure 4.9 *Metacentric height for a floating*

Thus the tilting of the body has the effect of moving the wedge of liquid DOK to COH, which would result in shifting of the centre of gravity of the inside liquid and this in turn would cause the centre of gravity G to shift to new position G'. The moment of the restoring couple acting on the body is then equal to $W\ \overline{MN}\ \sin\theta$ (or $W\ \overline{MN}\ \theta$ for θ being small), where W is the weight of the floating body including the liquid contained by it, M is the metacentre and N is the point of intersection between the vertical through G' and the vertical axis of the body. As such for this case the metacentric height is $\overline{MN}$.

The movement of the wedge of the liquid DOK to COH develops a turning couple acting on the body in the same direction in which the body tilts, due to which the restoring couple acting on the body is reduced. The moment of this turning couple due to the movement of the liquid is $(w_1 V_1 z)$ where V_1 is the volume of the either wedge and z is the distance between the centre of gravity of these wedges. If θ is small,

$$CH = DK = (b/2)\,\theta \text{ and hence}$$

$$V_1 = (b^2/8)l\,\theta \text{ and } z = (2b/3)$$

$$\therefore \quad (w_1 V_1 z) = w_1 (lb^3/12)\theta = (w_1 I_1 \theta)$$

where $I_1 = (lb^3/12)$ is the moment of inertia of the area of the free surface of the liquid contained by the floating body about its longitudinal axis.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus the restoring couple acting on the body is

$$\left(W\overline{MN}\ \theta\right) = \left(W\overline{GM}\ \theta\right) - \left(w_1 I_1 \theta\right)$$

from which

$$\overline{MN} = \overline{GM} - \left(\frac{w_1 I_1}{W}\right)$$

Further since $W = wV$, where V is the volume of the liquid displaced by the floating body,

$$\overline{MN} = \overline{GM} - \left(\frac{w_1 I_1}{wV}\right)$$

In the above expression $\overline{GM}$ is given by Eq. 4.5 which is applicable for this case also. It is thus seen that when a floating body contains liquid the metacentric height is reduced by an amount $(w_1 I_1 / wV)$.

If the liquid contained by a floating body is put into a number of separate compartments, the movement of the liquid within the body is minimized and consequently there would be less reduction in the metacentric height of the floating body. For this reason when a liquid is to be carried by a ship it is put into a number of separate compartments. It may be shown by the same analysis that when the liquid is put into a number of separate compartments the metacentric height would be reduced by an amount $[\Sigma(w_1 I'_1)/wV]$ where I'_1 is the moment of inertia of the area of the free surface of the liquid in each compartment.

4.6 TIME PERIOD OF TRANSVERSE OSCILLATION OF A FLOATING BODY

A floating body may be set in a state of oscillation as if suspended at the metacentre M in the same manner as a simple pendulum. This may happen when an overturning couple by which a floating body is tilted through an angle θ, is suddenly removed. The body is then acted upon by a torque equal to the moment of the restoring couple only, which is equal to $(W\ \overline{GM} \sin\theta)$ and sets it in a state of oscillation. But the torque exerted on an oscillating body is equal to the rate of change of angular momentum, which is equal to the moment of inertia (i.e., second moment of mass) multiplied by angular acceleration. Thus we may write

$$W\ \overline{GM}\ \sin\theta = -\left(Mk_G^2\right)\ \frac{d^2\theta}{dt^2} \qquad \ldots(4.6)$$

where M is the mass of floating body ; k_G is radius of gyration of the body about its centre of gravity so that (Mk_G^2) represents the moment of inertia of the body about its axis of rotation ; and $\left(\frac{d^2\theta}{dt^2}\right)$ is angular acceleration. Here the minus sign has been introduced because the torque acts so as to decrease θ, that is the angular acceleration $\left(\frac{d^2\theta}{dt^2}\right)$ is negative. Thus for smaller angular movements (sin θ) is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

proportional to $\left(-\frac{d^2\theta}{dt^2}\right)$ as for a simple pendulum. If θ is small so that $\sin\theta \simeq \theta$ (radians), Eq. 4.6 may be expressed as

$$W\overline{GM}\ \theta = -(Mk_G^2)\ \frac{d^2\theta}{dt^2}$$

or
$$\frac{d^2\theta}{dt^2} = -\left(\frac{W\overline{GM}}{Mk_G^2}\right)\theta \qquad ...(4.6\ a)$$

The solution of the differential Eq. 4.6 (a) is as noted below

$$\theta = A\sin\left(t\sqrt{\frac{W\overline{GM}}{Mk_G^2}}\right) + B\cos\left(t\sqrt{\frac{W\overline{GM}}{Mk_G^2}}\right)$$ where A and B are constants.

Let T be the time period of one complete oscillation of the floating body from one side to the other and back again. Then if at $t = 0$, $\theta = 0$; again at $t = (T/2)$, $\theta = 0$. Introducing these values in the above expression, we get

$$B = 0 \text{ and } A\sin\left(\frac{T}{2}\sqrt{\frac{W\overline{GM}}{Mk_G^2}}\right) = 0$$

Since $A \neq 0$; $\therefore\ \sin\left(\frac{T}{2}\sqrt{\frac{W\ \overline{GM}}{Mk_G^2}}\right) = 0$

from which

$$T = 2\pi\left\{\frac{Mk_G^2}{W\overline{GM}}\right\}^{1/2} = 2\pi\left\{\frac{k_G^2}{g\overline{GM}}\right\}^{1/2} \qquad ... (4.7)$$

where g is acceleration due to gravity.

It may however be indicated that the oscillating motion of a floating body ultimately dampens by the frictional forces between the floating body and the liquid.

A ship or a boat may have two types of oscillatory motions viz., *rolling* and *pitching*. The oscillatory motion of a ship or a boat about its longitudinal axis is designated as rolling. On the other hand the pitching movement or simply pitching may be defined as the oscillatory motion of a ship or a boat about its transverse axis. Although the above noted expressions for the metacentric height (Eq. 4.5) and the time period of oscillation (Eq. 4.7) are derived for the rolling motion, the same may be adopted for the pitching motion also. It may, however, be noted that since the moment of inertia of the cross-sectional area of the ship or boat at the liquid surface about its transverse axis is much more than the same about its longitudinal axis, the metacentric height for the pitching motion is invariably greater than that for the rolling motion. As such if a ship or a boat has a safe metacentric height for rolling motion then it will be safe in pitching motion also. As regards the time period of oscillation it has been found that for rolling motion of ships or boats the theoretical values of T given by Eq. 4.7 agree reasonably with the experimental values, but the agreement is less good for pitching movements.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In the case of ships the shifting of cargo may cause the ships to roll. As such along with the considerations of the stability of a ship, its period of roll is also required to be determined. This is so because, as indicated earlier, increasing the metacentric height gives greater stability to a floating body, but as indicated by Eq. 4.7 an increase in the metacentric height reduces the time period of rolling of the body. A smaller value of time period of rolling of a passenger ship is quite uncomfortable for the passengers. Further a ship with a smaller time period of rolling is subjected to undue strains which may damage its structure. In the case of cargo ships the metacentric height varies with the loading and hence some control on the value of the metacentric height as well as the time period of rolling is possible by adjusting the position of the cargo. However, in the case of warships and racing yachts, the stability is more important than comfort, and hence such vessels have larger metacentric height.

The metacentric height of ocean-going vessels is usually of the order of 0.3 m to 1.2 m. On the other hand for warships it may range from 1 m to 1.5 m and for some of the river crafts it may be as large as about 3.6 m.

ILLUSTRATIVE EXAMPLES

Examples 4.1. *A wooden block of rectangular section 1.25 m wide, 2 m deep, 4 m long floats horizontally in sea water. If the specific gravity of wood is 0.64 and water weighs 1 025 kg(f)/m^3, find the volume of liquid displaced and the position of the centre of buoyancy.*

Solution

According to Archimedes' principle weight of the liquid displaced by the body = weight of the body

Weight of the block= $(1.25 \times 2 \times 4) \times 0.64 \times 1000$

$= 6\,400$ kg(f)

Thus

Volume of sea water displaced by the block = $\dfrac{6400}{1025} = 6.24$ m^3

Let h be the depth of the block under water, then

$$(4 \times 1.25 \times h) = 6.24$$

$$\therefore \quad h = \frac{6.24}{4 \times 1.25} = 1.248 \text{ m}$$

Centre of buoyancy B is at a height of $\dfrac{1.248}{2} = 0.624$ m above the base.

Example 4.2. *A wooden cylinder of circular section and uniform density, specific gravity 0.6 is required to float in oil of specific gravity 0.8. If the diameter of the cylinder is d and its length is l, show that l cannot exceed about 0.817d for cylinder to float with its longitudinal axis vertical.*

Solution

As shown in the Fig. Ex. 4.2 if the depth of immersion is x then

$$\left(\frac{\pi d^2 x}{4}\right) \times 0.8 = \left(\frac{\pi d^2}{4} \times l\right) \times 0.6$$

or $$x = \left(\frac{0.6}{0.8}\right) l = \frac{3}{4} l$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

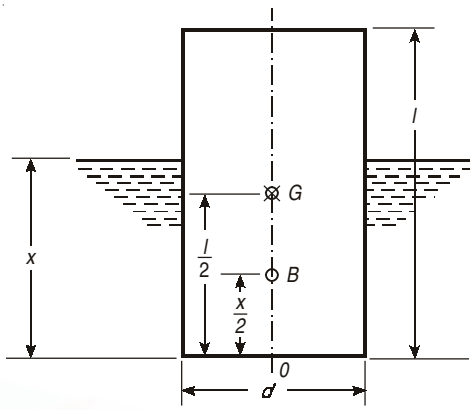


Figure. Ex. 4.2

So $\overline{OB} = \frac{x}{2} = \frac{3}{8} l$

Since $\overline{BM} = \frac{I}{V}$

$$I = \frac{\pi d^4}{64}$$

$$V = \left(\frac{\pi d^2}{4} \times \frac{3}{4} l \right) = \frac{3\pi d^2}{16} l$$

$\therefore$ $\overline{BM} = \left(\frac{\pi d^4}{64} \times \frac{16}{3\pi d^2 l} \right) = \frac{d^2}{12l}$

Further $\overline{BG}$

$$= (\overline{OG} - \overline{OB})$$

$$= \left(\frac{l}{2} - \frac{3}{8} l \right) = \frac{l}{8}$$

For stable equilibrium

$$\overline{BM} > \overline{BG}$$

or $$\frac{d^2}{12l} > \frac{l}{8}$$

or $$l^2 < \frac{2}{3} d^2$$

or $$l < \left(\sqrt{2/3} \right) d$$

or $$l < 0.817\, d$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 4.3. *Show that a cylindrical buoy 1.25 m diameter and 3.25 m high weighing 11 127 N will not float vertically in sea water weighing 10 055 N/m³. Find the tension necessary in a vertical chain attached to the centre of the base of the buoy that will just keep the cylinder vertical.*

Solution

Weight of the buoy = 11 127 N

∴ Volume of sea water displaced is

$$V = \frac{11127}{10055} = 1.107 \text{ m}^3$$

Depth of immersion

$$= \frac{1.107}{\frac{\pi}{4}\times(1.25)^2} = 0.90 \text{ m}$$

∴ Height of the centre of buoyancy B above the base

$$= \frac{0.90}{2} = 0.45 \text{ m}$$

Since $\overline{BM} = \frac{I}{V}$

$$I = \frac{\pi d^4}{64} = \frac{\pi\times(1.25)^4}{64} = 0.119\,8 \text{ m}^4$$

$$V = 1.107 \text{ m}^3$$

∴ $$\overline{BM} = \frac{0.1198}{1.107} = 0.1082 \text{ m}$$

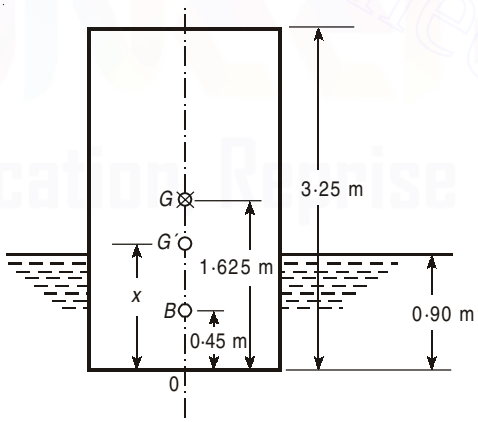


Figure. Ex. 4.3

Further $\overline{BG} = \left(\overline{OG} - \overline{OB}\right)$

$$= \left[\left(\frac{1}{2}\times 3.25\right) - 0.45\right]$$

$$= 1.175 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $\overline{BM} < \overline{BG}$, metacentre M lies below the centre of gravity G, the cylinder will be in an unstable state of equilibrium.

Let T be the tension required to be applied vertically in the chain attached to the centre of the base. Hence total effective weight of the body becomes

$$(11\,127 + T)\ N$$

$\therefore$ Volume of sea water displaced is

$$V = \left(\frac{11\,127 + T}{10\,055}\right) \text{m}^3$$

Depth of immersion

$$\left(\frac{11127 + T}{10055}\right) = \times \frac{1}{\frac{\pi}{4} \times (1.25)^2}$$

$$= \frac{(11127 + T) \times 64}{(10055 \times 78.54)} \text{m}$$

If B' is new centre of buoyancy, then the height of B' above the base of the buoy is

$$OB' = \frac{1}{2}\left[\frac{(11\,127 + T) \times 64}{(10\,055 \times 78.54)}\right] \text{m}$$

Also $$\overline{B'M'} = \frac{I}{V}$$

$$I = \frac{\pi d^4}{64} = \frac{\pi \times (1.25)^4}{64}$$

$$V = \left(\frac{11\,127 + T}{10\,055}\right)$$

$\therefore$ $$\overline{B'M'} = \frac{\pi \times (1.25)^4 \times 10055}{64 \times (11127 + T)} \text{ m}$$

The effect of applying a tension T is to lower the centre of gravity of the buoy. Thus if G' is new centre of gravity then since,

Weight acting at G = 11 127 N and weight acting at G' = (11 127 + T) N taking moments about O, we get

$$(11\,127 + T)\, x = 11\,127 \times \frac{3.25}{2}$$

$\therefore$ $$x = \frac{(11127 \times 3.25)}{2(11127 + T)} \text{m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For stable equilibrium the metacentre M' must be above G', i.e.,

$$\overline{B'M'} > \overline{B'G'}$$

Since $$\overline{B'G'} = \left(\overline{OG} - \overline{OB'}\right)$$

$$= \left[\frac{(11127 \times 3.25)}{2(11127 + T)} - \frac{(11127 + T) \times 64}{2(10055 \times 78.54)}\right]$$

$$\therefore \quad \frac{\pi \times (1.25)^4 \times 10055}{64 \times (11127 + T)} > \left[\frac{(11127 \times 3.25)}{2(11127 + T)} \frac{(11127 + T) \times 64}{2(10\,055 \times 78.54)}\right]$$

$$\text{or} \quad \frac{(11127 + T) \times 64}{2(10055 \times 78.54)} > \frac{(11127 \times 3.25 \times 32) - (\pi \times 1.25^4 \times 10055)}{(11127 + T) \times 64}$$

$$\text{or} \quad (11127 + T)^2 > 4.165 \times 10^8$$

$$\text{or} \quad (11127 + T) > 20\,408$$

$$\text{or} \quad T > 9281$$

$\therefore$ Minimum tension in the chain is

$$T = 9281 \text{ N}$$

Example 4.4. *A cylindrical buoy, diameter 1.5 m and 1.1 m high weighing 4.450 kN is floating in sea water with its axis vertical. Find the maximum permissible height above the top of the buoy, of the centre of gravity of a 450 N load which is placed centrally on top of the buoy. Take specific gravity of sea water as 1.025.*

Solution

The weight of sea water displaced

$$= (4\,450 + 450) = 4\,900 \text{ N}$$

$\therefore$ Volume of sea water displaced is

$$V = \frac{4900}{1.025 \times 9810} = 0.487 \text{ m}^3$$

Depth of immersion $$= \frac{0.487}{\frac{\pi}{4} \times (1.5)^2} = 0.276 \text{ m}$$

$\therefore$ The height of the centre of buoyancy above the base

$$= \frac{0.276}{2} = 0.138 \text{ m}$$

The combined centre of gravity of the buoy and the load may be determined by taking the moments above the base of the buoy. Thus if the distance of the centre of gravity of the load from the base is x and that of the combined centre of gravity G' is $\bar{x}$, then by taking the moments about the mid-point of base O, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$4\,900\ \bar{x} = 4\,450 \times \frac{1.1}{2} + 450 \times x$$

$$\therefore \quad \bar{x} = \left(\frac{2447 \times 5 + 450x}{4900}\right)$$

$$= (0.499 + 0.092\ x\ )$$

Thus $\quad \overline{BG'} = \overline{OG'} - \overline{OB'}$

$$= \bar{x} - 0.138$$

$$= (0.499 + 0.092x\ ) - 0.138$$

$$= (0.361 + 0.092x\ )$$

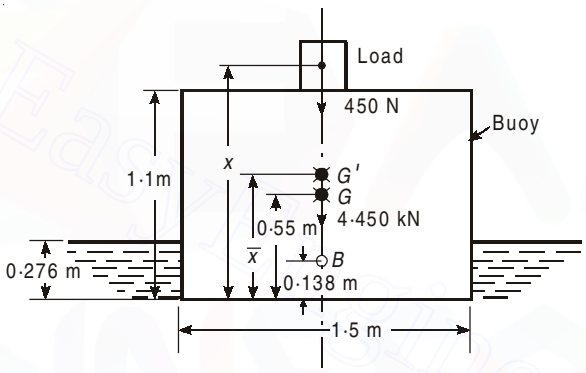


Figure Ex. 4.4

Since $\quad \overline{BM} = \dfrac{I}{V}$

$$I = \frac{\pi}{64}(1.5)^4\ \text{m}^4\ ;\ V = 0.487\ \text{m}^3$$

$$\therefore \quad \overline{BM} = \frac{\pi}{64} \times \frac{(1.5)^4}{0.487} = 0.510\ \text{m}$$

For stable equilibrium of the floating buoy

$$\overline{BM} > \overline{BG'}$$

or $\quad 0.510 > (0.361 + 0.092\ x\ )$

or $\quad 0.092x < 0.149$

$$\therefore \quad x < 1.62\ \text{m}$$

i.e., the centre of gravity of the load must not be more than (1.62 – 1.1) = 0.51 m above the top of the buoy.

Example 4.5. *A wooden cylinder of diameter d and length 2d floats in water with its axis vertical. Is the equilibrium stable? Locate the metacentre with reference to water surface. Specific gravity of wood is 0.6.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Weight of the cylinder = $\left(\frac{\pi}{4}d^2 \times 2d\right) \times 0.6 \times 9\,810\ \text{N} = 9\,245.7\ d^3\ \text{N}$

$\therefore$ Volume of water displaced by the cylinder

$$= \frac{9245.7d^3}{9810} = 0.942\ d^3$$

Depth of immersion $= \frac{0.942d^3}{\frac{\pi}{4}d^2} = 1.2\ d$

$\therefore$ Height of the centre of buoyancy above the base

$$= \frac{1}{2}(1.2d) = 0.6\ d$$

$\therefore$ $\overline{BG} = (d - 0.6d) = 0.4d$

Since $\overline{BM} = \frac{I}{V}$

$$I = \frac{\pi d^4}{64};\ V = 0.942\ d^3$$

$\therefore$ $\overline{BM} = \frac{\pi d^4}{64} \times \frac{1}{0.942d^3} = 0.052\ d$

In this case since, $\overline{BM} > \overline{BG}$ the metacentre lies below the centre of gravity. As such the cylinder is in unstable equilibrium.

The metacentre is at a depth of $(0.6d - 0.052d) = 0.548d$ below the water surface.

Example 4.6. *A hollow cylinder of outside diameter 1.25 m, length 3.5 m and specific weight 75 537 N/m^3, floats just in stable equilibrium in sea water. Find the minimum permissible thickness of the cylinder. Sea water weighs 10 055 N/m^3.*

Solution

Let t be the thickness of the cylinder and x be the depth of the portion immersed in sea water. Then by the principle of floatation, weight of the liquid displaced by the body is equal to the weight of the body.

Weight of sea water displaced

$$= \left[\frac{\pi}{4} \times (1.25)^2 \times x \times 10055\right] \text{N}$$

$$= 12\,339.34\ x\ \text{N}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Weight of the body

$$= [\pi \times (1.25) \times t \times 3.5 + 2 \times \frac{\pi}{4} \times (1.25)^2 \times t\,] 75\,537$$

$$= 1\,223\,611.40\,t \text{ N}$$

In calculating the weight of the body it has been assumed that the thickness t is very small and hence the weight of the two end sections and the circular portion have been separately taken. Thus

$$12\,339.34\,x = 1\,223\,611.40\,t$$

$$\therefore \quad x = 99.16\,t$$

As $$\overline{BM} = \frac{I}{V}$$

$$I = \frac{\pi}{64}(1.25)^4 = 0.12 \text{ m}^4$$

and $$V = \frac{1223611.40\,t}{10\,055} = 121.69\,t \text{ m}^3$$

$$\therefore \quad \overline{BM} = \frac{0.12}{121.69t} = \frac{0.986 \times 10^{-3}}{t}$$

$$\overline{BG} = \left[\frac{3.5}{2} - \frac{99.16t}{2}\right]$$

$$= (1.75 - 49.58\,t)$$

For the cylinder to be just in stable equilibrium

$$\overline{BM} = \overline{BG}$$

or $$\frac{0.985 \times 10^{-3}}{t} = (1.75 - 49.58\,t)$$

or $$49.58\,t^2 - 1.75\,t + 0.986 \times 10^{-3} = 0$$

$$t = 0.000\,57 \text{ m or } 0.035 \text{ m}$$

Thus minimum permissible thickness of the cylinder = 0.000 57 m = 0.57 mm.

Example 4.7. *A cone floating in water with its apex downwards has a diameter d and vertical height h. If the specific gravity of the cone is S, prove that for stable equilibrium*

$$h^2 < \frac{1}{4}\left[\frac{d^2 S^{1/3}}{1 - S^{1/3}}\right]$$

Solution

Let the height of the portion of the cone under water be x. Then, we have

$$\frac{1}{3} \times \frac{\pi d^2}{4} \times h \times wS = \frac{1}{3}\frac{\pi d_1^2}{4} \times x \times w$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where d_1 is the diameter of the cone at the water surface level.

or $$(d^2 h) \times S = (d_1^2 \times x) \quad \text{....(i)}$$

Also $$\left(\frac{d_1}{d}\right) = \frac{x}{h}$$

or $$d_1 = \frac{xd}{h} \quad \text{....(ii)}$$

Substituting the value of d_1 from Eq. (ii) in Eq. (i), we get

$$(d^2 h) \times S = x \times \left(\frac{xd}{h}\right)^2$$

or $$x = h\,(S)^{1/3}$$

The centre of gravity of the cone is at a depth of $\frac{1}{4}$ th of the height of the cone below the base

$$\therefore \quad \overline{BG} = \left(\frac{3h}{4} - \frac{3x}{4}\right) = \frac{3}{4}(h - hS^{1/3})$$

As $$\overline{BM} = \frac{I}{V}$$

$$I = \frac{\pi d_1^4}{64}; \text{ and } V = \frac{1}{3}\frac{\pi d_1^2}{4} \times x$$

$$\therefore \quad \overline{BM} = \frac{\frac{\pi d_1^4}{64}}{\frac{1}{3}\frac{\pi d_1^2}{4} \times x} = \frac{3d_1^2}{16x}$$

By substituting for d_1 and x, we get

$$\overline{BM} = \frac{3S^{1/3} d^2}{16h}$$

For stable equilibrium

$$\overline{BM} > \overline{BG}$$

or $$\frac{3S^{1/3} d^2}{16h} > \frac{3}{4}(h - hS^{1/3})$$

or $$h^2 < \frac{1}{4}\left[\frac{d^2 S^{1/3}}{1 - S^{1/3}}\right]$$

Example 4.8. *An object which has a volume of 0.18 m^3 requires a force of 265 N [27 kg(f)] to keep it immersed in water. If a force of 157 N [16 kg(f)] is required to keep it immersed in another liquid, find the specific gravity of the liquid.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

SI Units

Let S_1 and S_2 be the specific gravities of the object and the liquid respectively.

When the object is immersed in water, the total force acting in the downward direction

$= (265 + 0.18 \times 9\,810 \times S_1)$ N

Buoyant force $= (0.18 \times 9\,810)$ N

Thus equating the two, we get

$(265 + 0.18 \times 9\,810 \times S_1) = (0.18 \times 9\,810)$

$S_1 = 0.85$

When the object is immersed in another liquid, the total force acting in the downward direction

$= (157 + 0.18 \times 9\,810 \times 0.85)$ N

Buoyant force $= (0.18 \times 9\,810 \times S_2)$ N

Thus equating the two, we get

$(157 + 0.18 \times 9\,810 \times 0.85) = (0.18 \times 9\,810 \times S_2)$

$S_2 = 0.94$

Metric Units

Let S_1 and S_2 be the specific gravities of the object and the liquid respectively.

When the object is immersed in water, the total force acting in the downward direction

$= (27 + 0.18 \times 1000 \times S_1)$ kg (f)

Buoyant force $= (0.18 \times 1000)$ kg (f)

Thus equating the two, we get

$(27 + 0.18 \times 1000\, S_1) = (0.18 \times 1000)$

$S_1 = 0.85$

When the object is immersed in another liquid, the total force acting in the downward direction

$= (16 + 0.18 \times 1000 \times 0.85)$ kg (f)

Buoyant force $= (0.18 \times 1000 \times S_2)$ kg(f)

Thus equating the two, we get

$(16 + 0.18 \times 1000 \times 0.85) = (0.18 \times 1000 \times S_2)$

$S_2 = 0.94$

Example 4.9. *A closed pontoon 4 m square by 1m deep is divided into 4 equal compartments by vertical diaphragms. It is constructed throughout from 6 mm thick metal weighing 78 480 N/m^3. Calculate the metacentric height when floating in sea water.*

(*i*) *With no water inside.*

(*ii*) *When sea water is filled in all the compartments to a depth of 0.3 m. Specific weight of sea water is 10 006 N/m^3.*

Solution

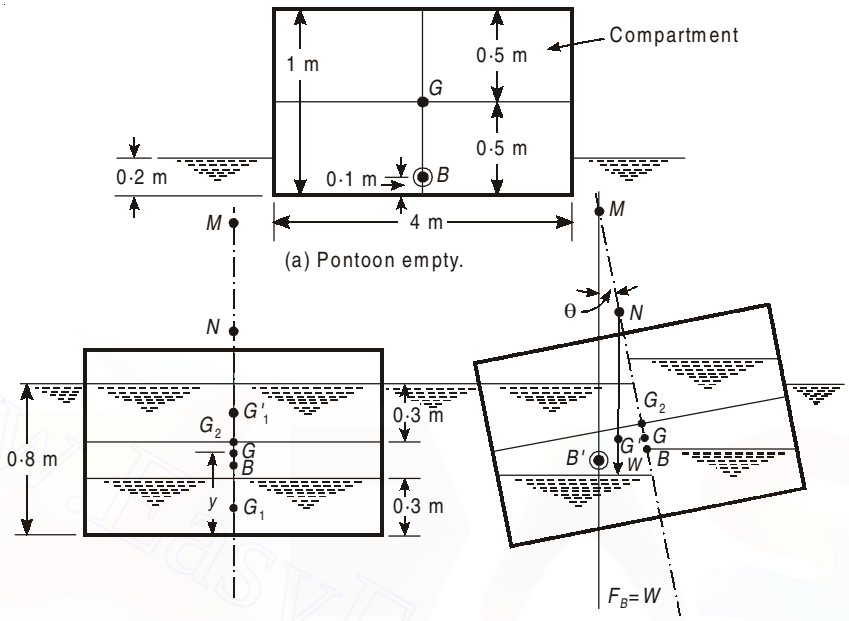
(i) The weight of the pontoon along with the diaphragm is

$$W = \left[5\left\{4\times1\times\frac{6}{1000}\right\}+3\left\{4\times4\times\frac{6}{1000}\right\}\right]78\ 480$$

$= 32\ 019.84$ N

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net



(b) Pontoon with compartments filled with sea water

Figure Ex. 4.9

∴ Volume of sea water displaced is

$$V = \frac{32019.84}{10006} = 3.2 \text{ m}^3$$

Depth of immersion is

$$x = \frac{3.2}{4\times4} = 0.2 \text{ m}$$

So the height of the centre of buoyancy above the bottom of the pontoon

$$= \frac{1}{2} \times (0.2) = 0.1 \text{ m}$$

$$\therefore \quad \overline{BG} = (0.5 - 0.1) = 0.4 \text{ m}$$

As $$\overline{BM} = \frac{I}{V}$$

$$I = \left[\frac{1}{12}\times4\times(4)^3\right] = 21.33 \text{ m}^4,$$

and $$V = 3.2 \text{ m}^3$$

$$\therefore \quad \overline{BM} = \frac{21.33}{3.2} = 6.67 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore \quad \overline{GM} = \overline{BM} - \overline{BG} = (6.67 - 0.4) = 6.27$ m

(ii) When sea water is filled in all the inner compartments the total weight of the pontoon is

$$W = 32\,019.84 + 4\,[4 \times 2 \times 0.3] \times 10\,006$$
$$= 128\,077.44 \text{ N}$$

$\therefore$ Volume of sea water displaced is

$$V = \frac{128077.44}{10006} = 12.8 \text{ m}$$

$\therefore$ Depth of immersion $= \frac{12.8}{4 \times 4} = 0.8$ m

$\therefore$ Height of the centre of buoyancy above the bottom of the pontoon

$$= \frac{1}{2} \times (0.8) = 0.4 \text{ m}$$

As shown in Fig. Ex. 4.9 (b) let G_1 be the centre of gravity of lower tanks and G'_1 be the centre of gravity of the top tanks, and G_2 be the centre of gravity of the unloaded pontoon. Now if G is centre of gravity of the loaded pontoon and its height above the bottom of the pontoon is y, then the value of y can be determined by taking moments about the base:

$$[\,32\,019.84 \times 0.5 + 2(4 \times 2 \times 0.3) \times 10\,006 \times \frac{0.3}{2} + 2\,(4 \times 2 \times 0.3) \times 10\,006 \times (0.5 + \frac{0.3}{2})\,]$$
$$= 128\,077.44\, y$$

$\therefore \quad y = 0.425$ m

As shown in Fig. Ex. 4.9 (b) when the loaded pontoon heels through a small angle θ, the centre of buoyancy B moves to B', i.e., the centre of gravity of the displaced volume V moves such that

$$\overline{BM'} = \overline{BM} \times \theta = \frac{I}{V} \times \theta$$

$\therefore$ The centre of gravities of each of the compartments will also be displaced by

$$\frac{I_1}{V_1} \times \theta$$

where the subscript 1 refers to the compartments.

These movements will cause the point G to move to G' and if w is the specific weight of sea water then GG' is given by

$$V \times w \times \overline{GG'} = 4\left(V_1 \times w \times \frac{I_1}{V_1} \times \theta\right)$$

$\therefore \quad \overline{GG'} = \frac{4I_1}{V} \times \theta$

Also $\quad \overline{GG'} = \overline{NG} \times \theta$

$\therefore \quad \overline{NG} = \frac{4I_1}{V}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where N is the point of intersection between the new vertical through G' and the original vertical through G.

The unbalanced righting couple is now ($W\ \overline{MN} \sin\theta$)

$\therefore$ Metacentric height $= \overline{MN}$

From Fig. Ex. 4.9 (b)

$$\overline{MN} = \overline{GM} - \overline{NG}$$
$$= \overline{BM} - \overline{BG} - \overline{NG}$$

$$\overline{BM} = \frac{I}{V} = \frac{\frac{1}{12}\times 4\times(4)^3}{12.8} = 1.67$$

$$\overline{BG} = (0.425 - 0.4) = 0.025 \text{ m}$$

$$\overline{NG} = \frac{4I_1}{V} = \frac{4\left(\frac{1}{12}\times 4\times 2^3\right)}{12.8} = 0.83 \text{ m}$$

$\therefore$ $\overline{MN} = (1.67 - 0.025 - 0.83)$

$= 0.815$ m

Example 4.10. *A battle ship weighs 127.53 MN (13 000 tonnes). On filling the ship's boats on one side with water weighing 588.6 kN (60 tonnes) and its mean distance from the centre of the ship being 10 m, the angle of displacemnt of the plumb line is 2° 16′. Determine the metacentric height.*

Solution

SI Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W\tan\theta}$$

$w = 588.6$ kN $= 0.588\ 6$ MN

$W = (127.53 + 0.588\ 6) = 128.1186$ MN

$x = 10$ m

and $\theta = 2°16'$, so $\tan(2° 16') = 0.039\ 6$

$\therefore$ By substitution, we get

$$\overline{GM} = \frac{0.5886\times 10}{128.11860\times 0.0396} = 1.16 \text{ m}$$

Metric Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W\tan\theta}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

w = 60 tonnes; x = 10 m

W = (13 000 + 60) = 13 060 tonnes

and θ = 2° 16′ so tan (2°16°) = 0.039 6

∴ By substitution, we get

$$\overline{GM} = \frac{60 \times 10}{13\,060 \times 0.0396} = 1.16 \text{ m}$$

Example 4.11. *In finding the metacentre of a ship of 98.1 MN (10 000 tonnes) displacement, a weight of 490.5 kN (50 tonnes) at a distance of 6 m from the longitudinal centre plane causes the ship to heel through an angle of 3°. What is the metacentric height? Hence find the angle of heel and its direction when the ship is going ahead and 2.833 6 MW (3 850 metric h.p.) is being transmitted to a single propeller shaft which is rotated at a speed of 3π rad/s (90 r.p.m).*

Solution

SI Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W \tan \theta}$$

w = 490.5 kN = 0.490 5 MN

W = 98.1 MN ; x = 6 m

θ = 3°, so tan 3° = 0.052 4

∴ By substitution, we get

$$\overline{GM} = \frac{0.4905 \times 6}{98.1 \times 0.0524} = 0.573 \text{ m}$$

If T is torque in N.m and ω is angular velocity in rad/s, then power transmitted in watts is

$P = T\omega$

or $2.8336 \times 10^6 = T \times 3\pi$

∴ T = 300 654.3 N.m

If θ is the angle of heel, then we have

$\overline{W} \cdot \overline{GM} \tan \theta = T$

or $98.1 \times 10^6 \times 0.573 \times \tan \theta = 300\ 654.3$

or $\tan \theta = 0.005\ 35$

∴ $\theta = 0° 18.4'$

Metric Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W \tan \theta}$$

w = 50 tonnes ; x = 6 m

W = 10 000 tonnes

θ = 3°, so tan 3° = 0.0524

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore$ By substitution, we get

$$\overline{GM} = \frac{50 \times 6}{10000 \times 0.0524} = 0.573 \text{ m}$$

Horse power transmitted

$$= \frac{2\pi NT}{75 \times 60}$$

where N is in r.p.m., and T is torque in kg(f)-m.

Thus by substitution, we get

$$3\ 850 = \frac{2\pi \times 90 \times T}{75 \times 60}$$

$$\therefore \quad T = 30\ 637 \text{ kg-m}$$

If θ is the angle of heel, then we have

$$W\ \overline{GM}\ \tan\theta = T$$

or $10\ 000 \times 1000 \times 0.573 \times \tan\theta = 30\ 637$

$$\therefore \quad \tan\theta = \frac{30\ 637}{10\ 000 \times 1000 \times 0.573}$$

$$= 0.005\ 35$$

$$\therefore \quad \theta = 0° 18.4'$$

Example 4.12. *A merchant ship coming into port has a draught of 1.25 m. After unloading its cargo it has a draught of 1 m. Find the ratio of the periodic times before and after leaving the cargo if the breadth of the ship is 7 m and it is assumed that the centre of gravity remains at the water line.*

Solution

The periodic time is given by Eq. 4.7 as

$$T = 2\pi \left\{ \frac{k_G^2}{gGM} \right\}^{1/2}$$

Let T_1 and T_2 be the periodic times for the ship before and after leaving the cargo respectively.

In the first case since the centre of gravity lies in the water line,

$$\overline{BG} = \frac{1.25}{2} = 0.625 \text{ m}$$

Now if l is the length of the ship at the water level then

$$I = \frac{1}{12} \times l \times (7)^3 \text{ m}^4$$

and $$V = (l \times 7 \times 1.25) \text{ m}^3$$

$$\overline{BM} = \frac{I}{V}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{\frac{1}{12}\times l\times(7)^3}{(l\times 7\times 1.25)} = 3.27 \text{ m}$$

Thus $\overline{GM} = \overline{BM} - \overline{GM}$

$= (3.27 - 0.625) = 2.645$ m

$\therefore$ By substitution, we get

$$T_1 = 2\pi\sqrt{\frac{k_G^2}{g\times 2.645}}$$

Similarly after leaving the cargo

$$\overline{BG} = \frac{1}{2} = 0.5 \text{ m}$$

$$I = \frac{1}{12}\times l\times (7)^3 \text{ m}^4$$

$$V = (l\times 7\times 1) \text{ m}^3$$

$\therefore$ $$\overline{BM} = \frac{\frac{1}{12}\times l\times(7)^3}{(l\times 7\times 1)} = 4.08 \text{ m}$$

Thus $\overline{GM} = \overline{BM} - \overline{BG}$

$= (4.08 - 0.5) = 3.58$ m

$\therefore$ By substitution, we get

$$T_2 = 2\pi\sqrt{\frac{k_G^2}{g\times 3.58}}$$

$\therefore$ $$\frac{T_1}{T_2} = \sqrt{\frac{3.58}{2.645}} = 1.16$$

Example 4.13. *A log of wood of square section 0.36 m × 0.36 m and specific gravity 0.8 floats in water. One edge is depressed and released causing the log to roll. Estimate the period of a roll.*

Solution

Let x be the depth of immersion and l be the length of the log, then

$(l \times 0.36\times 0.36) \times 9\,810\times 0.8 = (x \times 0.36 \times l) \times 9\,810$

$x = 0.288$ m

Thus height of the centre of buoyancy above the bottom of the log

$$= \left(\frac{0.288}{2}\right) = 0.144 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore \quad \overline{BG} = (0.18 - 0.144) = 0.036$ m

As $\overline{BM} = \dfrac{I}{V}$

$I = \dfrac{1}{12} \times l \times (0.36)^3$

$V = (0.36 \times 0.288 \times l)$

$\therefore \quad \overline{BM} = \dfrac{\frac{1}{12} \times l \times (0.36)^3}{(0.36 \times 0.288 \times l)} = 0.037\,5$ m

$\therefore \quad \overline{GM} = \overline{BM} - \overline{BG}$

$\therefore \quad = (0.0375 - 0.036) = 0.001\,5$ m

Time period of rolling is given by

$$T = 2\pi \sqrt{\frac{k_G^2}{g\ \overline{GM}}}$$

$$k_G^2 = \frac{(0.36)^2}{12} = 0.010\,8 \text{ m}^2$$

$$\therefore \quad T = 2\pi \sqrt{\frac{0.0108}{9.81 \times 0.0015}} = 5.38 \text{ s.}$$

Example 4.14. *A wooden cone weighing 88 N floats with its apex downwards in a liquid of specific gravity 0.8. If the specific gravity of wood is 0.5, find what weight of a steel piece of specific gravity 7.8 suspended from the apex of the cone by a cord will just suffice to submerge the cone and what would then be the tension in the cord.*

Solution

Let W be the weight of the steel piece and V_1 and V_2 be the volumes of the cone and the steel piece respectively. Then we have

$$(W + 88) = (V_1 + V_2) \times 9\,810 \times 0.8 \quad \text{... (i)}$$

$$88 = 9\,810 \times 0.5 \times V_1 \quad \text{... (ii)}$$

and $$W = 9\,810 \times 7.8 \times V_2 \quad \text{... (iii)}$$

By substituting the values of $V1$ and $V2$ from Eq. (ii) and (iii) in Eq. (i), we get

$$(W + 88) = \left(\frac{88}{9810 \times 0.5} + \frac{W}{9810 \times 7.8}\right) \times 9810 \times 0.8$$

or $$W = 58.83 \text{ N}$$

The tension in the cord is equal to the weight of the steel piece in the liquid.

The weight of the steel piece in the liquid is equal to the weight of the steel piece in air minus the loss in the weight of the steel piece in the liquid.

Also the loss in the weight of the steel piece in the liquid is equal to the weight of the liquid displaced by it.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The weight of the liquid displaced by the steel piece = $\left(\frac{W \times 0.8}{7.8}\right)$

$\therefore$ Tension in the cord = $\left[W - \left(\frac{W \times 0.8}{7.8}\right)\right]$

$$= 58.83\left(1 - \frac{0.8}{7.8}\right) = 52.8 \text{ N}$$

Example 4.15. *In Fig. Ex. 4.15 (a) is shown a float valve regulating the flow of oil of specific gravity 0.8 into a cistern. The spherical float is 150 mm in diameter. AOB is a weightless link carrying the float at one end and a valve at the other end which closes the pipe through which oil flows into the cistern. The link is mounted on a frictionless hinge at O and the angle AOB is 135°. The length OA is 0.2 m and the distance between the centre of the float and the hinge is 0.5 m. When the flow is stopped AO will be vertical. The valve is to be pressed on the seat with a force of 10 N to completely stop the flow into the cistern. It was observed that the flow of oil is stopped when the free surface of oil in the cistern is 0.35 m below the hinge. Determine the weight of the float.*

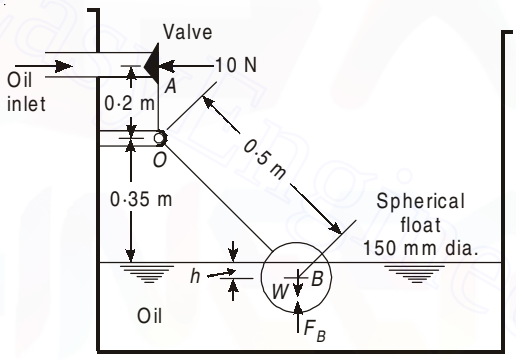


Figure Ex. 4.15 (a)

Solution

The depth of the centre of the spherical float below the oil surface in the cistern is

$$h = (0.5 \cos 45° - 0.35)$$
$$= 3.55 \times 10^{-3} \text{ m} = 3.55 \text{ mm}$$

The forces acting on the float are the weight of the float W acting vertically downwards at the centre of the float and the force of buoyancy F_B acting vertically upwards at the centre of buoyancy. Both these forces are acting along the same vertical line.

According to the principle of floatation the force of buoyancy F_B is equal to the weight of the oil displaced by the float.

Thus

$$F_B = (0.8 \times 9\,810)V = 7\,848V \text{ N}$$

where V is the volume of the oil displaced by the float in m^3.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

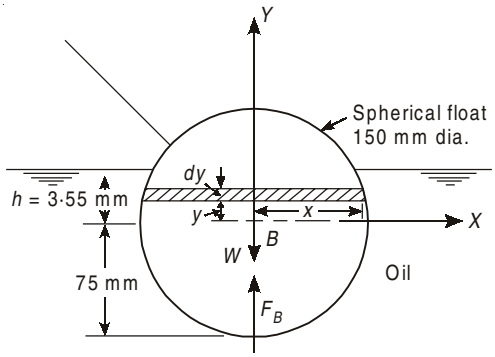


Figure Ex. 4.15 (b)

For determining the volume of oil displaced by the float consider on the surface of the float an element of radius x and thickness dy at a distance y from x-axis as shown in the Fig. Ex. 4.15 (b). The volume of the oil displaced by the portion of the float corresponding to the small element is

$$dV = \pi x^2 dy$$
$$= x\,[(75)^2 - y^2]\,dy$$

Thus the volume of the oil displaced by the float is

$$V = \pi \int_{-75}^{3.55} \left[(75)^2 - y^2\right] dy$$

$$= \pi \left[(75)^2 y - \frac{y^3}{3}\right]_{-75}^{3.55}$$

$$= 946\,260 \text{ mm}^3$$
$$= 946.260 \times 10^{-6} \text{ m}^3$$

Note. In this case since the portion of the float lying between its centre and the oil surface is very small, it may approximately be considered to be of cylindrical shape with radius 75 mm and height 3.55 mm. The volume of the oil displaced by the float may then be obtained as

$$V = \left[\frac{2}{3}\pi(75)^3 + \pi(75)^2 \times 3.55\right]$$
$$= 946\,307 \text{ mm}^3$$
$$= 946.307 \times 10^{-6} \text{ m}^3$$

which is very close to the value of V obtained above by the exact method.

The force of buoyancy is obtained as

$$F_B = 0.8 \times 9\,810 \times 946.26 \times 10^{-6}$$
$$= 7.426 \text{ N}$$

Since the hinge is frictionless, the moment of the net vertical force acting on the float about the hinge O is transmitted to the valve A without any loss. The net vertical force acting on the float is

$$(F_B - W) = (7.426 - W)$$

Thus taking moments about hinge O, we get

$$(7.426 - W)\,0.5 \cos 45° = 10 \times 0.2$$

$$\therefore \quad W = 1.769 \text{ N}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. When a body is immersed in a fluid, either wholly or partially, an upward force is exerted by the fluid on the body which tends to lift (or buoy) up the body. This tendency for an immersed body to be lifted up in the fluid, due to an upward force opposite to the action of gravity is known as 'buoyancy'.
2. The force tending to lift up a body immersed in a fluid either wholly or partially is known as 'buoyant force' or 'force of buoyancy' or 'upthrust'.
3. The point of application of the force of buoyancy on the body is known as 'centre of buoyacy'.
4. According to Archimedes' principle the buoyant force is equal to the weight of the fluid displaced by the body.
5. The principle of floatation states that the weight of a body floating in a fluid is equal to the buoyant force which in turn is equal to the weight of the fluid displaced by the body.
6. For a body floating in a fluid the centre of buoyancy coincides with the centroid of the volume of the fluid displaced by the body.
7. 'Metacentre' is defined as the point of intersection between the axis of a floating body and a vertical line passing through the new centre of buoyancy obtained when the floating body is slightly tilled or it undergoes a small angular displacement (or angle of heel) θ
8. The distance between the centre of gravity G and metacentre M of a floating body (i.e, GM) as $\theta \rightarrow 0$, is known as 'metacentric height'.
9. Conditions of equilibrium of a floating body and a wholly submerged body are as indicated below:

Conditions of equilibrium	*Floating body*	*Wholly submerged body*
(i) Stable equilibrium	M is above G	B is above G
(ii) Unstable equilibrium	M is below G	B is below G
(iii) Neutral equilibrium	M and G coincide	B and G coincide

10. The value of the metacentric height determined experimentally is given as

$$GM = \frac{wx}{W \tan\theta}$$

where

w = movable weight;
x = distance through which w is moved;
W = weight of the ship or floating body including w; and
θ = angle through which the ship or floating body is tilted.

Further

$$\tan\theta = \frac{d}{l}$$

where

l = length of the plumbline or pendulum; and
d = distnace moved by the plumbline or pendulum on the horizontal scale.

11. The metacentric height determined by analytical method is given as

$$GM = \frac{I}{V} - \overline{BG}$$

where

I = moment of inertia of the cross-sectional area of the ship or floating body about the longitudinal axis;
V = volume of liquid displaced by the ship or floating body; and
BG = distance between centre of buoyancy and centre of gravity of the ship or floating body.

12. A ship or a floating body may have two types of oscillatory motions, viz., 'rolling' and 'pitching.' The oscillatory motion of a ship or a floating body about its longitudinal axis is designated as rolling, and that about its transverse axis is designated as pitching.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

13. The time period of transverse oscillation (i.e, oscillation about the longitudinal axis) or rolling of a ship or a floating body is given by

$$T = 2\pi\left[\frac{k_G^2}{g\overline{GM}}\right]^{\frac{1}{2}}$$

where

T = time period of transverse oscillation;
k_G = radius of gyration of the floating body about its centre of gravity;
$\overline{GM}$ = metacentric height; and
g = acceleration due to gravity.

PROBLEMS

4.1 Explain the terms: Force of buoyancy and centre of buoyancy.

4.2 Explain the terms: Metacentre and metacentric height.

4.3 Explain the three states of equilibrium of a floating body in terms of the magnitude and direction of righting couple acting on the body as a result of slight angular displacement.

4.4 What is meant by stability of a floating body? Explain the stability of a floating body with reference to its metacentric height. Give neat sketches.

4.5 Define the terms: Stable, unstable and neutral equilibrium.

4.6 Describe briefly the experimental method of determination of the metacentric height of a floating object.

4.7 An iceberg weighing 8 976 N/m^3 [915 kg (f)/m^3] floats in the ocean with a volume of 600 m^3 above the surface. Determine the total volume of the iceberg if specific weight of ocean water is 10 055 N/m^3 [1 025 kg (f)/m^3].

[*Ans.* 5 591 m^3]

4.8 An empty balloon and its equipment weighs 441.45 N [45 kg (f)]. When inflated with gas weighing 5.415 N/m^3 [0.552 kg(f)/m^3], the balloon is spherical and 7 m in diameter. What is the maximum weight of cargo that the balloon can lift, assuming air to weigh 12.066 N/m^3 [1.23 kg (f)/m^3].

[*Ans.* 753.03 N { 76.76 kg (f)}]

4.9 A cylinder has a diameter 0.3 m and a specific gravity of 0.75. What is the maximum permissible length in order that it may float in water with its axis vertical ? [*Ans.* 0.245 m]

4.10 A buoy carrying a beacon light has the upper portion cylindrical 2.5 m diameter and 2 m deep. The lower portion which is curved displaces a volume of 0.4 m^3 and its centre of buoyancy is situated 2.5 m below the top of the cylinder. The centre of gravity of the whole buoy and beacon is situated 1.5 m below the top of the cylinder and the total displacement is 19.62 kN (2 tonnes). Find the metacentric height if the specific weight of sea water is 10 055 N/m^3 [1 025 kg (f)/m^3] [*Ans.* 0.505 m]

4.11 A rectangular pontoon weighing 1 716.75 kN (175 tonnes) has a length of 20 m. The centre of gravity is 0.3 m above the centre of the cross-section and the metacentric height is to be 1.25 m when the angle of heel is 9°. The free board must not be less than 0.6 m when the pontoon is vertical. Find the breadth and the height of the pontoon, if it is floating in fresh water.

[*Ans.* 5.792 m, 2.11 m]

4.12 A ship is 80 m long and 12 m broad and has a displacement of 14 715 kN (1500 tonnes). When a weight of 490.5 kN is moved 5 m across the deck it inclines the ship 6°. The moment of inertia of the section of the ship at water plane about its fore-and-aft axis is 60 per cent of the moment of inertia of the circumscribing rectangle. The centre of buoyancy is 2.5 m below the water line. Find the position of the metacentre and the centre of gravity. Specific weight of sea water is 10 055 N/m^3 [1025 kg (f)/m^3].

[*Ans.* 2.224 m and 0.638 m above water line]

4.13 A solid cylinder is to be 0.25 m in diameter. The base of axial length 25 mm is to be of metal which has a specific gravity 7, and the remainder of material which has a specific gravity 0.5. Find the maximum overall length of the cylinder in order that it may float in water in the stable equilibrium with its axis vertical.

[*Ans.* 0.801 m]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

4.14 A ship has a displacement of 49 050 kN. The second moment of area of the water line section about a fore- and-aft axis is 12 × 104 m^4 and the centre of buoyancy is 2.8 m below the centre of gravity. The radius of gyration is 4 m. Calculate the time period of oscillation. Take the weight of sea water as 10 055 N/m^3 [1025 kg (f)/m^3].

[*Ans.* 1.72 s]

4.15 Find the value of K_G for a ship which has a period of rolling of 20s. The displacement is 98 100 kN (10000 tonnes), the second moment of the load-water-plane above its fore-and-aft axis is 30× 102 m^4 and the centre of buoyancy is 3.25 m below the centre of gravity. Sea water weighs 10 055 N/m^3 [1 025 kg(f)/m^3].

[*Ans.* 4.174 m]

4.16 A solid conical float of wood weighing 7 553.7 N/m^3 [770 kg(f)/m^3] is to float in a liquid weighing 9417.6 N/m^3 [960 kg(f)/m^3]. Find the least apex angle in order that it may float with apex down and its axis vertical. [***Ans.*** 30° 50']

4.17 A hollow cylindrical of diameter 2 m and length 3 m weighs 29.43 kN [3 tonnes] and has its centre of gravity at the mid point of the longitudinal axis. Show that it will not float in sea water with its longitudinal axis vertical.

Neglecting the thickness of the plates, find to what depth the inside of the buoy should be filled with concrete, of specific weight 22 563 N/m^3 [2 300 kg/m^3], to give a depth of immersion of 2.5 m and what is then the meta-centric height? Sea water weighs 10 055 N/m^2 [1 025 kg(f)/m^3].

[*Ans.* 0.6984 m; 0.572 m]

4.18 A conical buoy floating with its apex pointing downward is 3.5 m high and 2 m diameter. Calculate its weight if it is just stable when floating in sea water weighing 10 055 N/m^3 [1025 kg(f)/m^3].

[*Ans.* 29.127 kN {2 969 kg(f)}]

4.19 A floating buoy in the Bombay harbour is to be assisted in floating upright by a submerged weight of concrete attached to the bottom of the buoy. How many m^3 of concrete weighing 22 759 N/m^3 [2 320 kg (f)/m^3] must be provided to get a net downward pull of 2217 N [226 kg(f)] from the weight? Explain the principle involved.

[*Ans.* 0.1745 m^3]

4.20 A rectangular pontoon 6 m by 3 m in plan, floating in water has a depth of immersion of 0.9 m and is subjected to a torque of 7848 N.m [800kg(f)-m] about the longitudinal axis. If the centre of gravity is 0.7 m up from the bottom, estimate the angle of heel. [*Ans.* 4°50']

4.21 A solid cylinder 1 m diameter and 0.75 m high is of uniform specific gravity 0.85. Calculate the periodic time of small oscillations when the cylinder floats with its axis vertical in still water.

[**Hint**. Take $k_G^2 = \left(\frac{l^2}{12} + \frac{r^2}{4}\right)$; where l is the length and r is the radius of the cylinder.]

[*Ans.* 3.25 s]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Liquids in Relative Equilibrium

Chapter 5

5.1 INTRODUCTION

In the previous chapters the fluids have been assumed to be in static equilibrium relative to the containers, which are also assumed to be at rest. However, if a vessel containing a fluid is made to move with a constant acceleration, then although the fluid is in relative equilibrium and is at rest with respect to its container, the fluid will take up a new position under the effect of the acceleration imparted to its container and come to rest in this new position relative to the container. Since the fluid after attaining a new position is still in static condition relative to its container, the laws of hydrostatics can be applied to evaluate the fluid pressure. Further in such cases, there being no relative motion of the particles of the fluid, there are no shear stresses and therefore the fluid pressure is everywhere normal to the surface on which it acts.

A vessel containing a fluid may be subjected to either translatory motion, in horizontal or vertical direction, or a rotational motion at constant accelerations. The new positions occupied by the fluids when their containers are subjected to different motions as well as the pressures exerted by the fluids may be determined as indicated in the following sections.

5.2 FLUID MASS SUBJECTED TO UNIFORM LINEAR ACCELERATION

Consider a small element of fluid of size $\delta x \times \delta y \times \delta z$ as shown in Fig. 5.1, in a fluid mass which is subjected to acceleration α which has components α_x, α_y and α_z in x, y and z directions respectively. Let p be the pressure intensity at the mid-point O of the element. Then the pressure intensity on the left-hand face of the element is $\left[p-\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]$ and the pressure intensity on the right-hand face of the element is $\left[p+\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]$. The corresponding pressure forces on the left-hand and the right-hand faces of the element are $\left[p-\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]\delta y\,\delta z$ and $\left[p+\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]\delta y\,\delta z$ respectively. In the same manner

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the pressure intensities and the corresponding pressure forces on the other faces of the element may be obtained as shown in Fig. 5.1. Further if w is the specific weight of the fluid then the weight of the element acting vertically downwards is ($w\,\delta x\delta y\delta z$).

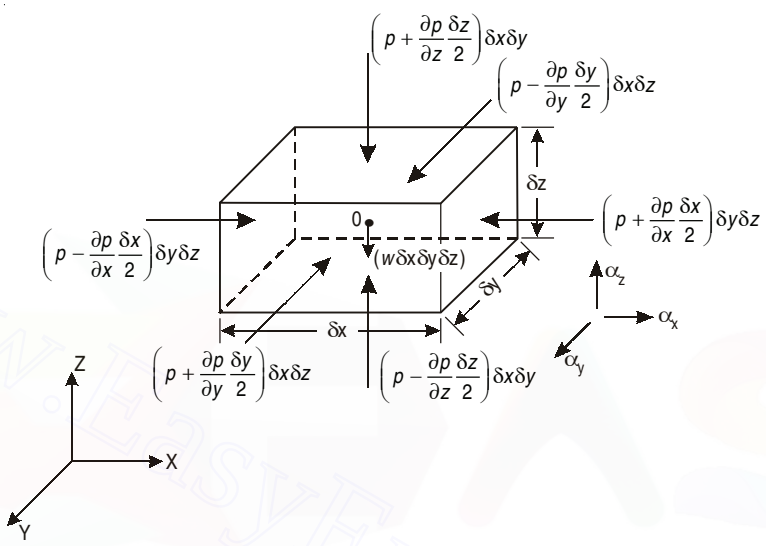


Figure 5.1 *Fluid mass subjected to acceleration*

According to Newton's second law of motion the net force acting on the fluid element in any direction is equal to the product of the mass of the element and the acceleration in the same direction as that of the force. Thus applying the Newton's second law of motion in the x, y and z directions the following equations are obtained

In the x-direction

$$\left[\left(p-\frac{\partial p}{\partial x}\frac{\delta x}{2}\right)\delta y\delta z-\left(p+\frac{\partial p}{\partial x}\frac{\delta x}{2}\right)\delta y\delta z\right]$$

$$= \frac{w}{g}(\delta x\delta y\delta z)\,\alpha_x$$

or $$-\frac{\partial p}{\partial x}\,\delta x\delta y\delta z = \frac{w}{g}(\delta x\delta y\delta z)\,\alpha_x$$

or $$-\frac{\partial p}{\partial x} = \frac{w}{g}\alpha_x \qquad \ldots (5.1)$$

In the y-direction

$$\left[\left(p-\frac{\partial p}{\partial y}\frac{\delta y}{2}\right)\delta x\delta z-\left(p+\frac{\partial p}{\partial y}\frac{\delta y}{2}\right)\delta x\delta z\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{w}{g}(\delta x \delta y \delta z)\,\alpha_y$$

or $$-\frac{\partial p}{\partial y}\delta x\,\delta y \delta z = \frac{w}{g}(\delta x \delta y \delta z)\,\alpha_y$$

or $$-\frac{\partial p}{\partial y} = \frac{w}{g}\alpha_y \qquad \ldots(5.2)$$

In the z-direction

$$\left[\left(p - \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x \delta y - \left(p + \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x \delta y\right] - (w \delta x \delta y \delta z)$$

$$= \frac{w}{g}(\delta x \delta y \delta z)\,\alpha_z$$

or $$\left(-\frac{\partial p}{\partial z} - w\right)\delta x\,\delta y \delta z = \frac{w}{g}(\delta x \delta y \delta z)\,\alpha_z$$

or $$-\frac{\partial p}{\partial z} = w\left(1 + \frac{\alpha_z}{g}\right) \qquad \ldots\ (5.3)$$

In vector notation Eqs 5.1, 5.2 and 5.3 may be expressed as

$$-\text{grad}\ p = \frac{w}{g}(\alpha + kg)$$

or $$-\left[i\frac{\partial p}{\partial x} + j\frac{\partial p}{\partial y} + k\frac{\partial p}{\partial z}\right] = \frac{w}{g}\left[i\alpha_x + j\alpha_y + k(\alpha_z + g)\right]$$

where i, j, k are the unit vectors parallel to x, y, z axes respectively.

Equations 5.1, 5.2 and 5.3 give the pressure gradients in the x, y and z directions respectively, from which the pressure variation throughout the fluid mass may be determined. For simplicity, the acceleration α may be considered to be in the x–z plane, so that it has only two components α_x and α_z and the component $\alpha_y = 0$. The pressure then varies only in the x and z directions. Hence the change in pressure dp may be expressed as

$$dp = \frac{\partial p}{\partial x}dx + \frac{\partial p}{\partial z}dz$$

For a surface of constant pressure $dp = 0$ and hence

$$\frac{dz}{dx} = -\frac{(\partial p/\partial x)}{(\partial p/\partial z)}$$

Introducing the values of $(\partial p/\partial x)$ and $(\partial p/\partial z)$ from Eqs 5.1 and 5.3, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{dz}{dx} = -\frac{a_x}{(g + a_z)} \qquad ...(5.4)$$

The term (dz/dx) represents the slope of the surface of constant pressure and its value is given by Eq. 5.4. The negative sign shows that the surface of constant pressure is slopping downwards.

For constant acceleration, since (dz/dx) is constant, a surface of constant pressure is a plane surface having a constant slope. Further as indicated in Section 5.1, since the pressure distribution in this case is hydrostatic, all the planes of constant pressure are parallel to each other.

The most common practical examples of the fluid mass subjected to uniform linear acceleration are those of the horizontal and vertical accelerations imparted to the containers or tanks containing liquid. Both these cases are discussed in the following sections.

5.3 LIQUID CONTAINERS SUBJECTED TO CONSTANT HORIZONTAL ACCELERATION

Consider a tank partly filled with liquid and given a constant horizontal acceleration α as shown in Fig. 5.2. Thus for this case $\alpha_x = \alpha$ and $\alpha_z = 0$ and introducing these values in Eq. 5.4, the slope of the surface of constant pressure is obtained as

$$\frac{dz}{dx} = -\frac{\alpha}{g} \qquad \ldots (5.5)$$

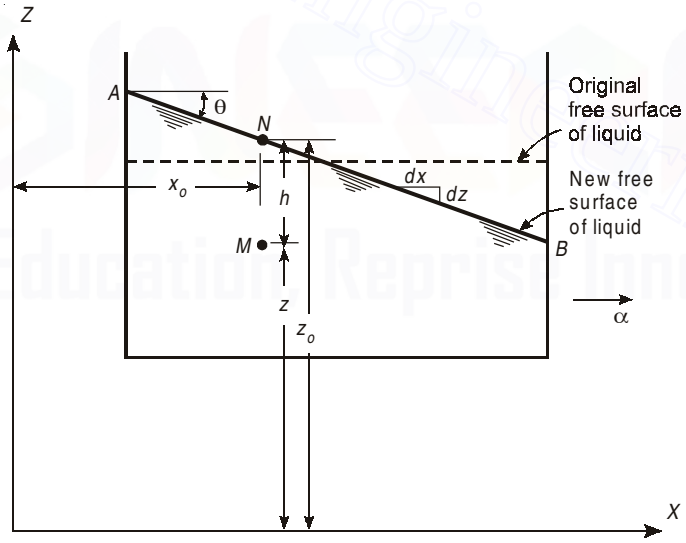


Figure 5.2 *Tank containing liquid subjected to constant horizontal acceleration*

Equation 5.5 indicates that in this case the surface of constant pressure is a downward slopping plane surface. The free surface of the liquid is a surface of constant pressure, because the pressure at the free surface is equal to atmospheric pressure which is constant. As such under the effect of the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

horizontal acceleration the free surface of the liquid will become a downward slopping inclined plane, with the liquid rising at the back end and the liquid falling at the front end. As shown in Fig. 5.2, if *AB* represents the new free surface of the liquid which is inclined at an angle θ with the horizontal, then

$$\tan\theta = -\frac{dz}{dx} = \frac{\alpha}{g} \qquad \ldots (5.6)$$

It may however be noted that angle θ will be the same for a container moving with an acceleration on one side or retardation of the same magnitude in the opposite direction.

If an open tank completely filled with liquid is imparted horizontal acceleration then it is evident that a portion of the liquid will spill out from the tank and new free surface with its slope as given by Eq. 5.6 will be developed. Similar spilling of liquid would result even from a partly filled open tank when horizontal acceleration imparted to the tank is increased beyond a certain value. On the other hand if a closed tank completely filled with liquid is imparted horizontal acceleration then the liquid cannot spill out from the tank and also there can be no adjustment in the surface elevation of the liquid. But Eq. 5.6 holds good for this case also and hence at the top of the tank starting from its front end, an imaginary slopping free surface with its slope as given by Eq. 5.6 may be considered. Further if the tank is completely filled with liquid under pressure, then the imaginary free surface at the top of the tank will have to be shifted above the top of the tank by a distance equal to the pressure head of the liquid. However, if a closed tank partly filled with liquid is subjected to horizontal acceleration then the liquid cannot spill out from the tank but the slopping free surface or the surface of constant pressure is developed with its slope as given by Eq. 5.6.

Pressure p at any point in the liquid may be determined by integrating Eq. 5.1 and 5.3. For this case $\alpha_x = \alpha$ and $\alpha_z = 0$, Eqs 5.1 and 5.3 become

$$-\frac{\partial p}{\partial x} = \frac{w}{g}\alpha \text{ and } -\frac{\partial p}{\partial z} = w$$

Thus
$$p = \int dp = \int \frac{\partial p}{\partial x}dx + \int \frac{\partial p}{\partial z}dz$$

$$= -\left(\frac{w}{g}\alpha\right)x - wz + C$$

where C is the constant of integration. The value of C may be determined by considering a point N in the free surface at $x = x_0$ and $z = z_0$ for which $p = p_a$ (the atmospheric pressure) and introducing these values in the above expression.

$$C = p_a + \left(\frac{w}{g}\alpha\right)x_0 + wz_0$$

Thus
$$p = p_a + \left(\frac{w}{g}\alpha\right)(x_0 - x) + w(z_0 - z) \qquad \ldots(5.7)$$

As shown in Fig. 5.2 for any point M in the liquid at a vertical section $x = x_0$ and vertical depth h below the free surface of liquid since $(z_0 - z) = h$, the pressure intensity is given by Eq. 5.7 as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$p = p_a + wh \quad \ldots (5.8)$$

Equation 5.8 represents the absolute pressure at any point in the liquid mass subjected to horizontal acceleration, and the corresponding gage pressure at this point is

$$(p - p_a) = wh \quad \ldots (5.9)$$

The expression for the pressure at any point in a liquid mass subjected to horizontal acceleration may also be obtained by an alternative method as showing in Fig. 5.3.

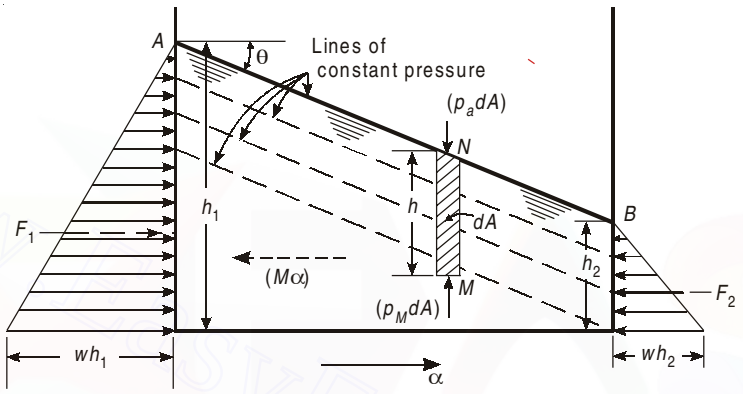


Figure 5.3 *Pressure distribution in liquid subjected to constant horizontal acceleration*

In order to find the pressure at any point *M* in the liquid consider an elementary prism *MN* of height *h* and cross-sectional area *(dA)*, as shown in Fig. 5.3. Since there is no vertical acceleration given to the tank, the only force acting on the elementary prism in the vertical direction are the atmospheric pressure force $(p_a dA)$ at the top end of the prism acting downwards, the weight of the element $(w \times hdA)$ acting in the downward direction; and the pressure force $(p_M dA)$ at the bottom end of the prism acting upwards. The elementary prism is in equilibrium under the action of these forces and hence

$$(p_M dA) - (p_a dA) - (w \times hdA) = 0$$

or $$p_M = (p_a + wh)$$

which is same as Eq. 5.8 derived earlier.

It is therefore, seen that the pressure head at any point in a liquid mass subjected to a constant horizontal acceleration, is equal to the height of the liquid column above that point. Therefore, the pressure distribution in a liquid mass subjected to a constant horizontal acceleration is same as hydrostatic pressure distribution. The planes of constant pressure are therefore, parallel to the inclined surface as shown in Fig. 5.3. Further if h_1 and h_2 are the depths of liquid developed at the rear and the front ends of the tank, then the total pressures exerted on the rear and the front sides of the tank are

$$F_1 = \left[w \times (b \times h_1) \times \frac{h_1}{2}\right] = \frac{wbh_1^2}{2} \quad \ldots(5.10)$$

and

$$F_2 = \left[w \times (b \times h_2) \times \frac{h_2}{2}\right] = \frac{wbh_2^2}{2} \quad \ldots(5.11)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where b is the width of the tank perpendicular to the plane of the paper, and w is the specific weight of the liquid. Since this difference of pressure is created by the acceleration α acting on the total mass M of the liquid contained in the tank it may be shown that

$$(F_1 - F_2) = (M \times \alpha) \quad \ldots (5.12)$$

5.4 LIQUID CONTAINERS SUBJECTED TO CONSTANT VERTICAL ACCELERATION

Consider a tank filled with liquid of specific weight w, upto a depth y, subjected to a constant vertical upward acceleration α as shown in Fig. 5.4. Thus for this case $\alpha_x = 0$ and $\alpha_z = \alpha$, and introducing these values in Eq. 5.4, $(dz/dx) = 0$. It, therefore, follows that in this case the surfaces of constant pressure are horizontal. Hence the free surface of the liquid in the tank will always remain horizontal. But the pressure at any point in the liquid will be different from that which would exist if the vessel was stationary. The pressure p at any point in the liquid may be determined by integrating Eqs 5.1 and 5.3 which may be expressed for this case as

$$-\frac{\partial p}{\partial x} = 0 \text{ and } -\frac{\partial p}{\partial z} = w\left(1+\frac{\alpha}{g}\right)$$

Thus

$$p = \int dp = \int \frac{\partial p}{\partial x}dx + \int \frac{\partial p}{\partial z}dz$$

$$= -w\left(1+\frac{\alpha}{g}\right)z + C$$

where C is the constant of integration. The value of C may be determined by considering a point in the free surface for which $z = z_0$ (say) and $p = p_a$ (the atmospheric pressure) and introducing these values in the above expression

$$C = p_a + w\left(1+\frac{\alpha}{g}\right)z_0$$

Thus

$$p = p_a + w\left(1+\frac{\alpha}{g}\right)(z_0 - z) \quad \ldots(5.13)$$

For any point in the liquid at a vertical depth h below the free surface of the liquid $(z_0 - z) = h$, the pressure intensity is given by Eq. 5.13 as

$$p = p_a + wh\left(1+\frac{\alpha}{g}\right) \quad \ldots(5.14)$$

Equation 5.14 represents the absolute pressure at any point in the liquid mass subjected to a constant vertical upward acceleration. The corresponding gage pressure at this point is

$$(p - p_a) = wh\left(1+\frac{\alpha}{g}\right) \quad \ldots(5.15)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The expression for the pressure at any point in a liquid mass subjected to vertical upward acceleration may also be obtained by an alternative method as indicated below.

Consider a vertical elementary prism of liquid AB of cross sectional area (dA) and height h below the free surface of the liquid as shown in Fig. 5.4. If p_a is the atmospheric pressure intensity then the pressure force acting on the top end of the prism is ($p_a\, dA$) acting in the vertical downward direction.

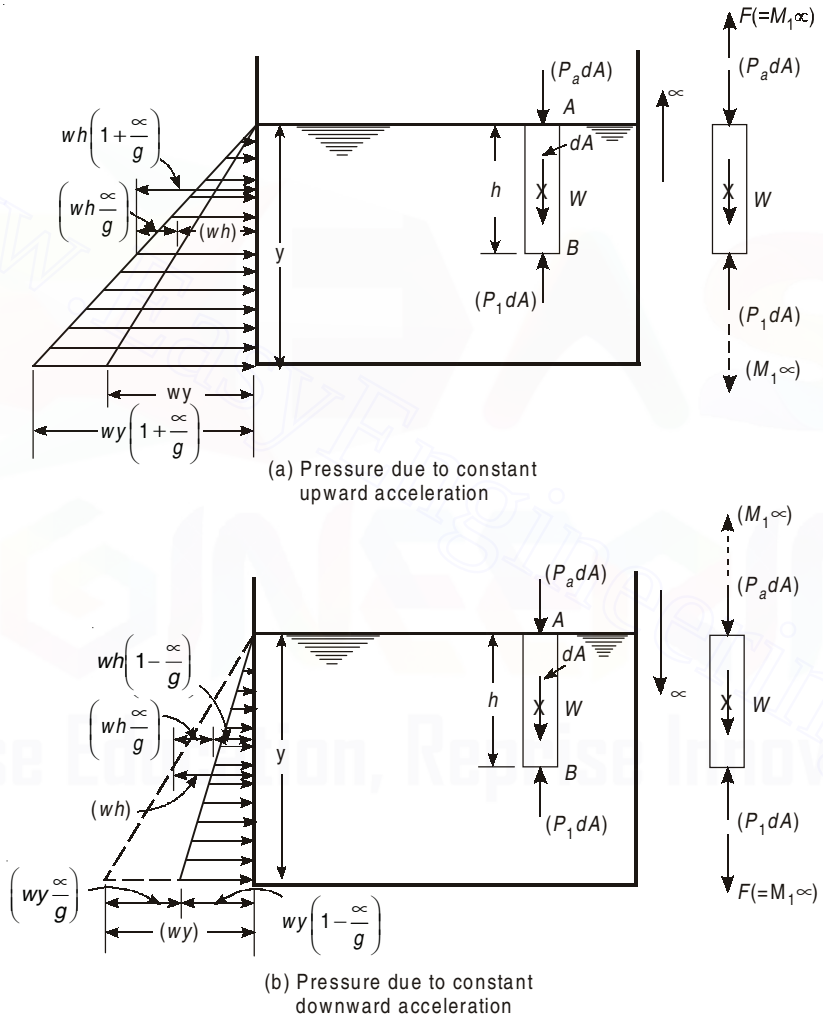


Figure 5.4 *Tank containing liquid subjected to a constant vertical : (a) upward (b) downward acceleration*

Similarly if p_1 is the pressure intensity in the liquid at point B, then the pressure force on the bottom end of the prism is ($p_1 dA$) acting in the vertical upward direction. The self weight of the prism W which

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

is equal to ($whdA$) acts in the vertical downward direction. Since the tank is moving upwards with a constant acceleration α, the liquid mass contained in the tank is also subjected to the same acceleration.

Therefore, in addition to the above noted forces acting on the elementary prism, there is an additional force F that will be acting on the prism in the direction of the acceleration. The force F must produce the same accelerartion α as that of the tank and therefore, it is equal and opposite to the inertial force. Now if M_1 is the mass of the liquid in the prism then according to Newton's second law of motion the force F may be evaluated as

$$F = (M_1\alpha) = \left[\frac{w \times hdA}{g} \times \alpha\right]$$

Since the elementary prism of fluid also moves upward along with the tank with the acceleration α, the force F producing the acceleration α must be the resultant of the net upward pressure force ($p_1 dA - p_\alpha dA$) on the elementary prism and the downward force W, the weight of the elementary prism. Therefore,

$$(p_1 dA - p_a dA) - W = F$$

or
$$(p_1 - p_a)dA - whdA = \left[\frac{w \times hdA}{g} \times \alpha\right]$$

or
$$p_1 = p_a + wh\left(1 + \frac{\alpha}{g}\right)$$

which is same as Eq. 5.14 derived earlier.

It may be seen from Eq. 5.15 that in this case also the pressure variation is linear. But the pressure at any point in the liquid mass subjected to a constant vertical upward acceleration is greater than the hydrostatic pressure by an amount $\left(wh \times \frac{\alpha}{g}\right)$, as shown in Fig. 5.4.

If the tank containing liquid is subjected to a constant vertical acceleration acting in the downward direction then $\alpha_z = -\alpha$ and the gage pressure at any point in the liquid, lying at a depth h below the free surface of the liquid will be

$$(p - p_a) = wh\left(1 - \frac{\alpha}{g}\right) \quad \ldots (5.16)$$

In Eq. 5.16 if $\alpha = g$, that is the tank containing liquid is moving downward with a constant acceleration equal to g, then $(p - p_a) = 0$, that is the gage pressure at any point in the liquid is equal to zero. It, however, means that there will be no pressure difference between any two points in a liquid mass contained in a tank which is subjected to a vertical downward acceleration equal to g. In other words it means that the pressure throughout the liquid mass will be the same and equal to that of the surrounding atmosphere. This conclusion is important in considering a stream of water falling freely in space.

When any tank containing a liquid is subjected to a constant acceleration in any direction other than the horizontal and the vertical, then the acceleration may be resolved along the horizontal and the vertical directions and each of these cases may be separately analysed in accordance with the above given analysis.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

5.5 FLUID CONTAINERS SUBJECTED TO CONSTANT ROTATION

(a) Cylindrical Vessel Containing Liquid Rotating with its Axis Vertical. Let the cylindrical vessel partly filled with liquid be rotated at a constant angular velocity ω, about a vertical axis Z–Z as shown in Fig. 5.5. The shape of the free surface of liquid which is horizontal before rotation, will become concave, because the liquid will rise above original free surface at the edges of the vessel and it will fall down below the original free surface in the centre of the vessel. After a short time when a steady state of rotation is reached the liquid attains equilibrium condition in this position and it rotates as a solid mass with the container at the same angular velocity. The liquid is then at rest with respect to its

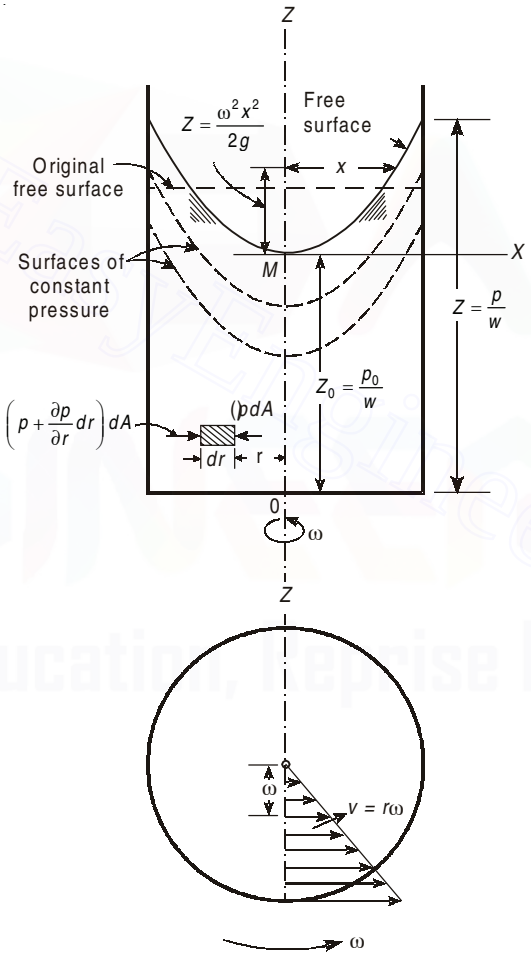


Figure 5.5 *Vessel containing liquid subjected to constant rotation with axis vertical*

container and therefore no shear stress will exist in the liquid mass. However, on account of centrifugal effects, there will be an acceleration exerted on the rotating liquid mass, acting radially inward from the outer periphery of the container towards the axis of rotation. It is therefore a centripetal acceleration,

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

on account of which the pressure will vary along the radius in the liquid mass. This variation of the pressure in the rotating liquid mass may be determined as indicated below.

Consider an elementary prism of liquid of length dr and cross-sectional area dA, at a distance r from the axis of rotation as shown in Fig. 5.5. As stated earlier the pressure varies along the radius, and therefore let p be the pressure at radius r and $[p + (\partial p/\partial r)dr]$ be the pressure at radius $(r + dr)$. Thus the pressure forces acting at the two ends of the elementary prism are

$$(pdA) \text{ and } [p + (\partial p/\partial r)dr] \times dA.$$

Further if w is the specific weight of liquid, then the mass of the elementary prism is $\left(\frac{w \times dA \times dr}{g}\right)$.

The centripetal acceleration exerted on the liquid mass at a radial distance r is $(\omega^2 r)$, acting in the radial direction towards the axis of rotation. Therefore the centripetal force exerted on the elementary prism of liquid is

$$F = \left[\left(\frac{w \times dA \times dr}{g}\right) \times (\omega^2 r)\right]$$

acting in the radial direction towards the axis of rotation and the corresponding inertial force developed will be equal to the force F, in magnitude but opposite in direction, acting radially outwards from the axis of rotation. Now considering the equilibrium of the elementary prism of liquid in the radial direction, the net pressure force exerted on the prism acting towards the axis will be equal to the inertial force acting outwards from the axis. Therefore

$$\left[p + \left(\frac{\partial p}{\partial r}\right) dr\right] dA - (pdA) = \left[\left(\frac{w \times dA \times dr}{g}\right) \times (\omega^2 r)\right]$$

or
$$\frac{\partial p}{\partial r} = \frac{w}{g}(\omega^2 r) \quad \ldots (5.17)$$

Since there is no vertical acceleration in this case, the equation representing the variation of the pressure in the vertical direction may be obtained from Eq. 5.3 as follows

$$-\frac{\partial p}{\partial z} = w \quad \ldots (5.18)$$

Equations 5.17 and 5.18 may be expressed in vector notation as

$$\text{grad } p = \frac{w}{g}\left[(\omega^2 r) r_1 - gk\right]$$

or
$$r_1 \frac{\partial p}{\partial r} + k \frac{\partial p}{\partial z} = \frac{w}{g}\left[(\omega^2 r) r_1 - gk\right]$$

where r_1 and k are the unit vectors along the radial direction and the z-axis respectively.

Pressure p at any point in the liquid in this case may be obtained as

$$p = \int dp = \int \frac{\partial p}{\partial r} dr + \int \frac{\partial p}{\partial z} dz$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$p = \frac{w}{g}\left(\frac{\omega^2 r^2}{2}\right) - wz + C$$

where C is the constant of integration, which may be evaluated for the known values of p, r and z. Thus as shown in Fig. 5.5, if the bottom of the container is considered as a reference level for measuring z, then for point M, $r = 0$, $z = z_0$ and $p = p_a$ (the atmospheric pressure) and introducing these values in the expression for p, the value of C is obtained as

$$C = p_a + wz_0$$

Thus

$$p = p_a + \frac{w}{g}\left(\frac{\omega^2 r^2}{2}\right) - w(z - z_0) \quad \ldots (5.19)$$

Equation 5.19 represents the variation of pressure in a liquid mass subjected to constant rotation. It may be seen from Eq. 5.19 that along any horizontal plane since z is constant, the pressure at any point in a rotating mass of liquid varies as the square of the distance of the point from the axis in the radial direction. The pressure is therefore maximum at the edges of the vessel where $r = R$, the radius of the vessel and it is minimum at the axis of rotation where $r = 0$, thereby indicating that in a rotating mass of liquid there is an increase in pressure in the outward radial direction from the axis of rotation towards the edges of the vessel. Further it may be seen from Eq. 5.19 that along any vertical section since r is constant, the pressure varies linearly with the distance z. It, therefore, follows that in this case the pressure distribution in the vertical direction is hydrostatic, which is evident since there is no vertical acceleration.

For any point at the free surface of the liquid since $p = p_a$, from Eq. 5.19, we get

$$(z - z_0) = \frac{\omega^2 r^2}{2g} \quad \ldots (5.20)$$

Equation 5.20 may be used to plot the free surface of liquid developed in this case. It is evident that in this case the free surface developed is a paraboloid of revolution in shape with its section being a parabola represented by Eq. 5.20. The free liquid surface developed can also be represented with respect to point M as the origin, in which case Eq. 5.20 becomes

$$z = \frac{\omega^2 x^2}{2g} \quad \ldots (5.21)$$

where x and z are the coordinates of any point on the free surface of liquid with point M as origin, measured along X and Z axes as shown in Fig. 5.5.

Since at any point on the free surface of liquid the pressure is constant, being equal to atmospheric pressure, the free surface of liquid is a surface of constant pressure, which in this case is a paraboloid of revolution, represented by Eq. 5.21. Further the pressure distribution in the liquid in the vertical direction being hydrostatic, the other surfaces of constant pressure in the liquid mass will also be the paraboloid of revolution, parallel to the free liquid surface, as shown in Fig. 5.5. Since the volume of a paraboloid of revolution is equal to half the volume of the circumscribing cylinder, it may be noted that the liquid will rise at the walls of the container by the same amount as it falls at the centre from its original level at rest.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If a cylindrical vessel completely filled with a liquid and closed at the top is rotated about its axis, then as in the case of an open vessel the pressure at any point in the horizontal plane in the liquid mass is increased by the rotation. Again in this case also along a horizontal plane the increase in the pressure is proportional to the square of the distance of the point from the axis of rotation, since Eq. 5.19 is applicable for this case as well. But in this case the free surface of liquid will not be developed as in the case of an open vessel. However, in this case an imaginary free surface exhibiting the pressure variation may be drawn in accordance with Eq. 5.19, which will also be a paraboloid of revolution. The vertex of the parabolic curve representing the imaginary free surface of liquid will coincide with the point of intersection of the axis of rotation and the confined liquid surface. However, if the vessel is filled with liquid under pressure then the parabolic curve representing the imaginary free surface can be drawn with its vertex lying on the axis above the confined liquid surface at a distance equal to the pressure head on the liquid surface before rotation commenced.

(b) Cylindrical Vessel Containing Liquid Rotating with its Axis Horizontal. Consider a closed cylindrical vessel partly filled with liquid rotated at a constant angular velocity ω, about a horizontal axis *X–X* as shown in Fig. 5.6. It is evident that in this case when a steady state of rotation is reached the liquid will rotate as a solid mass with the container at the same angular velocity and it will take the form of a cylinder with a hollow core. Let R_1 be the radius of the hollow core developed and R be the radius of the cylinder. The variation of the pressure in the rotating liquid mass may be determined in this case as shown in Fig. 5.6

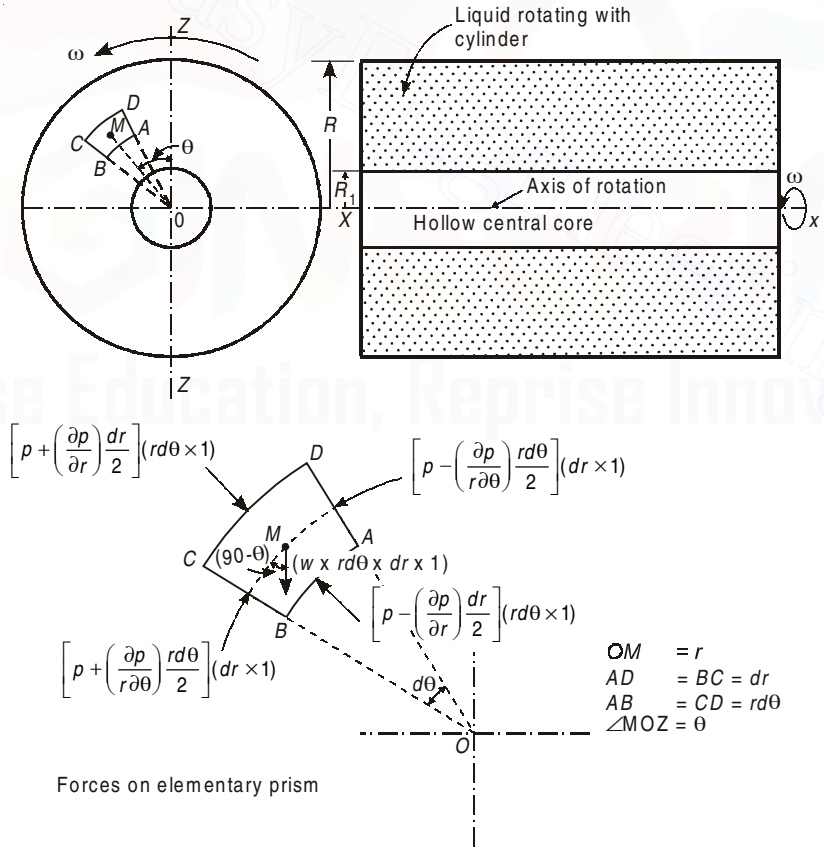


Figure 5.6 *Vessel containing liquid subjected to constant rotation with axis horizontal*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Let M be any point in the liquid mass at a distance r from the axis of rotation and at an angle θ from the vertical axis Z–Z. With M as mid-point consider an elementary prism of liquid of length dr, subtending an angle $d\theta$ at the centre of the cylinder and having unit thickness in the direction perpendicular to the paper. If p is the pressure intensity at point M, then the pressure intensities at the curved surfaces of the elementary prism AB and CD may be considered as

$$\left[p-\left(\frac{\partial p}{\partial r}\right)\frac{dr}{2}\right] \text{and} \quad \left[p+\left(\frac{\partial p}{\partial r}\right)\frac{dr}{2}\right]$$

respectively. The corresponding pressure forces on the curved surfaces AB and CD are

$$\left[p-\left(\frac{\partial p}{\partial r}\right)\frac{dr}{2}\right](rd\theta\times 1) \text{ and } \left[p+\left(\frac{\partial p}{\partial r}\right)\frac{dr}{2}\right](rd\theta\times 1)$$

respectively. Similarly the pressure intensities at the plane surfaces of the elementary prism AD and BC may be considered as

$$\left[p-\left(\frac{\partial p}{r\ \partial\theta}\right)\left(\frac{rd\theta}{2}\right)\right] \text{and} \quad \left[p+\left(\frac{\partial p}{r\partial\theta}\right)\left(\frac{rd\theta}{2}\right)\right]$$

respectively. The corresponding pressure forces on the plane surface AD and BC are

$$\left[p-\left(\frac{\partial p}{r\partial\theta}\right)\left(\frac{rd\theta}{2}\right)\right](dr\times 1) \text{ and } \left[p+\left(\frac{\partial p}{r\partial\theta}\right)\left(\frac{rd\theta}{2}\right)\right](dr\times 1)$$

respectively. Further if w is the specific weight of the liquid then the weight of the elementary prism, is $(w \times rd\,\theta \times dr \times 1)$ which is acting at M in the vertically downward direction. In addition to these forces there is centripetal force exerted on the elementary prism of liquid which is

$$F = \left[\frac{w\times rd\theta\times dr\times 1}{g}\times(\omega^2 r)\right]$$

acting in the radial direction towards the axis of rotation. The corresponding inertial force developed will be equal to the force F, in magnitude but opposite in direction, acting radially outwards from the axis of rotation. Now considering the equilibrium of the elementary prism in the radial direction, the algebraic sum of all the forces resolved in the radial direction towards the axis will be equal to the inertial force acting outwards from the axis. Therefore ignoring the components of the pressure forces on the plane surface AD and BC of the elementary prism in the radial direction, we have

$$\left[p+\left(\frac{\partial p}{\partial r}\right)\frac{dr}{2}\right](rd\theta\times 1)-\left[p-\left(\frac{\partial p}{\partial r}\right)\frac{dr}{2}\right](rd\theta\times 1)+(w\times rd\,\theta\times dr\times 1)\cos\theta$$

$$=\left[\left(\frac{w\times rd\theta\times dr\times 1}{g}\right)\times(\omega^2 r)\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{\partial p}{\partial r} = \frac{w}{g}(\omega^2 r) - w\cos\theta \quad \ldots (5.22)$$

Further considering the equilibrium of the elementary prism in the tangential direction, since there is no acceleration in the tangential direction, we have

$$\left[p + \left(\frac{\partial p}{r\partial\theta}\right)\left(\frac{rd\theta}{2}\right)\right](dr \times 1) - \left[p - \left(\frac{\partial p}{r\partial\theta}\right)\left(\frac{rd\theta}{2}\right)\right](dr \times 1) - (w \times rd\,\theta \times dr \times 1)\sin\theta = 0$$

or $$\frac{\partial p}{\partial\theta} = wr\sin\theta \quad \ldots (5.23)$$

Integrating Eq. 5.22 and 5.23, the pressure p at any point in the liquid in this case may be obtained as

$$p = \frac{w}{g}\left(\frac{\omega^2 r^2}{2}\right) - wr\cos\theta + C \quad \ldots(5.24)$$

where C is the constant of integration, which may be evaluated for the known values of p, r and θ. For example it may be assumed that the pressure of the air in the central core developed due to the rotation of the cylinder is same as the pressure of the air in the cylinder before rotation and it is equal to p_a, the atmospheric pressure. Thus the surface of the central core may be considered as a surface of constant pressure with pressure at every point on this surface being equal to the atmospheric pressure. Then since at $r = R_1$, $p = p_a$ for every value of θ, the value of C may be obtained from Eq. 5.24 as

$$C = p_a - \frac{w}{g}\left(\frac{\omega^2 R_1^2}{2}\right) + wR_1\cos\theta$$

Introducing the value of C in Eq. 5.24 it becomes

$$p = p_a\frac{w}{g}\frac{\omega^2}{2}\left(r^2 - R_1^2\right) - w\,(r - R_1)\cos\theta \quad \ldots(5.25)$$

or $$(p - p_a) = \frac{w}{g}\frac{\omega^2}{2}(r^2 - R_1^2) - w\,(r - R_1)\cos\theta \quad \ldots(5.25\,a)$$

Using Eq. 5.25 it is possible to determine the total pressure force on each end of the cylinder. Thus considering an elementary area $(rd\,\theta \times dr)$ on the flat end of the cylinder, the total pressure force in excess of the atmospheric pressure on the elementary area is

$$dF = (p - p_a)(rd\,\theta \times dr)$$

$$= \left[\frac{w}{g}\left(\frac{\omega^2}{2}\right)\left(r^2 - R_1^2\right) - w\left(r - R_1\right)\cos\theta\right](rd\theta \times dr)$$

By integrating the above expression for r varying from R_1 to R and θ varying from 0 to 2π the total pressure force on the flat end of the cylinder may be obtained as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F = \int_{R_1}^{R} \int_{0}^{2\pi} \left[\frac{w}{g}\left(\frac{\omega^2}{2}\right)\left(r^2 - R_1^2\right) - w\left(r - R_1\right)\cos\theta \right]\left(r d\theta \times dr\right)$$

Since r and θ are independent variables, the above noted double integration may be evaluated by first integrating it with respect to r and the result so obtained is then integrated with respect to θ. Thus,

$$F = \int_{0}^{2\pi} \left[\frac{w}{g}\left(\frac{\omega^2}{2}\right) \left(\frac{r^4}{4} - \frac{R_1^2 r^2}{2}\right) - w\left(\frac{r^3}{4} - \frac{R_1 r^2}{2}\right)\cos\theta \right]_{R_1}^{R} d\theta$$

or
$$F = \frac{\pi w}{g}\frac{\omega^2 (R^2 - R_1^2)^2}{4} \qquad \ldots (5.26)$$

In this case if the angular velocity ω is so great that the gravity can be neglected as compared with the centrifugal forces then Eqs 5.22 and 5.23 reduce to

$$\frac{\partial p}{\partial r} = \frac{w}{g}(\omega^2 r)$$

and
$$\frac{\partial p}{\partial \theta} = 0$$

(i.e., p does not vary with θ)

Integrating the above equations, we get

$$p = \frac{w}{g}\frac{\omega^2 r_1^2}{2} + C \qquad \ldots (5.27)$$

Again considering the surface of the central core as a surface of constant pressure with pressure at every point on this surface being equal to the atmospheric pressure, then since at $r = R_1$, $p = p_a$, the atmospheric pressure, the value of C is obtained as

$$C = p_a - \frac{w}{g}\frac{\omega^2 R_1^2}{2}$$

Introducing the value of C in Eq. 5.27, it becomes

$$p = p_a + \frac{w}{g}\frac{\omega^2}{2}(r^2 - R_1^2) \qquad \ldots(5.28)$$

or
$$(p - p_a) = \frac{w}{g}\frac{\omega^2}{2}(r^2 - R_1^2) \qquad \ldots(5.28\ a)$$

From Eq. 5.28 it may be observed that in this case the other surfaces of constant pressure are the concentric cylindrical surfaces parallel to the central core.

For determining the total pressure force on each end consider an elementary ring of width dr at a radial distance r from the axis of rotation on the flat end of the cylinder. The area of the elementary ring is $(2\pi r \times dr)$ and the total pressure force in excess of the atmospheric pressure on the elementary ring is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$dF = (p - p_a)\,(2\pi r \times dr)$$

$$= \left[\frac{w}{g}\frac{\omega^2}{2}(r^2 - R_1^2) \quad (2\pi r \times dr)\right]$$

By integrating the above expression for r varying from R_1 to R, the total pressure force on the flat end of the cylinder may be obtained as

$$F = \int_{R_1}^{R} \left[\frac{w}{g}\frac{\omega^2}{2}(r^2 - R_1^2) \quad (2\pi r \times dr)\right]$$

or
$$F = \frac{\pi w}{g}\frac{\omega^2 (R^2 - R_1^2)^2}{4}$$

which is same as Eq. 5.26 derived earlier. It may thus be noted that the total pressure force on the flat end of the cylinder has the same value whether the gravity force is considered or the same is neglected.

If the cylinder is completely filled with liquid and it is rotated with its axis horizontal then at the axis of rotation, the pressure before and after the rotation is same and it is equal to the pressure due to the column of liquid of height R. Thus for this case at $r = 0$, $p = wR$ and hence from Eq. 5.24.

$$C = wR$$

Thus
$$p = \frac{w}{g}\left(\frac{\omega^2 r^2}{2}\right) - wr\cos\theta + wR \qquad ...(5.29)$$

Again by adopting the same method as in the previous case the total pressure force on each end of the cylinder may be obtained for this case as

$$F = \frac{\pi w}{g}\frac{\omega^2 R^4}{4} + \pi w R^3 \qquad ....(5.30)$$

ILLUSTRATIVE EXAMPLES

Examples 5.1. *A rectangular tank 6 m long, 2 m deep and 2.4 m wide contains 1 m of water. If the tank moves horizontally in the direction of the length of the tank with a constant linear acceleration of 2.45 m/s², calculate (a) the angle of the water surface to the horizontal, (b) the maximum pressure intensity on the bottom, (c) the minimum pressure intensity on the bottom, (d) the total force due to the water acting on each end of the tank, and show that the difference between these forces equals the unbalanced force necessary to accelerate the liquid mass.*

Solution

(a) As shown in Fig. Ex. 5.1 if θ is the angle of the water surface to the horizontal then from Eq. 5.6.

$$\tan\theta = \frac{\text{Linear acceleration}}{\text{Gravitational acceleration}} = \frac{\alpha}{g}$$

$$\alpha = 2.45 \text{ m/s}^2 ; g = 9.81 \text{ m/s}^2$$

$$\therefore \quad \tan\theta = \frac{2.45}{9.81} = 0.250$$

and
$$\theta = 14° 2'.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

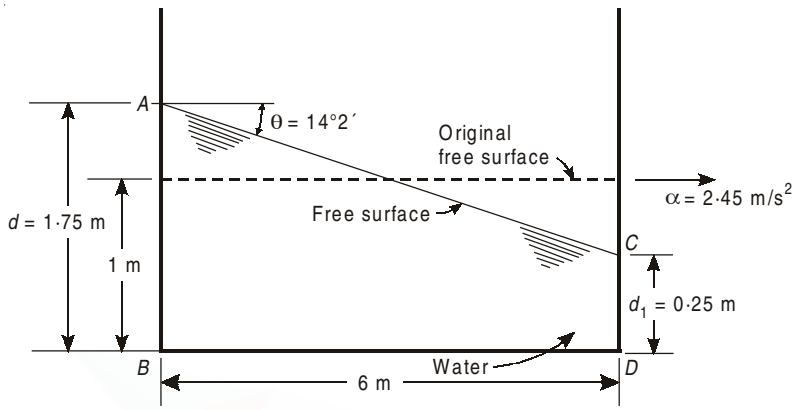


Figure Ex. 5.1

(b) From the figure the depth at deep end is

$$d = (1 + \frac{6}{2}\tan 14°2') = 1.75 \text{ m}$$

Then the maximum pressure intensity on the bottom is

$$p_{max} = (9810 \times 1.75) = 17\,167.5 \text{ N/m}^2$$

(c) The depth at the shallow end is

$$d_1 = (1 - \frac{6}{2}\tan 14° 2') = 0.25 \text{ m}$$

Then the minimum pressure intensity on the bottom is

$$p_{min} = (9\,810 \times 0.25) = 2\,452.5 \text{ N/m}^2$$

(d) The total forces acting on the two ends of the tank are

$$P_{AB} = wA\bar{x}$$

$$= 9\,810 \times (2.4 \times 1.75) \times \frac{1.75}{2}$$

$$= 36\,051.75 \text{ N}$$

$$P_{CD} = 9\,810 \times (2.4 \times 0.25) \times \left(\frac{0.25}{2}\right)$$

$$= 735.75 \text{ N}$$

$$P_{AB} - P_{CD} = 35\,316$$

Force needed = mass of water × linear acceleration

$$= \left(\frac{6 \times 2.4 \times 1 \times 9810}{9.81}\right) \times 2.45 = 35\,316 \text{ N}$$

Hence proved that the difference between the forces on the two ends is equal to the force necessary to accelerate the liquid mass.s

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 5.2. *A tank 6 m long, 2.5 wide and 2 m deep is completely filled with oil. If it is accelerated in the direction of its length at the rate of 1.5 m/s², how many litres of oil is spilled ?*

Solution

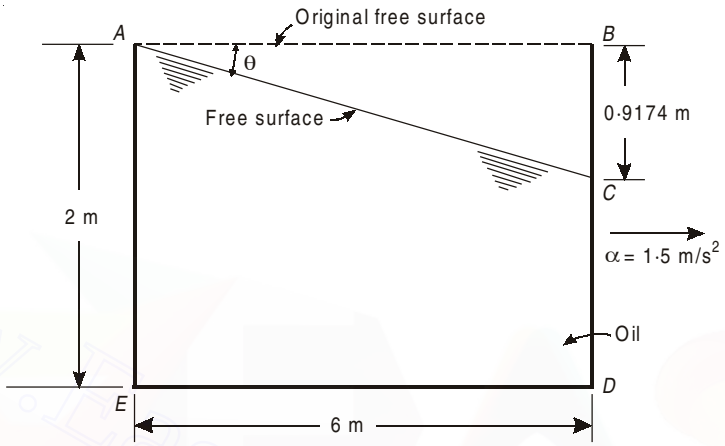


Figure Ex. 5.2

As shown in the Fig. Ex. 5.2 the slope of oil surface developed

$$= \tan\theta = \frac{\alpha}{g} = \frac{1.5}{9.81} = 0.1529$$

Drop in surface = $6 \tan\theta = 0.917\,4$ m

Volume of oil spilled = $2.5 \times$ triangular cross-section ABC

$$= 2.5 \times \frac{1}{2}(6 \times 0.9174) = 6.88 \text{ m}^3$$

$$= 6\,880 \text{ litres.}$$

Example 5.3. *A tank is 1.5 m square and contains 1 m of water. How high must its sides be if no water is to be spilled when the acceleration is 4 m/s² parallel to a pair of sides?*

Solution

$$\text{Slope of the water surface} = \tan\theta = \frac{4}{9.81} = 0.407\,7$$

$$\text{Rise (or fall) in surface} = \left(\frac{1.5}{2} \times \tan\theta\right) = 0.306 \text{ m}$$

∴ The tank must be at least

$(1.0 + 0.306) = 1.306$ m high.

Example 5.4. *An open tank of oil 5 m long contains 2 m of oil (specific gravity 0.8). If the tank accelerates up a 30° inclined plane at 3.6 m/s², what is the angle the oil surface makes with the horizontal? Also find the pressure intensities at the bottom of the vessel at the front and the rear ends.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

As shown in Fig. Ex. 5.4 the acceleration is along the direction inclined at 30° with the horizontal. As such it can be resolved along the horizontal and the vertical directions as

$$\alpha_x = (3.6 \cos 30°)$$
$$= 3.12 \text{ m/s}^2$$
$$\alpha_y = (3.6 \sin 30°)$$
$$= 1.8 \text{ m/s}^2$$

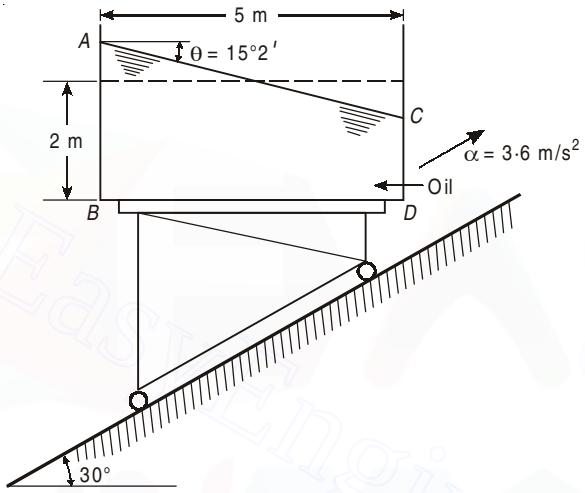


Figure. Ex. 5.4

Total acceleration in vertical direction

$$= (\alpha_y + g)$$

Hence the water surface slope is given by

$$\tan\theta = \frac{\alpha_x}{\alpha_y + g} = \frac{3.12}{(1.8 + 9.81)} = 0.268\,7$$

$$\therefore \quad \theta = 15° 2' \text{ with the horizontal}$$

The depth at the rear (deep) end is

$$h_1 = 2 + \left(\frac{5}{2}\tan 15°\ 2'\right) = 2.67 \text{ m}$$

∴ Pressure intensity at the bottom of the tank at the rear end is

$$p_{AB} = wh_1\left(1 + \frac{\alpha_y}{g}\right)$$

$$= (9\,810 \times 0.8) \times 2.67\left(1 + \frac{1.8}{9.81}\right)$$

$$= 24\,800 \text{ N/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Similarly the depth of oil at the front (shallow) end is

$$h_2 = 2 - \left(\frac{5}{2}\tan 15° 2'\right) = 1.33 \text{ m}$$

$\therefore$ pressure intensity at the bottom of the tank at the front end is

$$p_{CD} = wh_2\left(1 + \frac{\alpha_y}{g}\right)$$

$$= (9\,810 \times 0.8) \times 1.33\left(1 + \frac{1.8}{9.81}\right)$$

$$= 12\,353 \text{ N/m}^2$$

Example 5.5. *A rectangular tank 1.5 m wide, 3 m long, and 1.8 m deep contains water to a depth of 1.2 m. Find the horizontal acceleration which may be imparted to the tank in the direction of its length so that (a) there is just no spilling of water from the tank, (b) the front bottom corner of the tank is just exposed, (c) the bottom of the tank is exposed upto its mid-point. Calculate the volume of water that would spill out from the tank in the cases (b) and (c).*

Also calculate the total forces on each end of the tank in each of these cases and show that the difference between these forces equals the unbalanced force necessary to accelerate the liquid mass in the tank.

Solution

(a) When there is just no spilling of water then as shown in the Fig. Ex. 5.5 (a)

$$\tan\theta = \frac{0.6}{1.5} = 0.4$$

But from Eq. 5.6

$$\tan\theta = \frac{\alpha}{g}$$

$\therefore$ $$\frac{\alpha}{g} = 0.4$$

$$\alpha = 0.4 \times 9.81 = 3.924 \text{ m/s}^2$$

The total force acting on the end *AB* of the tank is

$$P_1 = 9\,810 \times (1.8 \times 1.5) \times \left(\frac{1.8}{2}\right) = 23\,838.3 \text{ N}$$

The total force acting on the end *CD* of the tank is

$$P_2 = 9\,810 \times (0.6 \times 1.5) \times \left(\frac{0.6}{2}\right) = 2648.7 \text{ N}$$

$$P_1 - P_2 = (23\,838.3 - 2648.7) = 21\,189.6 \text{ N}$$

The force needed to accelerate the liquid mass in the tank is

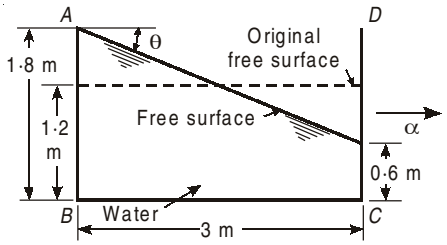
$$F = \text{mass of water} \times \text{linear acceleration}$$

$$= \left(\frac{3 \times 1.2 \times 1.5 \times 9\,810}{9.81} \times 3.924\right) = 21\,189.6 \text{ N}$$

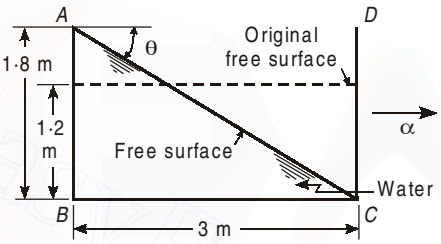
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

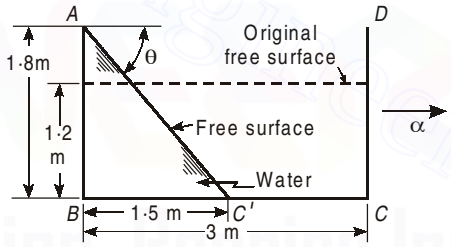
Hence proved that the difference between the forces on the two ends of the tank is equal to the force necessary to accelerate the liquid mass in the tank.



(a) Just no spilling of water from the tank



(b) Corner *C* of the tank just exposed



(c) Half of the tank bottom exposed

Figure Ex. 5.5

(b) Since the tank is open, maximum level of water near the rear end of the tank can be upto the top edge of the tank *A*. Thus when the front bottom corner of the tank is to be just exposed then the free surface of water in the tank will be along *AC* as shown in the Fig. Ex. 5.5 (b). Hence

$$\tan\theta = \frac{1.8}{3} = 0.6$$

But from Eq. 5.6

$$\tan\theta = \frac{\alpha}{g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad \frac{\alpha}{g} = 0.6$$

or $$\alpha = 0.6 \times 9.81 = 5.886 \text{ m/s}^2$$

The volume of water remaining in the tank after acceleration is imparted to it

$$= \frac{1}{2} \times (1.8 \times 3) \times 1.5 = 4.05 \text{ m}^3$$

The volume of water in the tank before acceleration is imparted to it

$$= (3 \times 1.5 \times 1.2) = 5.40 \text{ m}^3$$

∴ The volume of water spilled out from the tank

$$= (5.40 - 4.05) = 1.35 \text{ m}^3$$

The total force acting on the end *AB* of the tank is

$$P_1 = 9\,810 \times (1.8 \times 1.5) \times \left(\frac{1.8}{2}\right) = 23\,838.3 \text{ N}$$

In this case since there is no water against the end *CD* of the tank, the total force acting on this end is

$$P_2 = 0$$

$$\therefore \quad (P_1 - P_2) = 23\,838.3 \text{ N}$$

The force needed to accelerate the liquid mass in the tank is

$$F = \text{mass of water} \times \text{linear acceleration}$$

$$= \left(\frac{4.05 \times 9810}{9.81} \times 5.886\right) = 23838.3 \text{ N}$$

Hence proved that the difference between the forces on the two ends of the tank is equal to the force necessary to accelerate the liquid mass in the tank.

(c) In this case the free surface of the water in the tank will be along *AC′* as shown Fig. Ex. 5.5 (c).

Thus

$$\tan \theta = \frac{1.8}{1.5} = 1.2$$

But from Eq. 5.6

$$\tan \theta = \frac{\alpha}{g}$$

$$\therefore \quad \frac{\alpha}{g} = 1.2$$

or $$\alpha = 1.2 \times 9.81 = 11.772 \text{ m/s}^2$$

The volume of water remaining in the tank after acceleration is imparted to it

$$= \frac{1}{2} \times (1.8 \times 1.5) \times 1.5 = 2.025 \text{ m}^3$$

∴ The volume of water spilled out from the tank

$$= (5.40 - 2.025) = 3.375 \text{ m}^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again in this case also the difference between the forces acting on the ends *AB* and *CD* of the tank is equal to 23 838.3 N.

The force needed to accelerate the liquid mass in the tank is

$$F = \text{mass of water} \times \text{linear acceleration}$$

$$= \left(\frac{2.025 \times 9\ 810}{9.81} \times 11.772\right) = 23\ 838.3 \text{ N}$$

Hence proved that the difference between the forces on the two ends of the tank is equal to the force necessary to accelerate the liquid mass in this tank.

Example 5.6. *A closed rectangular tank 1.2 m high, 2.4 m long and 1.5 m wide is two-thirds full of gasoline (sp. gr. 0.8). Calculate the acceleration which may be imparted to the tank so that the bottom front end of the tank is just exposed. Also calculate the total forces on each end of the tank and show that the difference between these forces equals the unbalanced force necessary to accelerate the liquid mass in the tank.*

Solution

The depth of gasoline in the tank

$$= 1.2 \times \frac{2}{3} = 0.8 \text{ m}$$

In this case since the tank is closed the liquid cannot spill out from the tank under any acceleration imparted to it. As such for any acceleration imparted to the tank the volume of the liquid in the tank would be same as it was before the acceleration was imparted to the tank.

When the bottom front end of the tank is to be just exposed then the free surface of gasoline in the tank will be along *CE* as shown in the Fig. 5.6

Let the distance *AE* be *x* then by applying the above noted condition we have

$$\left[\frac{1}{2}(x + 2.4) \times 1.2 \times 1.5\right] = (2.4 \times 1.5 \times 0.8)$$

$$\therefore \quad x = 0.8 \text{ m}$$

Thus

$$\tan\theta = \frac{1.2}{(2.4 - 0.8)}$$

$$= 0.75$$

But from Eq. 5.6

$$\tan\theta = \frac{\alpha}{g}$$

$$\therefore \quad \frac{\alpha}{g} = 0.75$$

or $$\alpha = 7.357\ 5 \text{ m/s}^2$$

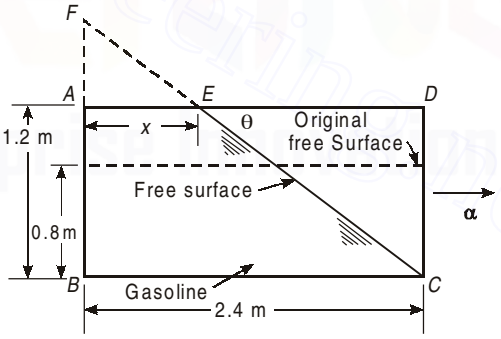


Figure Ex. 5.6

As shown in the figure, if the free surface *CE* when extended meets *AB* produced at *F*, then the pressure at *F* is same as that on the free surface of gasoline, which in this case is equal to the atmospheric pressure. Thus at *A* the pressure head is equal to an imaginary column of gasoline of height equal to *AF*.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $\frac{AF}{AE} = \tan\theta = 0.75$

$\therefore \quad AF = 0.75 \times (AE)$

$= 0.75 \times 0.8 = 0.6$ m

Thus the total force acting on the end *AB* of the tank is

$$P_1 = 9\,810 \times 0.8 \times \left(\frac{0.6+1.8}{2}\right) \times (1.2 \times 1.5)$$

$$= 16\,951.68 \text{ N}$$

In this case since there is no gasoline against the end *CD* of the tank, the total force acting on this end is

$$P_2 = 0$$

$\therefore \quad (P_1 - P_2) = 16\,951.68$ N

The force needed to accelerate the liquid mass in the tank is

F = mass of gasoline ∞ linear acceleration

$$= \left(\frac{9810 \;\; 0.8 \times 2.4 \times 1.5 \times 0.8}{9.81}\right) \times 7.357\,5$$

$$= 16\,951.68 \text{ N}$$

Hence proved that the difference between the forces on the two ends of the tank is equal to the force necessary to accelerate the liquid mass in the tank.

Example 5.7. *Calculate the acceleration which must be imparted to the tank of Ex. 5.6 in order that 0.6 m of the bottom of the tank from its front end is exposed. Also calculate the total pressure on the rear end of the tank. Hence show that in order to expose 0.8 m of the bottom of the tank the acceleration required to be imparted to the tank is infinite.*

Solution

Refer to the Fig. Ex. 5.6. The distance x may be obtained by applying the condition that the volume of the liquid in the tank would be same as it was before the acceleration was imparted to the tank, and hence

$$\left[\frac{1}{2}(x+1.8) \times 1.2 \times 1.5\right] = (2.4 \times 1.5 \times 0.8)$$

$$x = 1.4 \text{ m}$$

Thus $\tan\theta = \frac{1.2}{(2.4 - 1.4 - 0.6)} = 3$

But from Eq. 5.6

$$\tan\theta = \frac{\alpha}{g}$$

$\therefore \quad \frac{\alpha}{g} = 3$; or $\alpha = 29.43 \text{ m/s}^2$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further $\frac{AF}{AE} = \tan\theta = 3$

$\therefore \quad AF = 3 \times (AE) = 3x$

$= 3 \times 1.4 = 4.2$ m

Thus the total pressure acting on the rear end of the tank is

$$P_1 = 9\,810 \times 0.8 \times \left(\frac{4.2 + 5.4}{2}\right) \times (1.2 \times 1.5)$$

$$= 67\,806.72 \text{ N}$$

When 0.8 m of the bottom is exposed then by applying the above noted condition, we have

$$\left[\frac{1}{2}(x + 1.6) \times 1.2 \times 1.5\right] = (2.4 \times 1.5 \times 0.8)$$

$\therefore \quad x = 1.6$ m

Thus $\tan\theta = \frac{1.2}{(2.4 - 1.6 - 0.8)} = \infty$

But $\tan\theta = \frac{\alpha}{g}$

$\therefore \quad \alpha = \infty$

i.e., the acceleration required to be imparted to the tank to expose 0.8 m of its bottom is infinite.

Example 5.8. *A closed cylindrical tank of diameter 1.2 m and length 6 m is completely filled with oil of specific gravity 0.9. If the tank is placed with its axis horizontal and subjected to an acceleration of 2.5 m/s² in the horizontal direction, find the forces on the two ends of the tank, when (a) the oil is filled at atmospheric pressure, (b) the oil is filled under a pressure of 17.658 kN/m². Show that the difference between these forces equals the unbalanced force necessary to accelerate the liquid mass in the tank.*

Solution

(a) As shown in Fig. Ex. 5.8 (a) the slope of the imaginary free surface developed in this case is given by Eq. 5.6 as

$$\tan\theta = \frac{\alpha}{g} = \frac{2.5}{9.81} = 0.254\,842$$

Thus pressure head at A is equal to

$AE = 6 \tan\theta$

$= (6 \times 0.254\,842) = 1.529\,05$

$\therefore$ The total force acting on the end AB of the tank is

$$P_1 = 9810 \times 0.9 \times \left(\frac{1.529\,05 + 2.729\,05}{2}\right) \times \frac{\pi}{4} \times (1.2)^2$$

$$= 21\,259.34 \text{ N}$$

Similarly the total force acting on the end CD of the tank is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$P_2 = 9810 \times 0.9 \times \frac{\pi}{4} \times (1.2)^2\ 0.6$$

$$= 5\ 991.22 \text{ N}$$

$$\therefore \quad P_1 - P_2 = (21\ 259.34 - 5\ 991.22) = 15\ 268.12 \text{ N}$$

The force needed to accelerate the liquid mass in the tank is

$$F = \text{mass of oil} \times \text{linear acceleration}$$

$$= \left[\frac{\pi}{4} \times (1.2)^2 \times 6 \times \frac{9\,810 \times 0.9}{9.81}\right] \times 2.5$$

$$= 15\ 268.14 \text{ N}$$

Hence proved that the difference between the forces on the two ends of the tank is equal to the force necessary to accelerate the liquid mass in the tank.

(b) As shown in Fig. Ex. 5.8 (b), in this case also the slope of the imaginary free surface will be same as in the previous case, but it will be shifted above the top of the tank by a distance equal to the initial pressure head of the oil which is

$$= \left(\frac{17.658 \times 10^3}{9810 \times 0.9}\right) = 2 \text{ m}$$

Thus the total force acting on the end *AB* of the tank is

$$P_1 = 9\,810 \times 0.9 \times \left(\frac{3.52905 + 4.72905}{2}\right) \times \frac{\pi}{4} \times (1.2)^2$$

$$= 41\ 230.06 \text{ N}$$

Similarly the total force acting on the end *CD* of the tank is

$$P_2 = 9\,810 \times 0.9 \times \frac{2.0 + 3.2}{2} \times \frac{\pi}{4}(1.2)^2$$

$$= 25\ 961.94 \text{ N}$$

$$\therefore \quad (P_1 - P_2) = (41\ 230.06 - 25\ 961.94)$$

$$= 15\ 268.12 \text{ N}$$

Further in this case since the mass of the oil in the tank as well as the acceleration imparted to the tank are same as in the previous case, the force needed to accelerate the liquid mass in the tank would also be the same as in the previous case, which is therefore equal to 15 268.14 N.

Hence it is proved that the difference between the forces on the two ends of the tank is equal to the force necessary to accelerate the liquid mass in the tank.

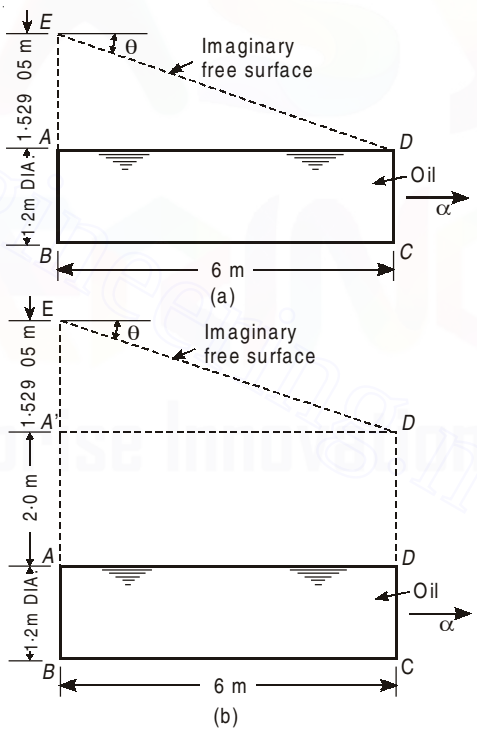


Figure Ex. 5.8

Example 5.9. *A cubical tank of side 2 m is filled with 1.5 m of glycerine of specific gravity 1.6. Find the force acting on the side of the tank when (a) it is accelerated vertically upward at 5 m/s², and (b) it is accelerated vertically downward at 5 m/s².*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Figure Ex. 5.9 shows the pressure distribution on a vertical side *AB*.

(a) When the tank is accelerated upwards then the pressure intensity at *B* is

$$p_B = wh\left(1+\frac{\alpha}{g}\right)$$

$$= (9\,810 \times 1.6) \times 1.5\left(1+\frac{5}{9.81}\right)$$

$$= 35\,544 \text{ N/m}^2$$

$\therefore$ Force P_{AB} = (area of pressure diagram *ABC*) $\times$ 2

$$= \left(\frac{1}{2} \times 35\,544 \times 1.5\right) \times 2 = 53\,316 \text{ N}$$

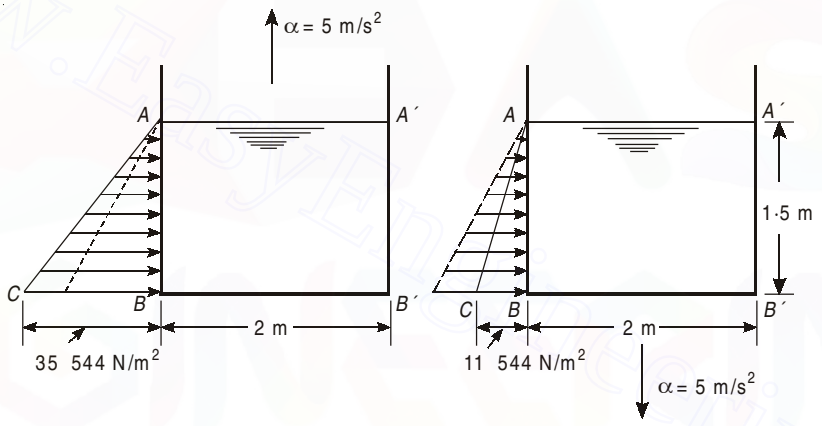


Figure Ex. 5.9

(b) When the tank is accelerated downwards then the pressure intensity at *B* is

$$p_B = wh\left(1-\frac{\alpha}{g}\right)$$

$$= (9\,810 \times 1.6) \times 1.5\left(1-\frac{5}{9.81}\right)$$

$$= 11\,544 \text{ N/m}^2$$

$\therefore$ Force P_{AB} = (area of pressure diagram *ABC*) $\times$ 2

$$= \left(\frac{1}{2} \times 11\,544 \times 1.5\right) \times 2 = 17\,316 \text{ N}$$

Example 5.10. *An open cylindrical tank 2 m high and 1 m diameter, contains 1.5 m of water. If the cylinder rotates about its geometric axis, (a) what constant angular velocity can be attained without spilling any water? (b) What is the pressure intensity at the bottom of the tank, at the centre and at the walls when w = 6.0 rad/s?*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

(a) As shown in the Fig. Ex. 5.10 if no water is spilled, the volume of the paraboloid of revolution *BOB* equals the volume above the original water level *A–A.*

Volume of paraboloid of revolution

$$= \frac{1}{2} \text{ (Volume of circumscribing cylinder)}$$

$$= \frac{1}{2}\left[\frac{\pi}{4}(1)^2(0.5+z_1)\right]$$

$$\therefore \quad \frac{1}{2}\left[\frac{\pi}{4}(1)^2(0.5+z_1)\right] = \frac{\pi}{4}(1)^2 \times 0.5$$

or $z_1 = 0.5$ m

From Eq. 5.21 for any point on the paraboloid of revolution

$$z = \frac{\omega^2 x^2}{2g}$$

For point *B* the coordinates *x* and *z* with *O* as origin are $x = 0.5$ m, $z = (0.5 + 0.5) = 1.0$ m. Then by substitution

$$1.0 = \frac{\omega^2 \times (0.5)^2}{2 \times 9.81}$$

or $$\omega = \sqrt{\frac{2 \times 9.81}{(0.5^2)}}$$

$$= 8.86 \text{ rad/s}$$

Since $\omega = \frac{2\pi N}{60}$, the corresponding speed in r.p.m. is

$$N = \frac{8.86 \times 60}{2\pi} = 84.61$$

$$\approx 85 \text{ r.p.m.}$$

(b) For $\omega = 6$ rad/s

$$z = \frac{\omega^2}{2g} x^2$$

$$= \frac{(6.0)^2}{2 \times 9.81} \times (0.5)^2$$

$$= 0.46 \text{ m from } O$$

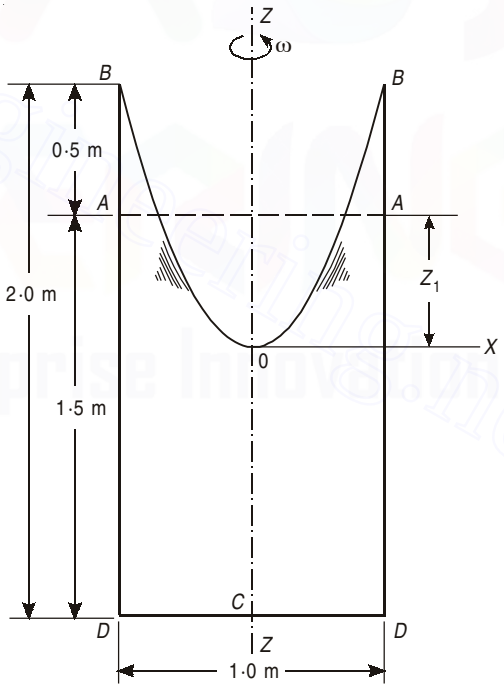


Figure Ex. 5.10

The origin *O* drops $\frac{1}{2} z = 0.23$ m below the original water level *AA* and hence point *O* is now (1.5 – 0.23)

$= 1.27$ m above the bottom of the tank.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

∴ Pressure intensity at the centre of the bottom of the tank is

$$p_C = (9\,810 \times 1.27) = 12\,458.7 \text{ N/m}^2$$

Similarly at the walls the depth of water will be

$$(1.5 + 0.23) = 1.73 \text{ m}$$

∴ Pressure intensity at the bottom of the tank near the wall is

$$p_D = (9\,810 \times 1.73) = 16\,971.3 \text{ N/m}^2$$

Example 5.11. *A cylindrical tank 0.6 m high and 0.5 m diameter is filled completely with water. If it is rotated at a speed of 120 r.p.m., how many litres of water will be spilled ? Also determine the speeds at which (i) the water surface will just touch the top rim and the centre bottom of the tank (ii) the water surface will just touch the top and the bottom of the tank has uncovered circular area of diameter 0.2 m.*

Solution

At 120 r. p.m., $\omega = \dfrac{2\pi \times 120}{60} = 12.57$ rad/s. From Eq. 5.21 for any point on the free surface developed

$$z = \frac{\omega^2}{2g} x^2$$

For a point at the rim of the tank,

$$x = 0.25 \text{ m}$$

$$\therefore \quad z = \frac{(12.57)^2 \times (0.25)^2}{2 \times 9.81} = 0.503 \text{ m}$$

$$\text{Volume of water spilled} = \text{Volume of paraboloid}$$

$$= \frac{1}{2}\left[\frac{\pi}{4}(0.5)^2 \times 0.503\right]$$

$$= 0.049 \text{ m}^3 = 49 \text{ litres}$$

(i) When the water surface just touches the top rim and the base, then

$$z = 0.6 \text{ m and } x = 0.25 \text{ m}$$

So

$$\omega = \sqrt{\frac{2gz}{x^2}}$$

$$= \sqrt{\frac{2 \times 9.81 \times 0.6}{(0.25)^2}} = 13.72 \text{ rad/s}$$

and

$$N = \frac{60 \times 13.72}{2\pi} = 131 \text{ r.p.m.}$$

(ii) When the water surface just touches the top of the rim and the bottom of the tank has uncovered circular area of diameter 0.2 m, then let the origin of the parabolic water surface be at a depth z_1 below the bottom of the tank. Thus for a point at the bottom of the tank.

$$z = z_1 \text{ and } x = 0.1 \text{ m}$$

∴ From Eq. 5.21

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$z_1 = \frac{\omega^2}{2g}(0.1)^2 \quad \ldots (1)$$

Further for a point at the rim of the tank,

$$z = (z_1 + 0.6)$$

and $$x = 0.25 \text{ m}$$

Thus from Eq. 5.21

$$(z_1 + 0.6) = \frac{\omega^2}{2g}(0.25)^2 \quad \ldots (2)$$

Solving Eqs (1) and (2)

$$\omega = 14.97 \text{ rad/s}$$

and $$N = \frac{60 \times 14.97}{2\pi} = 142.95 \text{ r.p.m.}$$

Example 5.12. *A closed tank 2 m high and 1 m in diameter contains 1.5 m water. The air in the space above water is subjected to a pressure of 117.72 kN/m². If the tank is rotated at an angular velocity of 12.0 rad/s, what are the pressure intensities at the centre of the bottom and near the walls at the bottom of the tank?*

Also find the speed at which this tank should be rotated in order that the centre of the bottom has zero depth of water.

Solution

Since there is no change in the volume of air within the vessel, volume above AA = volume of paraboloid. Thus as shown in Fig. Ex. 5.12

$$\frac{\pi}{4}(1)^2 \times 0.5 = \frac{1}{2}\pi x_2^2 z_2 \quad \ldots (1)$$

Also from Eq. 5.21

$$z_2 = \frac{(12.0)^2}{2 \times 9.81} x_2^2 \quad \ldots (2)$$

Solving Eq. (1) and (2) simultaneously

$$x_2 = 0.43 \text{ m}; z_2 = 1.36 \text{ m}$$

From the figure the origin is located (2.00 – 1.36) = 0.64 m above the centre of the base.

Then the pressure at the centre of base is

$$p_C = \left[117.72 + \frac{9 \quad 810 \times 0.64}{1000}\right]$$

$$= 124 \text{ kN/m}^2$$

To evaluate the pressure intensity at the bottom near the walls of the tank the pressure head

$$z_1 = \frac{(12.0)^2}{2 \times 9.81} \times (0.5)^2 = 1.84 \text{ m above } O$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$p_D = \left[117.72 + \frac{9 \quad 810(1.84 + 0.64)}{1000}\right]$$

$$= 142 \text{ kN/m}^2$$

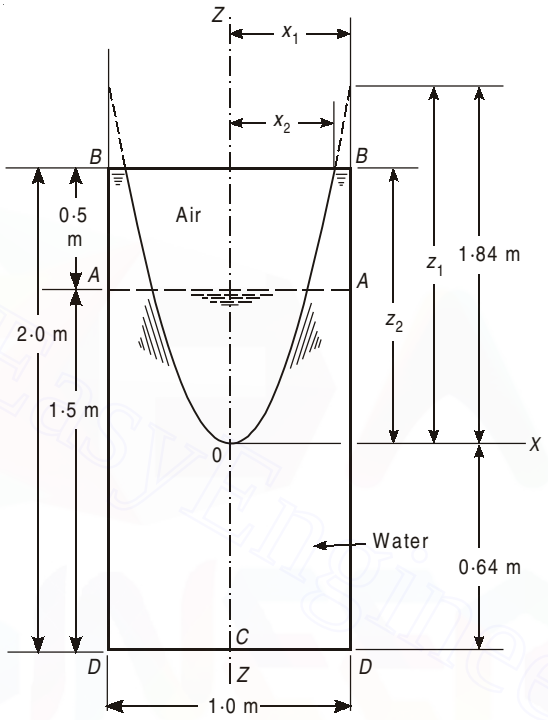


Figure Ex. 5.12

For zero depth of water at the centre of the bottom, the origin *O* of the parabola will be at point *C* shown in the figure.

Volume above liquid surface

= Volume of paraboloid

So $$\frac{\pi}{4}(1)^2 \times 0.5 = \frac{1}{2}\pi x_2^2 \times (2.00) \quad \text{... (1)}$$

Also $$z_2 = 2.00 = \frac{\omega^2}{2 \times 9.81} \times x_2^2 \quad \text{... (2)}$$

Solving Eqs (1) and (2) simultaneously

$$\omega^2 = (32 \times 9.81) = 313.92$$

$$\omega = \sqrt{313 \cdot 92} = 17.72 \text{ rad/s}$$

and $$N = 169.2 \text{ r.p.m.}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 5.13. *A closed cylindrical tank 2 m high and 1 m in diameter contains 1.5 m of water. When the angular velocity is constant at 20.0 rad/s, how much of the bottom of the tank is uncovered ?*

Solution

As shown inFig. Ex. 5.13

$$z_3 = \frac{(20)^2}{2\times 9.81} \times (0.5)^2$$

$$= 5.097 \text{ m}$$

Let the origin O be at a depth of z_1 below the base of the tank then

$$z_1 = \frac{(20)^2}{2\times 9.81} \times x_1^2 \quad \ldots (1)$$

Also

$$z_2 = (2 + z_1) = \frac{(20)^2}{2\times 9.81} \times x_2^2 \quad \ldots (2)$$

Further, since the volume of the air is constant

$$\frac{\pi}{4}(1)^2 \times 0.5 = \text{volume of (paraboloid } OAB - \text{paraboloid } OCD)$$

or

$$\frac{\pi}{4}(1)^2 \times 0.5$$

$$= \left(\frac{1}{2}\pi x_2^2 z_2 - \frac{1}{2}\pi x_1^2 z_1\right) \quad \ldots (3)$$

Solving the Eqs (1), (2) and (3) simultaneously

$$x_1^2 = 0.0135 \text{ and } x_1 = 0.116 \text{ m}$$

Hence the area uncovered

$$= \left(\pi x_1^2\right) = \pi \times (0.116)^2$$

$$= 0.042 \text{ m}^2.$$

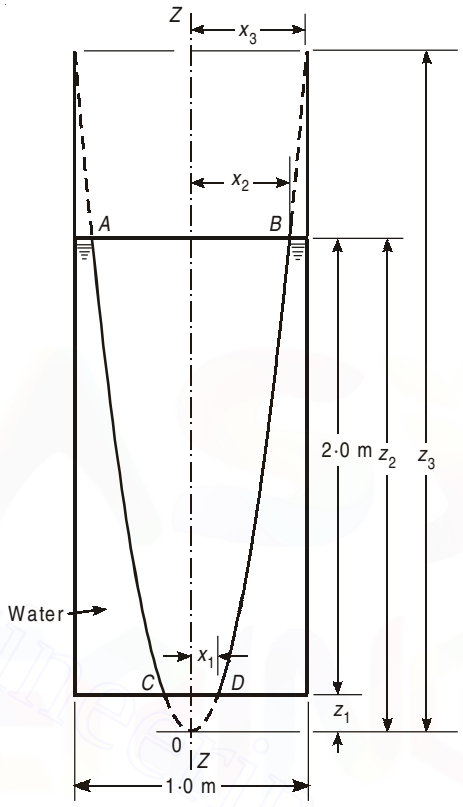


Figure Ex. 5.13

Example 5.14. *A 2 m diameter cylinder 3 m high is completely filled with glycerine of specific gravity 1.60 under a pressure of 245.25 kN/m² at the top. What maximum speed in r.p.m. can be imposed on the cylinder so that the pressure at the bottom may not exceed 1 177.2 kN/m².*

Solution

The pressure intensity at the bottom of the tank is

p = [245.25 kN/m² imposed + 3 m of glycerine + due to rotation]

$$= \left[245.25 + \frac{3\times 9\,810\times 1.6}{1000} + \frac{\omega^2}{2\times 9.81}\times(1)^2 \times \frac{1.6\times 9\,810}{1000}\right] \text{kN/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \left[245.25 + 47.09 + 0.8\omega^2\right] \text{kN/m}^2$$

But the limiting value of p = 1 177.2 kN/m^2

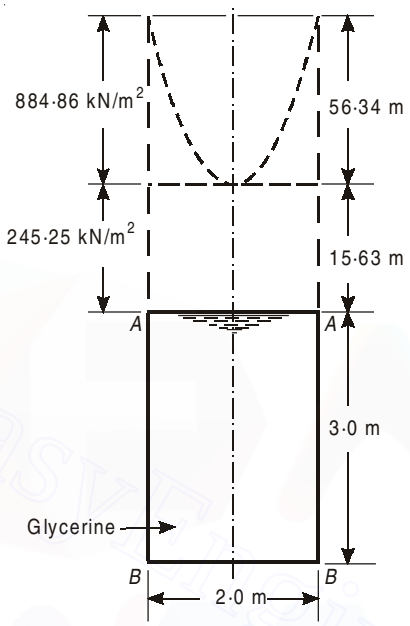


Figure Ex. 5.14

Thus $1177.2 = (245.25 + 47.09 + 0.8\ \omega^2)$

Solving

$$\omega = 33.26 \text{ rad/s}$$

and $N = 317.61$ r.p.m

Figure Ex. 5.14 shows the pressure distribution in this case.

Example 5.15. *A U-tube having its vertical legs 0.6 m apart is partly filled with carbon tetrachloride (specific gravity = 1.6) and rotated about a vertical axis 0.15 m from one leg. What will be the difference in elevation of the two free surfaces when the angular velocity is 100 revolutions per minute?*

Solution

The difference in elevation of the free surfaces in the two legs is given by

$$(z_2 - z_1) = \frac{\omega^2}{2g}(x_2^2 - x_1^2)$$

$$\omega = \frac{2\pi \times 100}{60}$$

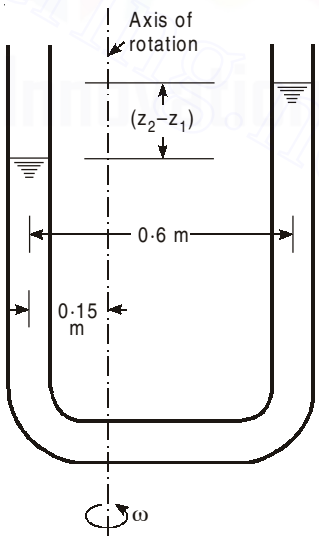


Figure Ex. 5.15

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= 10.47$ rad/s;

$x_2 = 0.45$ m;

$x_1 = 0.15$ m

$$\therefore \quad (z_2 - z_1) = \frac{(10.47)^2}{2 \times 9.81} \times [(0.45)^2 - (0.15)^2]$$

$$= 1.006 \text{ m.}$$

Example 5.16. *An open tank 10 m long and 2 m deep is filled with 1.5 m of oil of sp. gr. 0.82. The tank is accelerated uniformly from rest to a speed of 20 m/s. What is the shortest time in which this speed may be attained without spilling any oil?*

Solution

Let t be the required time, then the rate of increase of velocity of tank (or acceleration)

$$\alpha = \frac{20}{t} \text{ m/s}^2$$

Further for no spilling of oil the maximum slope of the oil surface may be

$$\tan \theta = \frac{0.5}{5} = 0.1$$

But $$\tan \theta = \frac{\alpha}{g} = \frac{20}{t \times 9.81}$$

Thus equating the two, we get $0.1 = \dfrac{20}{t \times 9.81}$

$$\therefore \quad t = 20.387 \text{ s}$$

Example 5.17. *A closed cylindrical vessel of diameter 0.65 m is half filled with water and then rotated with its axis horizontal at a speed of 600 r.p.m. If the water rotates as a forced vortex at the same angular velocity as the vessel and takes the form of a cylinder with a hollow core, what would be the total axial thrust tending to burst the flat ends?*

Also determine the total axial thrust on the flat ends if the cylinder is completely filled with water and rotated with its axis horizontal at the same speed.

Solution

When the vessel is half filled with water, the air above the water in the cylinder may be assumed to be at atmospheric pressure. Further it is assumed that the pressure and the volume of the air in the cylinder before and after the rotation remains same. Thus if L is the length of the cylinder and R_1 is the radius of the hollow core developed when it rotates, then

$$\pi R_1^2 \times L = \frac{1}{2}\frac{\pi}{4} \times (0.65)^2 \times L$$

$$\therefore \quad R_1 = 0.23 \text{ m}$$

The total axial thrust tending to burst each end of the cylinder is given by Eq. 5.26 as

$$F = \frac{\pi w}{g} \frac{\omega^2 (R^2 - R_1^2)^2}{4}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substituting the given values, we get

$$F = \frac{\pi \times 9\ 810}{9.81} \times \left(\frac{2\pi \times 600}{60}\right)^2 \times \frac{1}{4}\ [(0.325)^2 - (0.23)^2]^2$$

$$= 8\ 619.5 \text{ N}$$

When the vessel is completely filled with water and it is rotated with its axis horizontal then the total axial thrust on each end of the vessel is given by Eq. 5.30 as

$$F = \frac{\pi w}{g}\frac{\omega^2 R^4}{4} + \pi w R^3$$

By substituting the given values, we get

$$F = \left[\frac{\pi \times 9810}{9.81} \times \left(\frac{2\pi \times 600}{60}\right)^2 \times \frac{(0.325)^4}{4}\right] + \left[\pi \times 9\ 810 \times (0.325)^3\right]$$

$$= 35\ 650.54 \text{ N}$$

Example 5.18. *A conical vessel with base uppermost is rotated about its axis which is vertical. The vessel was filled completely with water when at rest. After rotating the vessel at 60 r.p.m, only 0.0142 m^3 of water remained in it. Calculate the ratio of the radius of the base to its height.*

Solution

Let R be the radius of the base of the cone and H be the height of the cone.

As shown in Fig. Ex. 5.18 if z is the total depth of the paraboloid of revolution developed, then from Eq. 5.21, we have

$$z = \frac{\omega^2 R^2}{2g}$$

Let AO' be the tangent to the parabolic water surface at A. From Eq. 5.21 the slope of the tangent AO' at A is obtained as

$$\frac{dz}{dx} = \frac{\omega^2 R}{g} = \frac{2z}{R}$$

Now if $OO' = z'$ then

$$\frac{z + z'}{R} = \tan\theta_1 = \frac{dz}{dx} = \frac{2z}{R}$$

$$\therefore \quad z' = z$$

i.e., the tangent AO' at A meets the axis of rotation at a point O' which is at a distance z below the point O as shown in Fig. Ex. 5.18.

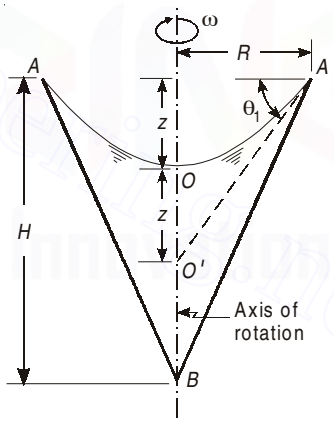


Figure Ex. 5.18

The volume of water in the cone before rotation $= \frac{1}{3}\ \pi R^2 H$

The volume of the paraboloid of revolution

$$AOA = \frac{1}{2}\pi R^2 z$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Now if V is the volume of the water that remains in the cone after rotation then

$$\frac{1}{3}\pi R^2 H - \frac{1}{2}\pi R^2 z = V$$

In general it may be considered that $z = kH$, where k is a factor, the value of which depends on the speed of rotation. Introducing the value of z in the above expression, we have

$$kH = \left(\frac{\omega^2 R^2}{2g}\right) \quad \text{... (i)}$$

and $$\frac{1}{3}\pi R^2 H - \frac{1}{2}\pi R^2 kH = V \quad \text{... (ii)}$$

From Eq. (i) and (ii), we obtain

$$H = \left[\frac{V\omega^2}{2gk\pi\left(\frac{1}{3}-\frac{k}{2}\right)}\right]^{1/2}$$

and $$R = \left[\frac{2gkV}{\pi\omega^2\left(\frac{1}{3}-\frac{k}{2}\right)}\right]^{1/4}$$

$$\therefore \quad \frac{R}{H} = \left[\left(\frac{2gk}{\omega^2}\right)^3 \times \left(\frac{1}{3}-\frac{k}{2}\right) \times \frac{\pi}{V}\right]^{1/4} \quad \text{... (iii)}$$

The above expression gives the general value of the ratio of the radius of the base and the height of a conical vessel which when rotated about its vertical axis at an angular velocity ω has V volume of water remaining in it.

In the present case it is given that

$$V = 0.01\,42 \text{ m}^3; \text{ N} = 60 \text{ r.p.m}$$

$$\therefore \quad \omega = \frac{2\pi N}{60} = \frac{2\pi \times 60}{60} = 2\pi \text{ rad/s}.$$

Further in the present case it may be assumed that corresponding to the given speed of rotation the slanting surface AB of the cone is tangential to the parabolic water surface at A, in which case $z = (H/2)$ and hence $k = (1/2)$.

Thus substituting these values in Eq. (iii), we get

$$\frac{R}{H} = \left[\left(\frac{2\times 9.81}{2\times 4\pi^2}\right)^3 \times \frac{\pi}{12\times 0.0142}\right]^{1/4} = 0.73.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. If a vessel containing a fluid is made to move with a constant acceleration, then although the fluid is in relative equilibrium and is at rest with respect to its container, the fluid takes up a new position under the effect of the acceleration imparted to its container, and comes to rest in this new position relative to the container.
2. Since the fluid after attaining a new position is still in static condition relative to its container, the laws of hydrostatics can be applied to evaluate the fluid pressure, and also there are no shear stresses the fluid pressure is everywhere normal to the surface on which it acts.
3. A vessel containing a fluid may be subjected to either translatory motion in horizontal or vertical direction, or a rotational motion at constant accelerations.
4. In a liquid container subjected to a constant horizontal acceleration α, the slope the surface of constant pressure is given by

$$\frac{dz}{dx} = -\frac{\alpha}{g}$$

where g = acceleration due to gravity.

5. In a liquid container subjected to a constant vertical acceleration α, the surface of constant pressure are horizontal.
6. In a liquid container subjected to a constant vertical upward acceleration α, the pressure at a depth h below the free surface of the liquid is given as

$$p = wh\left(1+\frac{\alpha}{g}\right).$$

7. In a liquid container subjected to a constant vertical downward acceleration α, the pressure at a depth h below the free surface of the liquid is given as

$$p = wh\left(1-\frac{\alpha}{g}\right).$$

8. If a cylindrical vessel containing liquid is rotated at a constant angular velovity ω with its axis vertical, then in this case the free surface of liquid developed is a paraboloid of revolution in shape with its section being a parabola represnted by the following equation

$$z = \frac{\omega^2 x^2}{2g}$$

where x and z are the coordinates of any point on the free surface of liquid with origin at the vertex of the parabola.

PROBLEMS

5.1 A tank partly filled with water is accelerated horizontally at a constant rate. The inclination of the water surface is 30°. What is the acceleration of the vessel ? [*Ans.* 5.66 m/s^2]

5.2 A cylindrical tank 1 m in diameter and 2 m long is half filled with a liquid (sp.gr. 1.6) and its ends are capped. It is then subjected to a horizontal acceleration of 4.9 m/s^2 with its axis horizontal. Find the pressure at the bottom of tank; at the rear and leading ends.

[*Ans.* 15.696 kN/m^2; zero]

5.3 A cylindrical vessel 2 m in diameter and 2 m high is completely filled with water. It is then rotated about its vertical axis at a speed of 60 r.p.m. Calculate the volume of water spilled out.

[*Ans.* 3.14 m^3]

5.4 A U-tube has a horizontal part 0.6 m long with vertical end limbs. If the whole tube is rotated about a vertical axis 0.2 m from one limb, calculate the speed when the difference of level in the two limbs is 0.3 m. [*Ans.* 67 r.p.m.]

5.5 A cylindrical vessel 4 m in diameter and 3 m high is filled with water to a depth of 2 m. Calculate (a) the speed at which water commences to spill ;(b) the speed at which the base of the vessel begins to be uncovered ; (c) the area of the base uncovered when speed reaches 50 r.p.m.

[*Ans.* (a) 30 r.p.m. ; (b) 37 r.p.m. ; (c) 5.82 m^2]

5.6 A vessel containing oil of specific gravity 0.76 moves vertically upward with an acceleration of 2.45 m/s^2. What is the pressure at a depth of 2 m [*Ans.* 18.635 kN/m^2]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

5.7 A closed vessel 0.6 m in diameter is completely filled with glycerine of specific gravity 1.6. If the vessel is rotated about its axis at 1200 r.p.m., what increase in pressure would occur at the top of the tank at the circumference ?

[*Ans.*1137 kN/m^2]

5.8 When an open rectangular tank 1.5 m wide, 3 m long and 2 m deep containing 1.2 m of water is accelerated horizontally parallel to its length at the rate of 4.9 m/s^2, how much water is spilled ? At what acceleration must this tank move in order that the depth at the front end be zero. [*Ans.* None ; 6.54 m/s^2]

5.9 An open tank containing oil of specific gravity 0.80 moves up a 30° inclined plane with an acceleration of 2.45 m/s^2. Find the angle which the free surface will make with the horizontal. Also find the slope of the free surface if the tank moves down the plane with the same acceleration. [*Ans.* 10° 53'; 13° 53']

5.10 A closed cylindrical tank 1 m in diameter and 3 m high is filled with water to a depth of 1.5 m. The pressure in the tank at its top is raised to 147.15 kN/m^2. If the tank is rotated at 200 r.p.m., calculate the pressure on the axis and at the wall, on the top and the bottom of the tank.

[**Note**: In this case due to excessive speed of rotation a portion of the bottom is uncovered]

[*Ans.*On the axis 147.15 kN/m^2 at top and bottom; at the walls 159.84 kN/m^2 at top and 189.27 kN/m^2 at bottom]

5.11 A cylindrical vessel filled with water is rotated about its axis. At one point on the axis, the pressure intensity is the same as at another point which is 2.5 m higher and 1 m from the axis. What is the speed of rotation in r.p.m ?

[*Ans.* 67 r.p.m.]

5.12 An open cubical tank 2 m side is filled with water. If the tank is accelerated such that one-third of the water spills out, what is the acceleration ?

[*Ans.* 6.54 m/s^2]

5.13. A 90° conical vessel of height 1.5 m contains water whose volume is one-half of the cone. Find the uniform speed at which the vessel may be rotated so that water will just begin to spill.

[*Ans.* 2.088 rad/s ; or 19.95 r.p.m.]

5.14 Mercury completely fills a closed tube 0.6 m long held vertically. If the tube is rotated about an axis 0.15 m from one end in a vertical plane at a constant speed of 90 r.p.m., what would be the pressure difference between the ends as the tube attains (i) its upper most, and (ii) its lower most, position. Take sp. gr. of mercury as 13.6.

[*Ans.* (i) 28.547 kN/m^2; (ii) 188.646 kN/m^2]

5.15 An open tank 10 m long and 2 m deep is filled with 1.5 m of oil of specific gravity 0.82. The tank is accelerated uniformly from rest to a speed of 20 m/s. What is the shortest time in which this speed may be attained without spilling any oil? [*Ans.* 20.39 s]

Apprise Education, Reprise Innovations

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Fundamentals of Fluid Flow

Chapter **6**

6.1 INTRODUCTION

In the preceding chapters the fluids at rest have been considered, in which case the only fluid property of significance is the specific weight of the fluids. On the other hand when a fluid is in motion various other fluid properties become significant. As such the nature of flow of a real fluid is complex and not always subject to exact mathematical analysis, and often recourse to experimentation is required. However, in some cases the mathematical analysis of problems of fluid flow is possible if some simplifying assumptions are made.

The science which deals with the geometry of motion of fluids without reference to the forces causing the motion is known as *hydrokinematics* or simply *kinematics*. Thus kinematics involves merely the description of the motion of fluids in terms of space-time relationship. On the other hand the science which deals with the action of the forces in producing or changing motion of fluids is known as *hydrokinetics* or simply *kinetics*. Obviously the study of fluids in motion involves the consideration of both the kinematics and the kinetics.

A fluid unlike solid, is composed of different particles, which move at different velocities and may be subject to different accelerations. Moreover, the velocity and acceleration of a fluid particle may change both with respect to time and space. Therefore in the study of fluid flow it is necessary to observe the motion of fluid particles at various points in space and at successive instants of time.

There are in general two methods by which the motion of a fluid may be described. These are the *Lagrangian* method and the *Eulerian* method.

In the Lagrangian method any individual fluid particle is selected, which is pursued throughout its course of motion and the observation is made about the behaviour of this particle during its course of motion through space. In the Eulerian method any point in the space occupied by the fluid is selected and observation is made of whatever changes of velocity, density and pressure which take place at that point. Out of these two methods the Eulerian method is commonly adopted in fluid mechanics and therefore the same is used in the following analysis.

6.2 VELOCITY OF FLUID PARTICLES

The motion of a fluid like that of solid is described quantitatively in terms of the characteristic known as velocity. However, in case of solids it is generally sufficient to measure the velocity of the body as a whole, but in the case of fluids the motion of fluid may be quite different at different points of observation.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Therefore the velocity V at any point of fluid mass is expressed as the ratio between the displacement of a fluid element along its path and the corresponding increment of time as the later approaches zero. As shown in Fig. 6.1, a particular point in the space occupied by a fluid in motion is selected, which may be denoted by the coordinates (x, y, z). Since this point is fixed in space, the coordinates x, y, z and the time t are independent variables. At this point if ds is the distance travelled by a fluid particle in time dt then the velocity V of the fluid particle at this point may be expressed as

$$V = \lim_{dt \to 0} \frac{ds}{dt}$$

The velocity is a vector quantity and hence it has magnitude as well as direction. Therefore the velocity V at any point in the fluid can be resolved into three components u, v and w along three mutually perpendicular directions x, y and z respectively. Each of these components can also be expressed as the limiting rate of displacement in the corresponding direction. Thus if dx, dy and dz are the components of the displacement ds in x, y and z directions respectively, then

$$u = \lim_{dt \to 0} \frac{dx}{dt}, \; v = \lim_{dt \to 0} \frac{dy}{dt} \text{ and } w = \lim_{dt \to 0} \frac{dz}{dt}$$

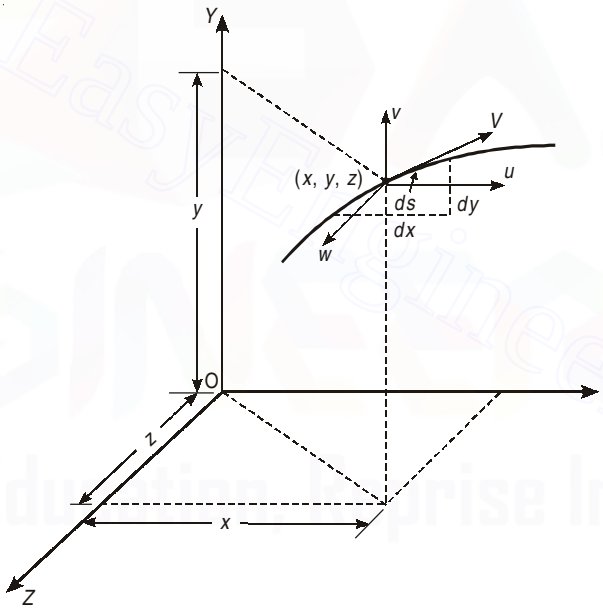


Figure 6.1 *Velocity at a point in fluid in motion*

Since the velocity V at any point in a flowing mass of fluid in general depends on x, y and z, i.e., the coordinate position of the point under consideration and the time t. Therefore the components u, v and w will also in general depend on x, y, z and t. Hence the velocity V and its components u, v and w may be expressed in terms of the following functional relationships

$$\left.\begin{aligned} V &= f_1(x, y, z, t) \\ u &= f_2(x, y, z, t) \\ v &= f_3(x, y, z, t) \\ w &= f_4(x, y, z, t) \end{aligned}\right\} \quad \ldots(6.1)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In vector notation velocity V may be expressed in terms of its components as

$$V = iu + jv + kw$$

where i, j, k are the unit vectors parallel to the x, y, z axes respectively.

For a particular point since the coordinates x, y and z may be regarded constant, the above expressions will indicate how u, v and w vary as the time t varies. On the other hand, if time t is regarded constant then since (x, y, z) may represent any point in the flowing mass of fluid the above expression will indicate how u, v and w vary at different points at a particular instant of time under consideration.

6.3 TYPES OF FLUID FLOW

According to different considerations fluid flows may be classified in several ways as indicated below:

(i) Steady flow and Unsteady flow.

(ii) Uniform flow and Non-uniform flow.

(iii) One-dimensional flow, Two dimensional flow and Three dimensional flow.

(iv) Rotational flow and Irrotational flow

(v) Laminar flow and Turbulent flow.

Steady Flow. Fluid flow is said to be steady if at any point in the flowing fluid various characteristics such as velocity, pressure, density, temperature etc., which describe the behaviour of the fluid in motion, do not change with time. In other words a steady flow may be defined as that in which at any point in the flowing fluid various characteristics which describe the behaviour of the fluid in motion are independent of time. However, these characteristics may be different at different points in the flowing fluid. Thus the steady flow may be expressed mathematically by the following expression at any point in the flowing fluid.

$$\left(\frac{\partial u}{\partial t}\right) = 0;\ \left(\frac{\partial v}{\partial t}\right) = 0;\ \left(\frac{\partial w}{\partial t}\right) = 0;\ \left(\frac{\partial p}{\partial t}\right) = 0;\ \left(\frac{\partial \rho}{\partial t}\right) = 0$$

Unsteady Flow. Fluid flow is said to be unsteady if at any point in the flowing fluid any one or all the characteristics which describe the behaviour of the fluid in motion change with time. Thus a flow of fluid is unsteady, if at any point in the flowing fluid

$$\left(\frac{\partial V}{\partial t}\right) \neq 0 \text{ ; and or } \left(\frac{\partial p}{\partial t}\right) \neq 0 \text{ etc.}$$

It may however be indicated that in the case of vector quantities such as velocity of flow etc., even the change in the direction of such quantities with respect to time at any point in the flowing fluid may also make the flow unsteady.

Obviously a steady flow is simpler to analyse than unsteady flow. Moreover, most of the practical problems of engineering involve only steady flow conditions. Therefore in this book the discussion has been mainly confined to the steady flow conditions only.

Uniform Flow. When the velocity of flow of fluid does not change, both in magnitude and direction, from point to point in the flowing fluid, for any given instant of time, the flow is said to be uniform. In the mathematical form a uniform flow may therefore be expressed as

$$\left(\frac{\partial V}{\partial s}\right) = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where time is held constant and s represents any direction of displacement of the fluid elements. The above expression states that there is no change in the velocity vector in any direction throughout the flowing fluid at any instant of time. For example flow of liquids under pressure through long pipe lines of constant diameter is uniform flow.

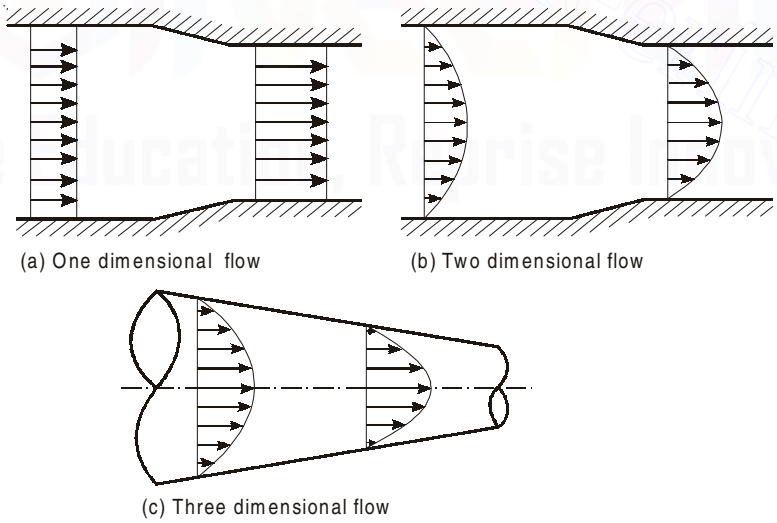
Non-uniform Flow. If the velocity of flow of fluid changes from point to point in the flowing fluid at any instant, the flow is said to be non-uniform. In the mathematical form a non-uniform flow may be expressed as

$$\left(\frac{\partial V}{\partial s}\right) \neq 0$$

For example flow of liquids under pressure through long pipelines of varying diameters is non-uniform flow.

All these types of flows can exist independent of each other so that any of the four types of combinations of flows is possible, viz., (a) steady-uniform flow; (b) steady-non-uniform flow; (c) unsteady uniform flow; and (d) unsteady-non-uniform flow. Some of the common examples of these combinations of flows are: flow of liquid through a long pipe of constant diameter at a constant rate is *steady uniform flow*; flow of liquid through a long pipe line of constant diameter, at either increasing or decreasing rate is *unsteady-uniform flow* ; flow of liquid through a tapering pipe at a constant rate is *steady-non-uniform flow* and flow through a tapering pipe at either increasing or decreasing rate is *unsteady-non-uniform flow*.

One-dimensional, Two-dimensional and Three-dimensional Flows. The various characteristics of flowing fluid such as velocity, pressure, density, temperature etc., are in general the functions of space and time i.e., these may vary with the coordinates of any point x, y and z and time t. Such a flow is known as a *three-dimensional flow*. If any of these characteristics of flowing fluid does not vary with respect to time, then it will be a steady three-dimensional flow.



Figuree 6.2 *One,Two and Three dimensional flows*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When the various characteristics of flowing fluid are the functions of only any two of the three-coordinate directions, and time t, i.e., these may not vary in any one of the directions, then the flow is known as *two-dimensional flow*. For example, if the characteristics of flowing fluid do not vary in the coordinate direction Z, then it will be a two-dimensional flow having flow conditions indentical in the various planes perpendicular to the Z-axis. Again it will be a steady two-dimensional flow if the characteristics of flowing fluid do not vary with respect to time.

When the various characteristics of flowing fluid are the functions of only one of the three coordinate directions and time t, i.e., these may vary only in one direction, then the flow is known as *one-dimensional flow*. Similarly, it will be a steady one dimensional flow if the characteristics of flowing fluid do not vary with respect to time. Considering one of the characteristics of flowing mass of fluid, say velocity of flow V, the following expressions may be written which clearly exhibit the difference between these three types of flows :

Types of Flow	*Unsteady*	*Steady*
Three-dimensional	$V = f(x, y, z, t)$	$V = f(x, y, z)$
Two-dimensional	$V = f(x, y, t)$	$V = f(x, y)$
One-dimensional	$V = f(x, t)$	$V = f(x)$

Similar expressions can also be written for various other characteristics of flowing fluid for the three different types of flows. Figure 6.2 shows the examples of one-dimensional, two-dimensional and three-dimensional flows.

Obviously the problems involving three-dimensional flow are quite complicated because of the variation of the characteristics of flowing fluid with x, y and z, and t. Even in steady three-dimensional flow the variation of the characteristics of flowing fluid with x, y and z render the problems fairly complicated to solve. In this respect the assumption of either two-dimensional flow or one-dimensional flow, considerably simplifies the problem making it amenable to analytical solution.

Rotational Flow. A flow is said to be rotational if the fluid particles while moving in the direction of flow rotate about their mass centres. In Chapter 5, the liquid in the rotating tanks illustrates rotational flow where the velocity of each particle varies directly as the distance from the centre of rotation.

Irrotational Flow. A flow is said to be irrotational if the fluid particles while moving in the direction of flow do not rotate about their mass centres. It may however be stated that a true irrotational flow exists only in the case of flow of an ideal fluid for which no tangential or shear stresses occur. But the flow of practical fluids, may also be assumed to be irrotational if the viscosity of the fluid has little significance.

Laminar Flow. A flow is said to be laminar when the various fluid particles move in layers (or laminae) with one layer of fluid sliding smoothly over an adjacent layer. Thus in the development of a laminar flow, the viscosity of the flowing fluid plays a significant role. As such the flow of a very viscous fluid may in general be treated as laminar flow.

Turbulent Flow. A fluid motion is said to be turbulent when the fluid particles move in an entirely haphazard or disorderly manner, that results in a rapid and continuous mixing of the fluid leading to momentum transfer as flow occurs. In such a flow eddies or vortices of different sizes and shapes are present which move over large distances. These eddies and their random movement give rise to fluctuations in the velocity and pressure at any point in the flow field, which are necessarily the functions of time. Thus at any point in a turbulent flow the velocity and pressure are the functions of time thereby rendering such a flow as unsteady. However, temporal mean values of the velocity and pressure considered over sufficiently long time do not change with time. Hence a turbulent flow in terms of temporal mean values of the velocity and pressure may be considered to be steady.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The occurrence of turbulent flow is more frequent than that of laminar flow. Flow in natural streams, artificial channels, water supply pipes, sewers etc., are a few examples of turbulent flow.

6.4 DESCRIPTION OF THE FLOW PATTERN

The flow pattern may be described by means of streamlines, stream-tubes, path lines and streak-lines.

Streamline. A *streamline* is an imaginary curve drawn through a flowing fluid in such a way that the tangent to it at any point gives the direction of the velocity of flow at that point. Since a fluid is composed of fluid particles, the pattern of flow of fluid may be represented by a series of stream-lines, obtained by drawing a series of curves through the flowing fluid such that the velocity vector at any point is tangential to the curves. Figure 6.3 shows some of the streamlines for a flow pattern in the xy plane in which a streamline passing through a point $P(x, y)$ is tangential to the velocity vector V at P. If u and v are the components of V along x and y directions, then

$$\frac{v}{u} = \tan\theta = \frac{dy}{dx}$$

where dy and dx are the y and x components of the differential displacement ds along the streamline in the immediate vicinity of P. Therefore the differential equation for streamlines in the xy plane may be written as

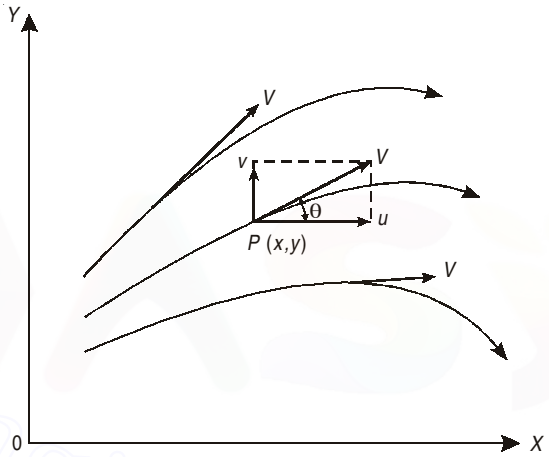


Figure 6.3 *Streamlines for a flow pattern in plan*

$$\frac{dx}{u} = \frac{dy}{v} \text{ ; or } (udy - vdx) = 0 \quad \ldots (6.2)$$

A general differential equation for streamlines in space for three-dimensional flow may however be obtained in the same manner as

$$\frac{dx}{u} = \frac{dy}{v} = \frac{dz}{w} \quad \ldots (6.3)$$

Since a streamline is everywhere tangent to the velocity vector, there can be no component of the velocity at right angles to the streamline and hence there can be no flow of fluid across any streamline. Further in steady flow, since there is no change in direction of the velocity vector at any point, the flow pattern is not changing. But in unsteady flow, since the direction of the velocity vector at any point may change with time, the flow pattern also changes with time. Therefore for a steady flow the streamline pattern remains the same at different times, but for an unsteady flow the streamline pattern may change from time to time, and hence in the case of unsteady flow the streamlines are to be regarded as instantaneous streamlines representing an instantaneous flow pattern only.

Stream-tube. A *stream-tube* is a tube imagined to be formed by a group of streamlines passing through a small closed curve, which may or may not be circular, as shown in Fig. 6.4. Since the stream-tube is bounded on all sides by streamlines and since the velocity has no component normal to a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

streamline, there can be no flow across the bounding surface of a stream-tube. Therefore a fluid may enter or leave a stream-tube only at its ends. A stream-tube with a cross-sectional area small enough for the variation of velocity over it to be negligible is sometimes termed as a *stream filament*. The concept of stream-tubes is quite useful in analysing several fluid flow problems, since the entire flow field may be divided into a large number of stream-tubes, thus yielding a clear picture of the actual pattern of flow. However, only in steady flow a stream-tube is fixed in space.

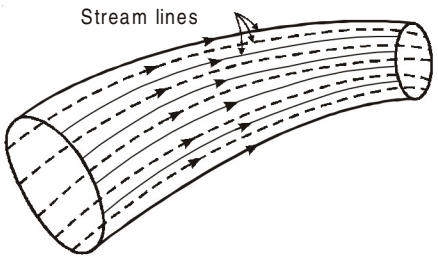


Figure 6.4 *Stream tube*

Path-line. A *path-line* may be defined as the line traced by a single fluid particle as it moves over a period of time. Thus a path-line will show the direction of velocity of the same fluid particle at successive instants of time. As indicated earlier a streamline on the other hand shows the direction of velocity of a number of fluid particles at the same instant of time. A fluid particle always moves tangent to the streamline, and in the case of steady flow since there is no change in direction of the velocity vector at any point with time, the streamline is fixed in space. Therefore in steady flow the path-lines and streamlines are identical. However, in unsteady flow since the direction of velocity vector at any point may change with time, streamline may shift in space from instant to instant. A particle then follows one streamline at one instant and another at the next instant and so on, so that the path of the particle may have no resemblance to any given instantaneous streamline. In other words, in unsteady flow path-lines and streamlines are different.

Streak-line. In experimental work often a colour or dye or some other substance (such as smoke in case of gases) is injected into the flowing fluid, in order to trace the motion of the fluid particles. The resulting trail of colour is known as a *streak-line* or *filament line*. Thus a streak-line may be defined as a line that is traced by a fluid particle passing through a fixed point in a flow field.

In steady flow, since there is no change in the flow pattern, a streak-line is same as a streamline and the path-line of a particle. In other words, in steady flow a streak-line, a streamline and a path-line are all identical. In unsteady flow a streak-line at any instant is the locus of end points of particle paths

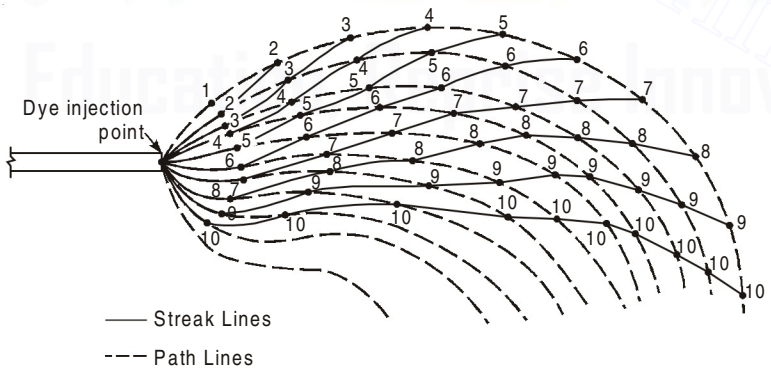


Figure 6.5 *Streak–lines in unsteady flow*

(or path-lines) that started at the instant the particle passed through the injection point. Figure 6.5 indicates streak-lines in a possible unsteady flow situation. It is assumed that a dye is injected into the fluid for a 10-second period. The position of the particle that was at the dye jet for each second t = 0, 1, 2, 3, 4.....10 is plotted for each subsequent second in the 10-second period. The dots represent

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the position of each of these particles at the time indicated, the dotted lines represent the paths of particles (i.e., path-lines) and the solid lines represent the position of the streak-line at each second of time. At t = 1, it is practically the path of the particle; at t =2 it is the line through the points marked 2 and moves so that it takes up the new positions marked by the time at the end of each one second interval, However, only one streak-line is visible at a given instant.

6.5 BASIC PRINCIPLES OF FLUID FLOW

Alike solid mechanics there are three basic principles used in the analysis of the problems of fluid in motion as noted below :

(i) Principle of conservation of mass.

(ii) Principle of conservation of energy.

(iii) Principle of conservation of momentum.

The *principle of conservation of mass* states that mass can neither be created nor destroyed. On the basis of this principle the continuity equation is derived.

The *principle of conservation of energy* states that energy can neither be created nor destroyed. On the basis of this principle the energy equation is derived.

The *principle of conservation of momentum* or *impulse momentum principle* states that the impulse of the resultant force, or the product of the force and time increment during which it acts, is equal to the change in the momentum of the body. On the basis of this principle the momentum equation is derived.

In applying these principles usually control volume approach is adopted, in which a definite volume with fixed boundary shape is chosen in space along the fluid flow passage. This definite volume is called the *control volume* and the boundary of this volume is known as the *control surface*. The boundaries of the control volume may be extended upto such an extent that it includes the portion of the flow passage which is to be studied. In fact the use of the Eulerian method for describing fluid motion, implies the use of the control volume approach without specifying it.

In this chapter only the continuity equation is derived. The derivation of energy and momentum equations and their application are dealt with in the subsequent chapters.

6.6 CONTINUTTY EQUATION

The continuity equation is actually mathematical statement of the principle of conservation of mass. A most general expression on the basis of this principle may be obtained by considering a fixed region within a flowing fluid as shown in Fig. 6.6. Since fluid is neither created nor destroyed within this region it may be stated that the rate of increase of the fluid mass contained within the region must be equal to the difference between the rate at which the fluid mass enters the region and the rate at which the fluid mass leaves the region. However if the flow is steady, the rate of increase of the fluid mass

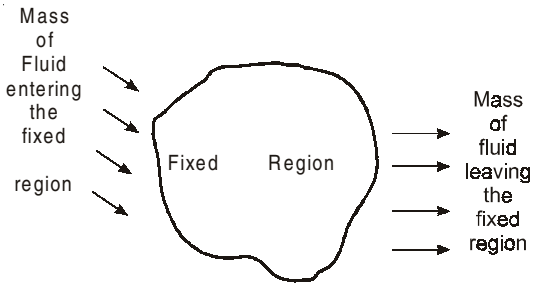


Figure 6.6 *Diagrammatic representation of the principle of conservation of mass*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

within the region is equal to zero, then the rate at which the fluid mass enters the region is equal to the rate at which the fluid mass leaves the region. This relation is used to derive a general equation of continuity for a three dimensional steady or unsteady flow, as indicated below.

Continuity Equation in Cartesian Coordinates. Consider an elementary rectangular parallelopiped with sides of length $\delta x, \delta y$ and δz as shown in Fig. 6.7. Let the centre of the parallelopiped be at a point $P(x, y, z)$ where the velocity components in the x, y, z direction are u, v and w respectively and ρ be the mass density of the fluid.

The mass of fluid passing per unit time through the face of area $\delta y\ \delta z$ normal to the x-axis through point P, is

$$(\rho u\ \delta y\ \delta z)$$

Then the mass of fluid flowing per unit time into the parallelopiped through the face $ABCD$ is

$$\left[(\rho u\ \delta y\ \delta z) + \frac{\partial}{\partial x}(\rho u\ \delta y\ \delta z)\left(-\frac{\delta x}{2}\right)\right]$$

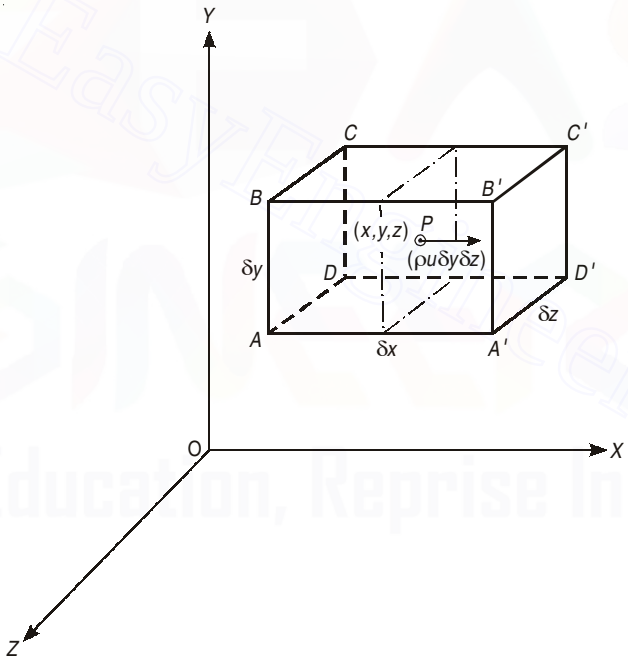


Figure 6.7 *Elementary rectangular parallelopiped*

In the above expression negative sign has been used because face $ABCD$ is on the left of point P.

Similarly the mass of fluid flowing per unit time out of the parallelopiped through the face $A'B'C'D'$ is

$$\left[(\rho u\ \delta y\ \delta z) + \frac{\partial}{\partial x}(\rho u\ \delta y\ \delta z)\left(\frac{\delta x}{2}\right)\right]$$

Therefore, the net mass of fluid that has remained in the parallelopiped per unit time through the pair of faces $ABCD$ and $A'B'C'D'$ is obtained as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left[\rho u\ \delta y\ \delta z - \frac{\partial}{\partial x}(\rho u\ \delta y\ \delta z)\frac{\delta x}{2}\right] - \left[\rho u\ \delta y\ \delta z + \frac{\partial}{\partial z}(\rho u\ \delta y\ \delta z)\frac{\delta x}{2}\right] = -\frac{\partial}{\partial x}(\rho u\ \delta y\ \delta z)\delta x$$

$$= -\frac{\partial}{\partial x}(\rho u)\,\delta x\,\delta y\,\delta z$$

The area ($\delta y\ \delta z$) has been taken out of the parentheses since it is not a function of x.

By applying the same procedure the net mass of fluid that remains in the parallelopiped per unit time through the other two pairs of faces of the parallelopiped may also be obtained as

$$= -\frac{\partial}{\partial y}(\rho v)\,\delta x\,\delta y\,\delta z, \text{ through pair of faces } AA'D'D \text{ and } BB'C'C$$

$$= -\frac{\partial}{\partial z}(\rho w)\,\delta x\,\delta y\,\delta z, \text{ through pair of faces } DD'C'C \text{ and } AA'B'B$$

By adding all these expressions the net total mass of fluid that has remained in the parallelopiped per unit time is obtained as

$$-\left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z}\right]\delta x\ \delta y\ \delta z$$

Since the fluid is neither created nor destroyed in the parallelopiped, any increase in the mass of the fluid contained in this space per unit time is equal to the net total mass of fluid that remained in the parallelopiped per unit time, which is represented by the above expression.

The mass of the fluid in the parallelopiped is ($\rho\ \delta x\ \delta y\ \delta z$) and its rate of increase with time is

$$\frac{\partial}{\partial t}(\rho\ \delta x\ \delta y\ \delta z) = \frac{\partial \rho}{\partial t}(\delta x\ \delta y\ \delta z)$$

Equating the two expressions

$$-\left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z}\right](\delta x\ \delta y\ \delta z) = \frac{\partial \rho}{\partial t}(\delta x\ \delta y\ \delta z)$$

Dividing both sides of the above expression by the volume of the parallelopiped ($\delta x\ \delta y\ \delta z$) and taking the limit so that the parallelopiped shrinks to the point $P(x, y, z)$, the continuity equation is obtained as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad \text{...(6.4)}$$

Equation 6.4 represents the continuity equation in cartesian coordinates in its most general form which is applicable for steady as well as unsteady flow, uniform and non-uniform flow, and compressible as well as incompressible fluids.

For steady flow since, $\frac{\partial \rho}{\partial t} = 0$, Eq. 6.4 reduces to

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad \text{... (6.5)}$$

Further for an incompressible fluid the mass density ρ does not change with x, y, z and t and hence Eq. 6.4 simplifies to

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad \text{... (6.6)}$$

In vector notation the continuity equation may be expressed as

$$\frac{D\rho}{Dt}+\rho\nabla.V=0$$

in which

$$\frac{D\rho}{Dt}=\frac{\partial\rho}{\partial t}+u\ \frac{\partial\rho}{\partial x}+v\ \frac{\partial\rho}{\partial y}+w\frac{\partial\rho}{\partial z}$$

and

$$\nabla.V=\text{div } V=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}$$

Continuity Equation in Cylindrical Polar Coordinates. Often a continuity equation is required to be used in terms of the cylindrical polar coordinates which may also be derived by adopting the procedure as indicated below.

Consider any point $P(r, \theta, z)$ in space. Let δr, $\delta\theta$ and δz be the small increments in the direcions r, θ and z respectively, so that $PS = \delta r$, $PQ = r\delta\theta$ and $PT = \delta z$. Construct an elementary parallelopiped as shown in Fig. 6.8. Let V_r, V_θ and V_z be the components of the velocity V in the directions of r, θ and z at point P. Further let ρ represent the mass density of fluid at point P.

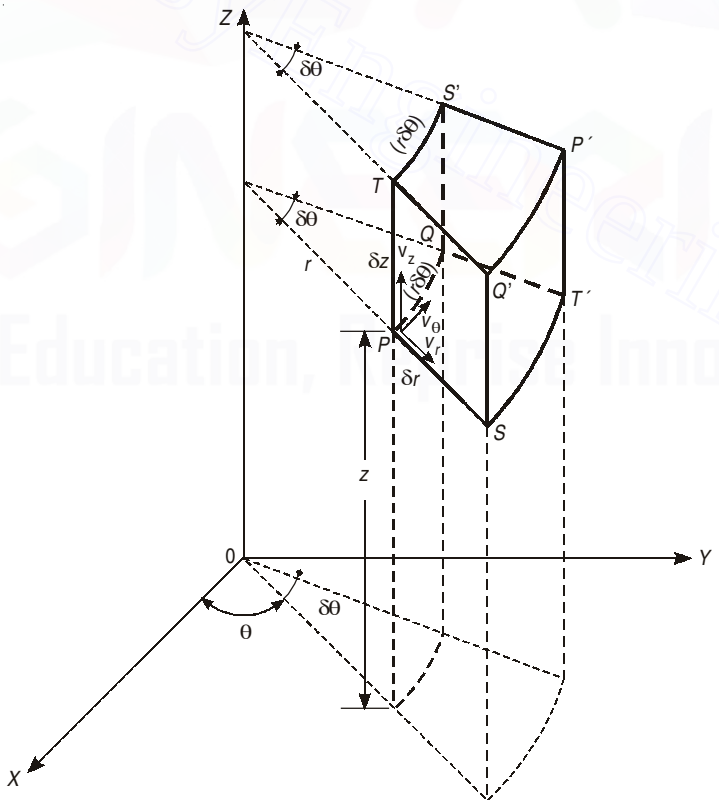


Figure 6.8 *Elementary cylindrical parallelopiped*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Considering the pair of faces $PST'Q$ and $P'S'TQ'$, the mass of fluid entering the parallelopiped per unit time through the face $PST'Q$

$$= \rho V_z \, (\delta r \times r\delta\theta)$$

Mass of fluid leaving the parallelopiped per unit time through the face $P'S'TQ'$

$$= \rho V_z \, (\delta r \times r\delta\theta) + \frac{\partial}{\partial z}(\rho V_z \delta r \times r\delta\theta)\delta z$$

Therefore the net mass of fluid that has remained in the parallelopiped per unit time through this pair of faces

$$= -\frac{\partial}{\partial z}(\rho V_z \delta r \times r\delta\theta)\delta z$$

Similarly the net mass of fluid that remains in the parallelopiped per unit time through the pair of faces $PTQ'S$ and $P'T'QS'$

$$= -\frac{\partial}{\partial \theta}(\rho V_\theta \delta r \, \delta z)\delta\theta$$

and that through the pair of faces $PQS'T'$ and $P'Q'ST$

$$= -\frac{\partial}{\partial r}(\rho V_r r\delta\theta\delta z)\delta r$$

By adding all these expressions the net total mass of fluid that has remained in the parallelopiped per unit time through all the three pairs of faces

$$= -\left[\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(\rho V_z)}{\partial z}\right](\delta r \delta z r\delta\theta)$$

The mass of fluid in the parallelopiped

$$= \rho \, (\delta r \, \delta z \, r\delta\theta)$$

and its rate of increase with time

$$= \frac{\partial}{\partial t}(\rho \delta r \delta z r\delta\theta)$$

$$= \frac{\partial \rho}{\partial t}(\delta r \delta z r\delta\theta)$$

Thus equating the two expressions

$$-\left[\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(\rho V_z)}{\partial z}\right](\delta r \delta z r\delta\theta) = \frac{\partial \rho}{\partial t}(\delta r \delta z r\delta\theta)$$

Dividing both sides of the above expression by the volume of the parallelopiped ($\delta r \, \delta z \, r\delta\theta$) and taking the limit so as to reduce the parallelopiped to point P, the continuity equation is obtained as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(\rho V_z)}{\partial z} = 0 \qquad \ldots(6.7)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 6.7 represents the continuity equation in cylindrical polar coordinates in its most general form which is applicable for steady or unsteady flow, uniform or non-uniform flow and compressible or incompressible fluids.

Again for steady flow since, $\frac{\partial \rho}{\partial t} = 0$, Eq. 6.7 reduces to

$$\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(V_z)}{\partial z} = 0 \qquad \ldots(6.8)$$

Further for an incompressible fluid the mass density ρ does not change with r, θ, z and t and hence Eq. 6.7 simplifies to

$$\frac{\partial(V_r r)}{r\partial r} + \frac{\partial(V_\theta)}{r\partial\theta} + \frac{\partial(V_z)}{\partial z} = 0 \qquad \ldots(6.9)$$

Continuity Equation in Spherical Polar Coordinates. By considering an elementary parallelopiped as shown in Fig. 6.9 and adopting the same procedure, the equation of continuity in terms of

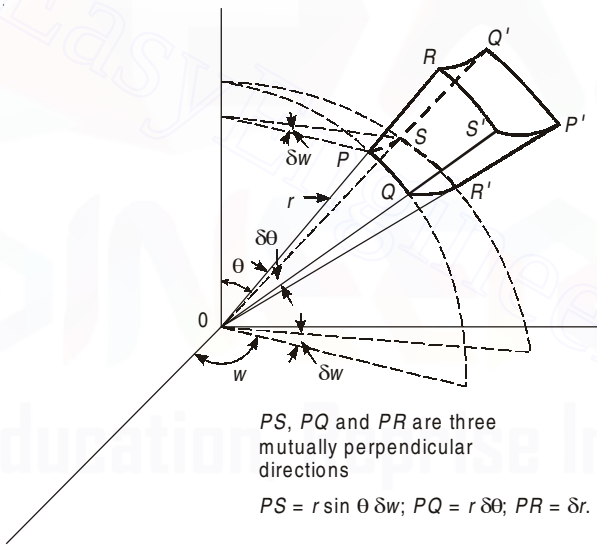


Figure 6.9 *Elementary spherical parallelopiped*

spherical polar coordinates can also be derived. However, it is not derived here, but simply mentioned in its most general form as noted below:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r^2}\frac{\partial}{\partial r}(\rho V_r r^2) + \frac{1}{r\sin\theta}\frac{\partial}{\partial\theta}(\rho V_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial w}(\rho V_w) = 0 \qquad \ldots(6.10)$$

in which the coordinates of any point are represented as (r, θ, w).

Fot steady flow since $\frac{\partial \rho}{\partial t} = 0$, Eq. 6.10 reduces to

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{1}{r^2}\frac{\partial}{\partial r}(\rho V_r r^2) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \theta}(\rho V_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial w}(\rho V_w) = 0 \quad \text{... (6.11)}$$

Further for an incompressible fluid the mass density ρ does not change with r, θ and w and hence Eq. 6.10 simplifies to

$$\frac{1}{r^2}\frac{\partial}{\partial r}\left(V_r r^2\right) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \theta}(V_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial w}(V_w) = 0 \quad \text{... (6.12)}$$

Continuity Equations for Two-Dimensional and One-Dimensional Flows. In several practical problems the flow may be considered as either two-dimensional or one-dimensional only. The consideration leads to a considerable simplification of the continuity equation as indicated below.

In two-dimensional flow the various characteristics of flowing fluid are the functions of only any two of the three coordinate directions. Thus in this case same flow pattern is displayed in all the planes perpendicular to the coordinate direction in which the characteristics of flowing fluid do not vary. The velocity vector has therefore components in only two coordinate directions and in the third coordinate direction in which the characteristics of flowing fluid do not vary, the component of the velocity is every where equal to zero. The continuity equation may then be modified accordingly for the case of a two-dimensional flow. Thus assuming that the characteristics of flowing fluid do not vary in the coordinate direction z $\left(\text{i.e.,}\frac{\partial(\rho u)}{\partial z} = 0\right)$ then the same flow pattern is displayed in all the planes parallel to x–y plane and the component of velocity in the z-direction, $w = 0$. The equation of continuity for two-dimensional flow may therefore be obtained in cartesian coordinates from Eq. 6.4 as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad \text{...(6.13)}$$

Equation 6.13 represents the continuity equation in the most general form for a two-dimensional flow which may also be derived directly by considering a two-dimensional elementary parallelopiped and adopting the same procedure as in the case of three-dimensional flow. For steady two-dimensional flow since $\frac{\partial \rho}{\partial t} = 0$, Eq. 6.13 reduces to

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad \text{...(6.14)}$$

Further for an incompressible fluid the mass density ρ does not change with x, y and t and hence Eq. 6.13 simplifies to

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad \text{...(6.15)}$$

Similarly for a two-dimensional flow continuity equation in cylindrical polar coordinates may be obtained from Eq. 6.7.

Thus if it is assumed that the component $V_z = 0$ then the continuity equation for a two-dimensional flow in cylindrical polar coordinates may be obtained from Eq. 6.7 as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial(\rho V_r r)}{\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} = 0 \quad \text{... (6.16)}$$

For steady flow Eq. 6.16 simplifies to

$$\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} = 0 \quad \text{... (6.17)}$$

and for an incompressible fluid Eq. 6.16 further simplifies to

$$\frac{\partial(V_r r)}{r\partial r} + \frac{\partial(V_\theta)}{r\partial\theta} = 0 \quad \text{... (6.18)}$$

In one -dimensional flow the various characteristics of flowing fluid are the functions of only one of the three coordinate directions which is the general direction of flow. One-dimensional consideration of flow leads to a simple form of continuity equation which may also be derived from Eq. 6.4. Thus if it is assumed that the characteristics of flowing fluid do not vary in the coordinate directions y and z $\left(\text{i.e.,} \frac{\partial(\rho u)}{\partial y} = 0, \frac{\partial(\rho u)}{\partial z} = 0\right)$ then for one-dimensional flow the equation of continuity Eq. in Cartesian coordinates may be obtained from equation 6.4 as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} = 0$$

The continuity equation in the above noted form does not involve the cross-sectional area of the flow passage and hence it is applicable to the case of one-dimensional flow in which the flow passage has uniform cross-sectional area. In fact a true one-dimensional flow would occur only in a straight flow passage of uniform cross-sectional area. However, one-dimensional flow may also be assumed to occur in the case of straight or curved flow passage with varying cross-sectional area, if the velocity of flow is uniform at each section of the flow passage. The continuity equation involving the variation of the cross-sectional area of the flow passage may be derived for one-dimensional flow as explained below.

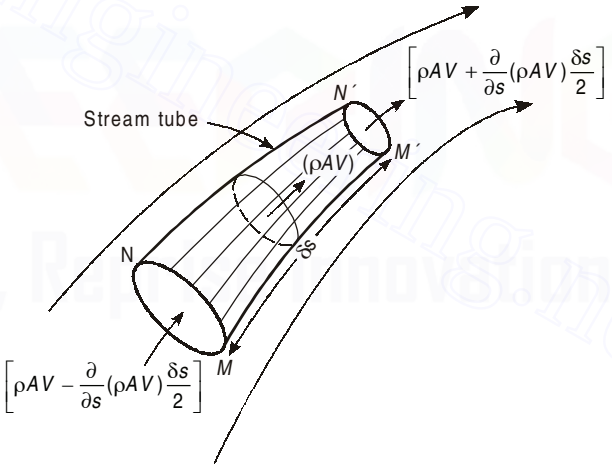


Figure 6.10 *Flow through a stream tube*

Consider a tube-shaped elementary parallelopiped along a stream-tube of length δs as shown in Fig. 6.10. Since the flow through a stream-tube is always along the tangential direction, there is no component of velocity in the normal direction. If at the central section of the elementary stream-tube, A is the cross-sectional area, V is the mean velocity of flow and ρ is the mass density of the fluid, then the mass of fluid passing through this section per unit time is equal to (ρAV).

The mass of fluid entering the parallelopiped per unit time at section NM

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \left[\rho AV - \frac{\partial}{\partial s}(\rho AV)\frac{\delta s}{2}\right]$$

Similarly the mass of fluid leaving the parallelopiped per unit time at section $N'M'$

$$= \left[\rho AV + \frac{\partial}{\partial s}(\rho AV)\frac{\delta s}{2}\right]$$

Therefore the net mass of fluid that has remained in the parallelopiped per unit time

$$= -\frac{\partial}{\partial s}(\rho AV)\,\delta s$$

The mass of fluid in the parallelopiped

$$= (\rho A \delta s)$$

and its rate of increase with time

$$= \frac{\partial}{\partial t}(\rho A \delta s) = \frac{\partial}{\partial t}(\rho A)\,\delta s$$

Since the net mass of fluid that has remained in the parallelopiped per unit time is equal to the rate of increase of mass with time,

$$-\frac{\partial}{\partial s}(\rho AV)\,\delta s = \frac{\partial}{\partial t}(\rho A)\,\delta s$$

Dividing both sides of the above expression by δs and taking the limit so as to reduce the parallelopiped to a point the continuity equation is obtained as

$$\frac{\partial(\rho A)}{\partial t} + \frac{\partial(\rho AV)}{\partial s} = 0 \qquad ...(6.19)$$

Equation 6.19 represents the continuity equation for one-dimensional flow in a most general form which will be applicable for steady or unsteady flow, uniform or non-uniform flow, and for compressible fluids.

For steady flow since there is no variation with respect to time Eq. 6.19 reduces to

$$\frac{\partial(\rho AV)}{\partial s} = 0 \qquad ...(6.20)$$

from which we obtain

$$\rho AV = \text{constant} \qquad ...(6.21)$$

or $$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 = \rho_3 A_3 V_3 = \text{constant} \qquad ...(6.21\ a)$$

where ρ, A and V with subscripts 1, 2 and 3 correspond to mass density, cross-sectional area and velocity of flow respectively at any three sections of stream-tube under consideration. Equation 6.21 (a) thus represents equation of continuity which is applicable to a steady one-dimensional flow of compressible as well as incompressible fluids. However, for an incompressible fluid since the mass density ρ is constant and does not vary from point to point, Eq. 6.21 may be further simplified as

$$\frac{\partial(AV)}{\partial s} = 0 \qquad ...(6.22)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

from which we obtain

$$AV = \text{constant} \quad ...(6.23)$$

or $$A_1V_1 = A_2V_2 = A_3V_3 = \text{constant} \quad ...(6.23\ a)$$

Further $AV = q$, where q is the volume of fluid flowing through any section per unit time or the volume rate of flow of fluid which is known as *discharge*. Discharge is expressed in terms of cubic metre per second (m^3/s) or litres per second. One cubic metre per second is usually written as one cumec. Equation 6.23 (a) may then be expressed as

$$q = A_1V_1 = A_2V_2 = A_3V_3 \quad ...(6.23\ b)$$

Equation 6.23 thus represents equation of continuity which is applicable to a steady one-dimensional flow of incompressible fluid. Equations 6.21 (a) and 6.23 (b) have been derived for a stream-tube having small cross-sectional areas A_1, A_2, A_3, etc., so that the velocity of flow at each section V_1, V_2, V_3 etc., may be assumed to be uniform for the entire cross-sectional area. However, these equations can also be applied to flow passages of larger areas, even if the velocity of flow at any cross-section of the flow passage is not uniform, i.e., it varies from point to point. In such cases the flow passage may be assumed to be divided into a number of small stream-tubes (or stream filaments) as shown in Fig. 6.11. If at any section *MN* of the flow passage the small stream-tubes have cross-sectional areas dA_1, dA_2, dA_3 etc., and the velocities of flow through these areas being v_1, v_2, v_3, etc., then according to Eq. 6.23 the discharge through each of these small areas is obtained as

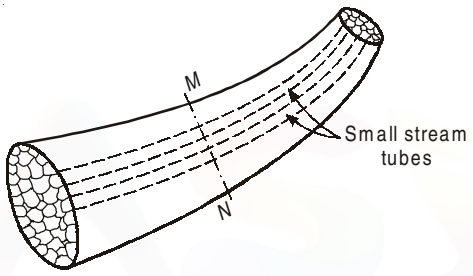


Figure 6.11 *Flow passage divided into a number of small stream-tubes*

$$dQ_1 = v_1dA_1,\ dQ_2 = v_2dA_2,\ dQ_3 = v_3dA_3; \text{ etc.}$$

The total discharge Q passing through the entire section of the flow passage of cross-sectional area A is then obtained as

$$\begin{aligned} Q &= dQ_1 + dQ_2 + dQ_3 + ... \\ &= v_1dA_1 + v_2dA_2 + v_3dA_3 + ... \\ &= \Sigma vdA \end{aligned}$$

Moreover the discharge Q may also be expressed in terms of the average or mean velocity of flow through the section. Thus if V is the mean velocity of flow at any section of the flow passage of area A then

$$Q = AV, \text{ where } V = \frac{1}{A}\int vdA$$

Further if the flow passage has cross-sectional areas A_1, A_2, A_3, etc., at different sections taken normal to the direction of flow, where the mean velocities of flow are V_1, V_2, V_3 etc., then from Eq. 6.23 (b) the discharge Q flowing through the passage may be expressed as

$$Q = A_1V_1 = A_2V_2 = A_3V_3 \quad ...(6.24)$$

It is observed from Eq. 6.24 that the velocity of flow at any section in inversely proportional to the area of flow section, that is, as the area of flow section increases the velocity of flow decreases and as the area of flow section decreases the velocity of flow increases.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Although Eq. 6.24 is derived for one-dimensional flow conditions and accordingly it is usually referred to as the one-dimensional equation of continuity, yet in most engineering problems of fluid flow in pipes and open channels, the one-dimensional approach is used by considering the mean velocities of flow at the flow sections.

Moreover the one-dimensional approach may also be used to derive a simplified continuity equation for the problems of fluid flow involving two-dimensional flow. In such a case as shown in Fig. 6.12 the space between two adjacent streamlines on the two planes parallel to the xy plane at a unit distance (in the z direction) apart can be considered as a stream-tube. Then the area along the stream-tube is numerically equal to the spacing dn of the streamlines. Now if v_1 and v_2 are the mean velocities at sections 1 and 2 where the spacings between the two adjacent streamlines are dn_1 and dn_2 respectively, then according to Eq. 6.23 for steady flow of incompressible fluid through a stream-tube in a two-dimensional flow, the discharge dq may be expressed as

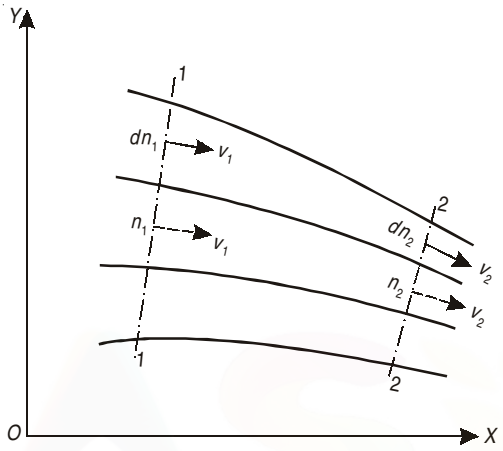


Figure 6.12 *Two-dimensional streamlines*

$$dq = v_1 dn_1 = v_2 dn_2$$

which also shows that the velocity of flow between two adjacent streamlines is inversely proportional to the spacing of the streamlines. The continuity equation for the total volume rate of flow of fluid per unit width of the entire section of the flow passage may be obtained by the integration of the above expression. Therefore

$$q = \int v_1 dn_1 = \int v_2 dn_2$$

or

$$q = V_1 n_1 = V_2 n_2 \qquad \text{...(6.25)}$$

where V_1 and V_2 are the mean velocities of flow for the entire flow passage at sections 1 and 2, and n_1 and n_2 are the total depths of flow of the entire flow passage in the directions normal to the mean velocities of flow at the sections 1 and 2 respectively.

6.7 ACCELERATION OF A FLUID PARTICLE

Acceleration is defined as the rate of change of velocity with respect to time. As stated earlier, the velocity of a fluid particle is a function of both the positions of the point and time. Thus if a certain fluid particle has a velocity V which has u, v and w as the three components along three mutually perpendicular directions x, y and z, then the acceleration in the x, y and z directions are given by

$$\left.\begin{aligned} a_x &= \lim_{dt \to 0} \frac{du}{dt} \\ a_y &= \lim_{dt \to 0} \frac{dv}{dt} \\ a_z &= \lim_{dt \to 0} \frac{dw}{dt} \end{aligned}\right\} \qquad \ldots(6.26)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

But Eq. 6.1 indicates that $u = f(x, y, z, t)$, and hence the total or substantial derivative of u with respect to time is

$$\frac{du}{dt} = \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\frac{dz}{dt} + \frac{\partial u}{\partial t}\frac{dt}{dt}$$

Further it has been shown that

$$\lim_{dt\to 0}\frac{dx}{dt} = u,\ \lim_{dt\to 0}\frac{dy}{dt} = v, \text{ and } \lim_{dt\to 0}\frac{dz}{dt} = w$$

Therefore substantial derivative of u becomes

$$\lim_{dt\to 0}\frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}$$

or
$$a_x = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t} \quad ...(6.27)$$

By adopting the same procedure as above the following expressions for the other two components of acceleration a_y and a_z may also be obtained.

$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t} \quad ...(6.28)$$

$$a_z = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t} \quad ...(6.29)$$

In vector notation acceleration may be represented as

$$a = \frac{DV}{Dt} = \frac{\partial V}{\partial t} + V.\nabla\ V$$

where
$$a = ia_x + ja_y + ka_z$$

Equations (6.27), (6.28) and (6.29) represent the expressions for the components of acceleration in the three mutually perpendicular directions. In these expressions $\left(\frac{\partial u}{\partial t}\right)$ or $\left(\frac{\partial v}{\partial t}\right)$ or $\left(\frac{\partial w}{\partial t}\right)$ represents the rate of increase of velocity with respect to time at a particular point in the flow and hence it is known as *local acceleration* or *temporal acceleration.* The remaining terms in these expressions represent the rate of increase of velocity due to the particle's change of position and hence it is known as *convective acceleration.* In steady flow the local acceleration in zero, but the convective acceleration is not necessarily zero and hence the total or substantial acceleration is not necessarily zero. However, in the case of uniform flow the convective acceleration is also zero.

Alike velocity, acceleration is also a vector quantity. However, unlike velocity vector the acceleration vector has no specific orientation with respect to the streamline i.e., it need not be always tangential to streamline. In other words the acceleration vector may have any direction so that at any point it has components both tangential and normal to the streamline. The tangential acceleration is developed for a fluid particle when the magnitude of the velocity changes with respect to space and time. On the other hand a normal acceleration is developed when a fluid particle moves in a curved path along

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which the direction of the velocity changes. Hence a normal acceleration may be developed simply due to the change in the direction of the velocity of the fluid particle, regardless of whether the magnitude of the velocity is changing. The expressions for the normal and the tangential accelerations may be developed as indicated below.

Tangential and Normal Accelerations. Let s and n represent the tangential and normal directions respectively at any point on the streamline as shown in Fig. 6.13. Let V_S and V_n be the components of velocity along tangential and normal directions respectively.

Then $V_s = f_1(s, n, t)$ and $V_n = f_2(s, n, t)$

The accelerations a_s and a_n in the tangential and the normal directions may be expressed as

$$a_s = \lim_{dt \to 0} \frac{dV_s}{dt} \text{ and } a_n = \lim_{dt \to 0} \frac{dV_n}{dt}$$

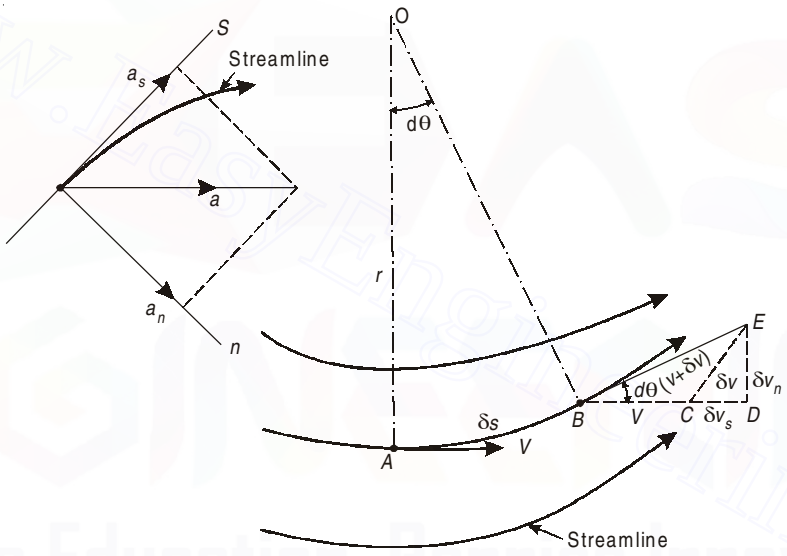


Figure 6.13 *Tangential and normal accelerations*

The tangential component of the acceleration is due to the change in the magnitude of velocity along the streamline and the normal component of the acceleration is due to the change in the direction of velocity vector.

Using the partial differentiation the total or substantial derivatives of V_s and V_n with respect to time are obtained as

$$\frac{dV_s}{dt} = \frac{\partial V_s}{\partial s}\frac{ds}{dt} + \frac{\partial V_s}{\partial n}\frac{dn}{dt} + \frac{\partial V_s}{\partial t}\frac{dt}{dt}$$

and

$$\frac{dV_n}{dt} = \frac{\partial V_n}{\partial s}\frac{ds}{dt} + \frac{\partial V_n}{\partial n}\frac{dn}{dt} + \frac{\partial V_n}{\partial t}\frac{dt}{dt}$$

Further $\lim_{dt \to 0} \frac{ds}{dt} = V_s$ and $\lim_{dt \to 0} \frac{dn}{dt} = V_n$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence $$\lim_{dt\to 0}\frac{dV_s}{dt} = a_s = V_s\frac{\partial V_s}{\partial s} + V_n\frac{\partial V_s}{\partial n} + \frac{\partial V_s}{dt}$$

and $$\lim_{dt\to 0}\frac{dV_n}{dt} = a_n = V_s\frac{\partial V_n}{\partial s} + V_n\frac{\partial V_n}{\partial n} + \frac{\partial V_n}{\partial t}$$

Since for any streamline $V_n = 0$, the above expressions for the tangential and the normal accelerations become

$$a_s = V_s\frac{\partial V_s}{\partial s} + \frac{\partial V_s}{\partial t} \qquad ...(6.30)$$

and $$a_n = V_s\frac{\partial V_n}{\partial s} + \frac{\partial V_n}{\partial t} \qquad ...(6.31)$$

In the above expressions it is to be noted that even though $V_n = 0$, $\left(\frac{\partial V_n}{\partial s}\right)$ need not be zero. This is so because $V_n = 0$ at any point on the streamline but at any other point on the streamline the component of the velocity in the direction parallel to that of V_n need not always be zero.

Consider a series of curved streamlines which are not equidistant as shown in Fig. 6.13. Let at any point A on the streamline the fluid particle has a velocity V. In small time dt the fluid particle moves through a distance δs and occupies new position at point B where the velocity becomes $(V + \delta V)$. The total change in velocity δV can be resolved into two components δV_s and δV_n along the tangential and the normal directions respectively. As shown in Fig. 6.13 from triangle AOB, $\delta s = rd\theta$ and from triangle BDE, $\delta V_n = Vd\theta$.

Thus

$$d\theta = \frac{\delta s}{r} = \frac{\delta V_n}{V}$$

or $$\frac{\delta V_n}{\delta s} = \frac{\partial V_n}{\partial s} = \frac{V}{r} = \frac{V_s}{r} \text{ (since} V = V_s)$$

in which r is the radius of curvature of the streamlines.

Substituting the value of $\left(\frac{\partial V_n}{\partial s}\right)$ in Eq. 6.31 the acceleration in the normal direction becomes

$$a_n = \frac{V_s^2}{r} + \frac{\partial V_n}{\partial t} \qquad ...(6.32)$$

Again in Eqs 6.30 and 6.31 or 6.32, $\left(\frac{\partial V_s}{\partial t}\right)$ and $\left(\frac{\partial V_n}{\partial t}\right)$ represent the rate of increase of velocities V_s and V_n with respect to time at a particular point in the flow and hence these are known as *local tangential acceleration* and *local normal acceleration* respectively. Similarly in these expressions

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\left(V_s \frac{\partial V_s}{\partial s}\right)$ and $\left(V_s \frac{\partial V_n}{\partial s}\right)$ $\left[\text{or } \frac{V_s^2}{r}\right]$ represent the rate of increase of velocities V_s and V_n due to the change in the position of the fluid particle and hence these are known as *convective tangential acceleration* and *convective normal acceleration* respectively.

For steady flow there is no variation of velocity with respect to time and hence there will be no local acceleration i.e.,

$$\frac{\partial V_s}{\partial t} = 0 \text{ and } \frac{\partial V_n}{\partial t} = 0$$

Therefore for steady flow there exists only convective acceleration and hence in such cases

$$a_s = V_s \frac{\partial V_s}{\partial s}$$

and

$$a_n = \frac{V_s^2}{r}$$

Further if the streamlines are straight lines then since $r = \infty$ the normal convective acceleration is zero. In other words, the normal convective acceleration is developed only if the fluid flows along a curved path, so that the streamlines are curved.

If the streamlines are straight and parallel to each other then even the tangential convective acceleration is also equal to zero. In other words, in such cases there is no acceleration. However if the streamlines are straight and converging then there will be tangential convective acceleration developed.

TABLE 6.1 *Types of Accelerations for Different Types of Streamline Patterns for Steady Flow*

S.No.	Streamline pattern	Type of acceleration
1	Straight parallel Streamlines	No acceleration
2	Straight converging Streamlines	Convective tangential acceleration
3	Concentric Streamlines	Convective normal acceleration
4	Curved converging Streamlines	Both tangential and normal convective acceleration

When the streamlines are curved and equidistant then the tangential convective acceleration is zero and there will be only normal convective acceleration developed. However if the streamlines are curved and converging then both normal and tangential convective accelerations will be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

developed. It may however be stated that if instead of converging streamlines there are diverging streamlines then instead of acceleration there will be corresponding retardation developed in the direction of flow.

Table 6.1 shows different streamline patterns and the types of accelerations for each of these patterns for the case of steady flow.

6.8 ROTATIONAL AND IRROTATIONAL MOTIONS

As shown in Fig. 6.14, a fluid particle as it moves can undergo the following four types of displacements viz.,

(i) Linear translation or pure translation.
(ii) Linear deformation.
(iii) Angular deformation.
(iv) Rotation.

A fluid particle is said to have undergone linear or pure translation if it moves bodily in such a way that the two axes *ab* and *cd* have their new positions represented by *a′b′* and *c′d′* which are parallel to the previous ones, as shown in Fig. 6.14 (a).

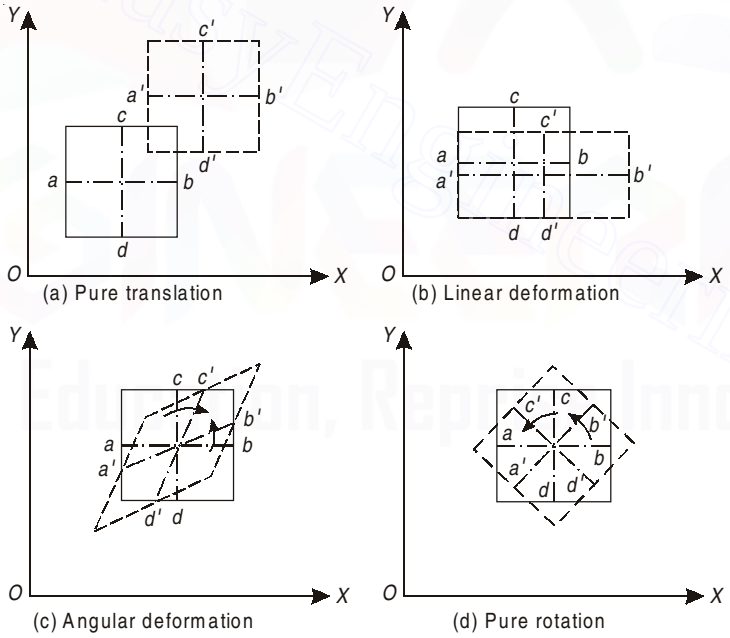


Figure 6.14 *Displacements of fluid particles*

A fluid particle is said to have undergone linear deformation if as it moves it gets deformed in the linear direction, so that two axes *a′b′* and *c′d′* of the deformed particle are parallel to the two axes *ab* and *cd* of the undeformed particle, as shown in Fig. 6.14 (b). It is thus observed that in the case of both linear translation and linear deformation the fluid particles are displaced parallel to their original position.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

A fluid particle is said to have undergone angular deformation if it deforms in such a way that the two axes *ab* and *cd* rotate by the same amount but in opposite directions with respect to their previous positions i.e., the clockwise rotation of the vertical axis *cd* is at the same rate as the counter-clockwise rotation of the horizontal axis *ab* as shown in Fig. 6.14 (c).

A fluid particle is said to have undergone rotation if its both the horizontal and the vertical axes rotate in the same direction as shown in Fig. 6.14 (d). When the fluid particles while moving in the direction of flow rotate about their mass centres then the flow is said to be rotational flow.

The rotation of a fluid element may be defined in terms of the components of rotation about three mutually perpendicular axes. Referring to Fig. 6.15 the mathematical expression for the rotation component about an axis parallel to *z*-axis is developed.

Let a fluid element at any point $P(x, y, z)$ has the velocity components u and v in the x and y directions respectively. Consider any two line segments *PA* and *PB* of lengths δx and δy, taken parallel to the x and y axes respectively for the sake of convenience. The velocity at A in the y direction will be $\left(v+\frac{\partial v}{\partial x}dx\right)$ and the velocity at B in the x direction will be $\left(u+\frac{\partial u}{\partial y}dy\right)$.

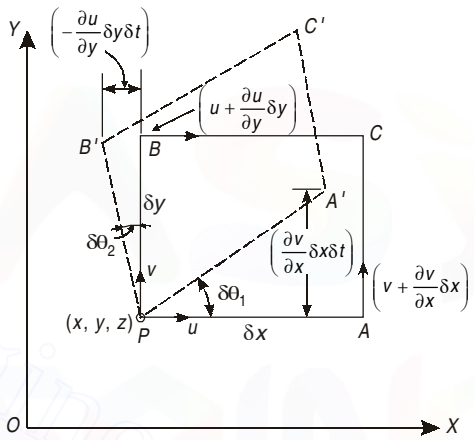


Figure 6.15 *Rotation of rectangular fluid element about z-axis.*

Since the velocities at P and A in the y direction are different, there will be an angular velocity developed for the linear element *PA*. Similarly the velocities at *P* and *B* in the x direction are different and hence there will be an angular velocity developed for the linear element *PB*.

Now if during a time interval of dt the elements *PA* and *PB* have moved, relative to *P*, to new positions *PA'* and *PB'* as indicated by the dotted lines, then the angular velocity (w_{PA}) of element *PA* about *Z* axis is

$$\omega_{P_A} = \lim_{\delta t\to 0}\frac{\delta\theta_1}{\delta t} = \lim_{\delta t\to 0}\frac{\left[\left(v+\frac{\partial v}{\partial x}\delta x\right)-v\right]\delta t}{\delta x\delta t} = \frac{\partial v}{\partial x}\ \text{rad/s.}$$

Similarly, the angular velocity (ω_{PB}) of element *PB* about *Z* axis is

$$\omega_{P_B} = \lim_{\delta t\to 0}\frac{\delta\theta_2}{\delta t} = \lim_{\delta t\to 0}\frac{-\left[\left(u+\frac{\partial u}{\partial y}\delta y\right)-u\right]\delta t}{\delta x\delta t} = -\frac{\partial u}{\partial x}\ \text{rad/s.}$$

The negative sign has been introduced because the motion in the anticlockwise direction has been considered as positive.

The rotation component about any axis may be defined as the average angular velocity of any two infinitesimal linear elements in the particle that are perpendicular to each other and to the axis of rotation (in this case it is *Z* axis).

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by the above definition the rotation component ω_z of a particle situated at point P is

$$\omega_z = \frac{1}{2}(\omega_{PA} + \omega_{PB}) = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \quad ...(6.33\ a)$$

By adopting the same procedure the rotation components about the x and y axes will be obtained as

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) \quad ...(6.33\ b)$$

and

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) \quad ...(6.33\ c)$$

If at every point in the flowing fluid the rotation components ω_x, ω_y and ω_z are equal to zero, then the flow is known as irrotational flow. Thus for a flow to be irrotational the following conditions must be satisfied throughout the flow field :

For $\omega_x = 0;\ \frac{\partial w}{\partial y} = \frac{\partial v}{\partial z}$ ...(6.34 a)

For $\omega_y = 0;\ \frac{\partial u}{\partial z} = \frac{\partial w}{\partial x}$ ...(6.34 b)

For $\omega_z = 0;\ \frac{\partial v}{\partial x} = \frac{\partial u}{\partial y}$ ...(6.34 c)

In vector notation the rotation of a fluid at a point is expressed as

$$\omega = \frac{1}{2}\ \text{curl}\ V = \frac{1}{2}\ \nabla \times V$$

where $\omega = i\omega_x + j\omega_y + k\omega_z$

Thus $\nabla \times V = i2\omega_x + j\,2\omega_y + k\,2\omega_z$

or $$\nabla \times V = i\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) + j\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) + k\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

The condition for the flow to be irrotational may be expressed as

$$\text{curl}\ V = \nabla \times V = 0$$

which is equivalent to the condition expressed by Eq. 6.34.

The rotation of a fluid particle is always associated with shear stress, because the rotation can be caused only by a torque exerted on the fluid particle and this will be produced by the shear forces. As such in the case of flow of fluids having larger viscosity or in the regions of flow field where the viscosity of the fluid has predominance the flow is invariably rotational flow. However, in the case of fluids such as air or water having small viscosity the flow in the region away from the boundary may for all practical purposes be treated as irrotational. Moreover, in the case of rapidly coverging or accelerating flows the flow may be treated as irrotational. The consideration of an irrotational flow in general leads to a simplified analysis of fluid flow problems.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

6.9 CIRCULATION AND VORTICITY

The flow along a closed curve is called *circulation* (i.e., the flow in eddies and vortices). The mathematical concept of circulation is the line integral, taken completely around a closed curve, of the tangential component of the velocity vector. Consider a closed curve *C* as shown in Fig. 6.16 (a), and let at any point on the curve the velocity of flow of fluid be *V*. If α is the angle between a small element *ds* along the curve in the tangential direction and the velocity *V*, then the component of the velocity in the direction tangential to the curve is $V \cos \alpha$. By the definition the circulation Γ (Greek, capital 'gamma') around a closed curve *C* is

$$\Gamma = \int_C V \cos \alpha \, ds \qquad ...(6.35)$$

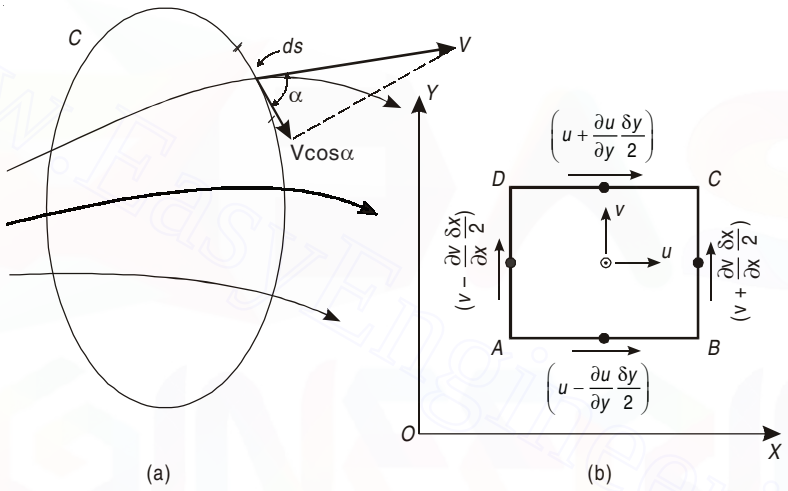


Figure 6.16 *Circulation around (a) a closed curve; (b) around an elementary rectangle, in the plane of a two-dimensional steady flow field*

Further if *u*, *v* and *w* are the components of velocity *V*, and *dx*, *dy* and *dz* are the components of the displacement *ds*, then the circulation can also be written as

$$\Gamma = \int_C (u dx + v dy + w dz) \qquad ...(6.35\,a)$$

The circulation around an elementary rectangle with sides parallel to the axes *x* and *y* as shown in Fig. 6.16 (b) may be written as follows:

$$\text{Circulation along } AB = \left(u - \frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x$$

$$\text{Circulation along } BC = \left(v + \frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y$$

$$\text{Circulation along } CD = -\left(u + \frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Circulation along $DA = -\left(v - \frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y$

The positive sense of integration is such that the enclosed surface is on left when viewed from the side of the outward normal.

Further whatever be the shape of the curve, the circulation around the periphery of the curve, must equal the sum of the circulation around the elementary surfaces of which it consists, provided the boundary of the curve is wholly in the fluid. Thus in this case the total circulation will be given by

$$\Gamma = \Gamma_{AB} + \Gamma_{BC} + \Gamma_{CD} + \Gamma_{DA}$$

$$= \left(u - \frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x + \left(v + \frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y - \left(u + \frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x - \left(v - \frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y$$

$$= \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\delta x\, \delta y \qquad ...(6.36)$$

The *vorticity* at any point is defined as the ratio of the circulation around an infinitesimal closed curve at that point to the area of the curve, i.e., it is defined as circulation per unit area. Thus from Eq. 6.36, the vorticity ζ (Greek 'zeta') may be expressed as

$$\zeta = \frac{\text{Circulation}}{\text{Area}}$$

$$= \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \qquad ...(6.37)$$

By comparing Eqs 6.37 and 6.33 (a), we get

$$\zeta = 2\omega_z \qquad ...(6.38)$$

that is vorticity (or circulation per unit area) is equal to twice the rotation component about an axis perpendicular to the plane in which the area is lying. Vorticity is a vector quantity whose direction is perpendicular to the plane of the small curve round which the circulation is measured. Thus in a general case of three-dimensional flow, Eq. 6.38 represents only a component of vorticity in the z direction, i.e., $\zeta_z = 2\omega_z$. Similarly the other two components of vorticity may also be obtained as $\zeta_x = 2\omega_x$ and $\zeta_y = 2\omega_y$. If the vorticity is zero at all points in a region then the flow in that region is said to be *irrotational*. On the other hand flow in regions where the vorticity is other than zero is said to be *rotational*.

The above results have been derived by considering a rectangular curve. But it may be stated that the results obtained in Eqs 6.36 and 6.38 are independent of the shape of the closed curve considered, and the rectangular curve has been chosen only for the sake of simplicity.

In vector notation circulation Γ may be expressed as

$$\Gamma = \iint_A \text{curl } V.\, dA$$

Similarly vorticity ζ may also be expressed in vector notation as

$$\zeta = \text{curl } V = \nabla \times V$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

6.10 VELOCITY POTENTIAL

The velocity potential ϕ (Greek 'phi') is defined as a scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction. Thus mathematically the velocity potential is defined as $\phi = f(x, y, z, t)$ for unsteady flow and $\phi = f(x, y, z)$ for steady flow such that

$$\left.\begin{aligned} u &= -\frac{\partial \phi}{\partial x} \\ v &= -\frac{\partial \phi}{\partial y} \\ w &= -\frac{\partial \phi}{\partial z} \end{aligned}\right\} \quad \ldots(6.39)$$

where u, v and w are the components of velocity in the x, y and z directions respectively. The negative sign signifies that ϕ decreases with an increase in the values of x, y and z. In other words it indicates that the flow is always in the direction of decreasing ϕ.

For an imcompressible fluid if the flow is steady then equation of continuity is given by Eq. 6.6 as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

By substituting the values of u, v and w in terms of ϕ from Eq. 6.39, we get

$$\frac{\partial}{\partial x}\left(-\frac{\partial \phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(-\frac{\partial \phi}{\partial y}\right) + \frac{\partial}{\partial z}\left(-\frac{\partial \phi}{\partial z}\right) = 0$$

or

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0 \quad \ldots(6.40)$$

This equation is known as *Laplace Equation*, which may be expressed in vector notation as

$$\nabla^2 \phi = 0$$

It is evident that any function ϕ that satisfies the Laplace equation will correspond to some case of fluid flow.

Further for a rotational flow the rotaion components are given by Eq. 6.33 as

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)$$

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

and

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substituting the values of u, v and w in terms of ϕ from Eq. 6.39, we get

$$\omega_x = \frac{1}{2}\left[-\frac{\partial^2\phi}{\partial y\partial z} + \frac{\partial^2\phi}{\partial z\partial y}\right]$$

$$\omega_y = \frac{1}{2}\left[-\frac{\partial^2\phi}{\partial z\partial x} + \frac{\partial^2\phi}{\partial x\partial z}\right]$$

and

$$\omega_z = \frac{1}{2}\left[-\frac{\partial^2\phi}{\partial x\partial y} + \frac{\partial^2\phi}{\partial y\partial x}\right]$$

However, if ϕ is a continuous function then

$$\frac{\partial^2\phi}{\partial y\partial z} = \frac{\partial^2\phi}{\partial z\partial y};\ \frac{\partial^2\phi}{\partial z\partial x} = \frac{\partial^2\phi}{\partial x\partial z} \text{ and } \frac{\partial^2\phi}{\partial x\partial y} = \frac{\partial^2\phi}{\partial y\partial x}$$

according to which $\omega_x = \omega_y = \omega_z = 0$ i.e., the flow is irrotational.

Therefore it may be stated that any function ϕ that satisfies Laplace equation is a possible irrotational flow case since the continuity is satisfied. In other words velocity potential exists only for irrotational flows of fluids. Hence often an irrotational flow is known as potential flow.

For two-dimensional flows the cylindrical polar coordinate system (with $z = 0$) is useful in several problems. As such the expressions for the velocity components in terms of the velocity potential can also be obtained in the cylindrical polar coordinates. Thus if V_r and V_θ are the velocity compoents in r and θ directions respectively then these can be expressed as

$$V_r = -\frac{\partial\phi}{\partial r};\ \text{and } V_\theta = -\frac{\partial\phi}{r\partial\theta}$$

By substituting the values of V_r and V_θ in the continuity Eq. 6.18, we get

$$\frac{\partial\left(-\frac{\partial\phi}{\partial r}r\right)}{r\partial r} + \frac{\partial\left(-\frac{\partial\phi}{r\partial\theta}\right)}{r\partial\theta} = 0$$

or

$$\frac{1}{r}\frac{\partial\phi}{\partial r} + \frac{\partial^2\phi}{\partial r^2} + \frac{1}{r^2}\frac{\partial^2\phi}{\partial\theta^2} = 0$$

which is Laplace equation in cylindrical polar coordinates for two-dimensional flows.

6.11 STREAM FUNCTION

The stream function ψ (Greek 'psi') is defined as a scalar function of space and time, such that its partial derivative with respect to any direction gives the velocity component at right angles (in the counter-clockwise direction) to this direction. For two-dimensional flow and three-dimensional flow with axial symmetry (i.e., axially symmetric flow) separate stream functions are considered. The stream function for the case of two-dimensional flow only is considered here. Thus mathematically stream function may be defined as $\psi = f(x, y, t)$ for unsteady flow and $\psi = f(x, y)$ for steady flow, such that

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left.\begin{aligned}\frac{\partial \psi}{\partial x} &= v \\ \frac{\partial \psi}{\partial y} &= -u\end{aligned}\right\} \qquad \ldots(6.41)$$

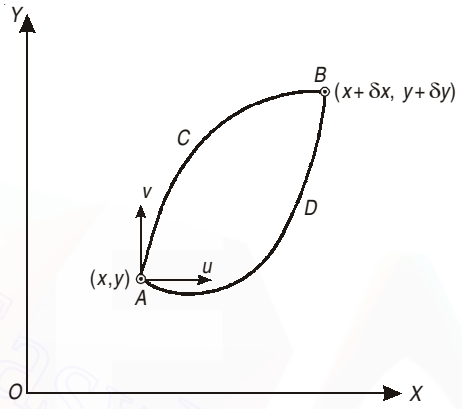


Figure 6.17 *Flow across curves between any two points in the fluid region*

The values of the velocity components u and v as represented by Eq. 6.41 may be derived as explained below.

Consider two points A and B with coordinates (x, y) and $(x + \delta x, y + \delta y)$ respectively, which are joined by any two arbitrary curves ACB and ADB as shown in Fig. 6.17. Let u and v be the velocity components at point A in the x and y directions respectively. Then the flow across the curve ACB in the x direction is equal to $(-u\delta y)$ and in the y direction it is equal to $(v\delta x)$. In the computation of the flows across the curve ACB the sign convention that is assumed is that as the observer views from A to B then the flow from left to right (or in the clockwise direction) is considered as negative and the flow from right to left (or in the counter-clockwise direction) is considered as positive. Now if $d\psi$ represents the total flow across the curve ACB then

$$d\psi = -u\delta y + v\delta x \qquad \ldots(6.42)$$

It may however be stated that if the fluid is homogeneous and incompressible, then the flow across ADB or any other curve must be the same as that across the curve ACB.

Further as stated earlier for steady flow, $\psi = f(x, y)$, the total derivative of ψ may be expressed as

$$d\psi = \frac{\partial \psi}{\partial x}\,\delta x + \frac{\partial \psi}{\partial y}\,\delta y \qquad \ldots(6.43)$$

Comparing the Eqs 6.42 and 6.43, we get

$$\frac{\partial \psi}{\partial x} = +v \text{ and } \frac{\partial \psi}{\partial y} = -u$$

which is same as Eq. 6.41. Similarly in terms of cylindrical polar coordinates if V_r and V_θ are the components of velocity in the r and θ directions respectively, then by adopting the same procedure the following expressions may be obtained

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{\partial \psi}{r\partial \theta} = -V_r \text{ and } \frac{\partial \psi}{\partial r} = V_\theta$$

By comparing Eqs 6.39 and 6.41 the relationship between u, v and ϕ and ψ are obtained as

$$-\frac{\partial \phi}{\partial x} = u = -\frac{\partial \psi}{\partial y} \text{ and } -\frac{\partial \phi}{\partial y} = v = \frac{\partial \psi}{dx}$$

That is

$$\left.\begin{aligned} \frac{\partial \phi}{\partial x} &= \frac{\partial \psi}{\partial y} \\ -\frac{\partial \phi}{\partial y} &= \frac{\partial \psi}{\partial x} \end{aligned}\right\} \quad \text{...(6.44)}$$

These equations are known as Cauchy–Rieman equations and they enable the computation of stream function if the velocity potential is known and vice-versa in a potential flow.

By substituting the values of the velocity, components u and v from Eqs 6.41 in Eq. 6.33(a) the rotation component ω_z becomes

$$\omega_z = \frac{1}{2}\left[\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial x}\right) - \frac{\partial}{\partial y}\left(-\frac{\partial \psi}{\partial y}\right)\right]$$

or

$$\omega_z = \frac{1}{2}\left[\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2}\right] \quad \text{...(6.45)}$$

Equation 6.45 is known as Poisson's equation. For an irrotational flow since $\omega_z = 0$, Eq. 6.45 becomes

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

which is Laplace equation for ψ.

Further by substituting the values of u and v from Eq. 6.41 in Eq. 6.15 which is the equation of continuity for two dimensional steady flow of an incompressible fluid, we get

$$\frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial y}\right) + \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial x}\right) = 0$$

or

$$\frac{\partial^2 \psi}{\partial x \partial y} = \frac{\partial^2 \psi}{\partial y \partial x}$$

This will be true if ψ is a continuous function and its second derivative exists. Therefore it may be stated that any function ψ which is continuous is a possible case of fluid flow (which may be rotational or irrotational) since the equation of continuity is satisfied.

However, if the function ψ is such that it satisfies Laplace equation then it is a possible case of an irrotational flow, as indicated above.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further the expressions for the velocity components in terms of stream function can also be obtained in the cylindrical polar coordinates. Thus, if V_r and V_θ are the velocity components in r and θ directions respectively then these can be expressed as

$$V_r = -\frac{\partial \psi}{r \partial \theta};$$

and

$$V_\theta = \frac{\partial \psi}{\partial r}$$

The Cauchy–Rieman equations in cylindrical polar coordinates may thus be expressed as

$$\frac{\partial \phi}{\partial r} = \frac{\partial \psi}{r \partial \theta}$$

and

$$\frac{\partial \phi}{r \partial \theta} = -\frac{\partial \psi}{\partial r}$$

6.12 STREAMLINES, EQUIPOTENTIAL LINES AND FLOW NET

A property of the stream function is that the difference of its values at two points represents the flow across any line joining the points. Therefore, when two points lie on the same streamline, then since there is no flow across a streamline, the difference between the stream functions ψ_1 and ψ_2 at these two points is equal to zero, i.e., $(\psi_1 - \psi_2) = 0$. In other words, it means that *streamline* is given by ψ = constant.

Similarly ϕ = constant, represents a curve for which the velocity potential is same at every point, and hence it represents an *equipotential line.*

Consider two curves, viz., ϕ = constant and ψ = constant, intersecting each other at any point. The slopes of these curves at the point of intersection may be determined as indicated below.

For the curve ϕ = constant,

$$\text{Slope} = \frac{\partial y}{\partial x} = \frac{\left(\frac{\partial \phi}{\partial x}\right)}{\left(\frac{\partial \phi}{\partial y}\right)} = \frac{-u}{-v} = \frac{u}{v}$$

Similarly for the curve ψ = constant,

$$\text{Slope} = \frac{\partial y}{\partial x} = \frac{\left(\frac{\partial \psi}{\partial x}\right)}{\left(\frac{\partial \psi}{\partial y}\right)} = \frac{v}{-u} = -\frac{v}{u}$$

Therefore the product of the slopes of these two curves at the point of intersection = –1, which indicates that these two sets of curves, viz., streamlines and equipotential lines intersect each other orthogonally at all points of intersection.

A grid obtained by drawing a series of streamlines and equipotential lines is known as a *flow net*. A flow net may be drawn for a two-dimensional irrotational flow and it provides a simple, yet valuable

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

indication of the flow pattern. Figure 6.18 shows the elements of a flow net which has been obtained by drawing a set of curves corresponding to $\psi = C_1, C_2, C_3$ etc., and $\phi = C_1, C_2, C_3$ etc. Let at one of the intersections V_s and V_n be the velocity components in the directions tangential to the curves ψ = constant, and ϕ = constant, respectively.

At any point in the direction n along the equipotential line since ϕ = constant, C_2

$$\frac{\partial \phi}{\partial n} = 0$$

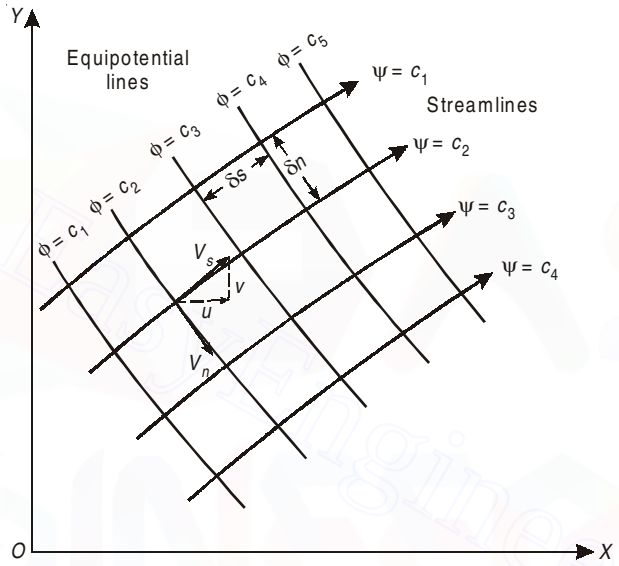


Figure 6.18 *Elements of a flow net*

But according to the definition of the velocity potential

$$-\frac{\partial \phi}{\partial n} = V_n$$

$$\therefore \quad V_n = 0$$

Further $$-\frac{\partial \phi}{\partial s} = V_s$$

The relations therefore indicate that there is no flow along the direction tangential to the equipotential lines, but the flow always takes place in the direction at right angles to the equipotential lines.

Similarly at the point shown in Fig. 6.18, by definition

$$\frac{\partial \psi}{\partial s} = V_n = 0$$

and $$\frac{\partial \psi}{\partial n} = -V_s$$

These relations also indicate that there is no flow in the direction normal to the streamlines but the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

flow is always along the direction tangential to the streamlines.

Moreover, since the flow is along the direction tangential to the streamlines, the space between two adjacent streamlines may be considered as a flow channel and the discharge flowing through it is proportional to $(\psi_2 - \psi_1)$, where ψ_1 and ψ_2 are the values of the stream functions for these two adjacent streamlines.

6.13 METHODS OF DRAWING FLOW NETS

The construction of the flow net for the flows is restricted by certain conditions which are as indicated below :

(i) The flow should be steady. This is so because it is only for the steady flow the streamline pattern will remain constant. For unsteady flow the streamline pattern will be instantaneous and it may change from instant to instant.

(ii) The flow should be irrotational, which is possible if the flowing fluid is an ideal fluid (having no viscosity) or it has negligible viscosity. However, in case of rapidly accelerating or converging flow of fluids, even if the fluids have low viscosity, the flow net analysis may be adopted.

(iii) The flow is not governed by the gravity force. This is so because under the action of gravity the shape of the free surface is constantly changing and with the shape of the extreme boundary surface (free surface in this case) undergoing a change, no fixed flow net pattern can be obtained. However, in such cases the flow nets can be drawn after fixing the correct shape of the free surface boundary.

The following are the different methods used for drawing the flow nets :

(1) Analytical Method.
(2) Graphical Method.
(3) Electrical Analogy Method.
(4) Relaxation Method.
(5) Hele Shaw or Viscous flow Analogy Method.

Out of these five methods, the first three methods are briefly described below.

Analytical Method. In this method the equations corresponding to the curves ϕ and ψ are first obtained and the same are plotted to give the flow net pattern for the flow of fluid between the given boundary shapes. In other words, this method involves a solution of Laplace equation for ϕ and ψ, which gives the corresponding equations for ϕ and ψ. But this method cannot be applied in various cases on account of the boundary shapes being such that it may not be possible to obtain the solution of the Laplace equation for ϕ and ψ. In such cases, other methods may be adopted to obtain the flow net pattern for the corresponding flow.

Graphical Method. The graphical method may be used to draw a flow net for the flow of fluid between the boundaries of any shape. The fixed solid boundaries correspond to streamlines, since they have no flow across them. In between these extreme streamlines (i.e., fixed boundaries) a number of other streamlines are suitably sketched by guess work. A set of smooth equipotential lines is then drawn so as to intersect the streamlines (including the fixed boundaries) perpendicularly and so spaced that streamlines and equipotential lines form approximate squares throughout the entire network. Successive adjustments of both the streamlines and the equipotential lines are then made to fulfil the above requirements. When the complete flow net or any part of it appears to consist of 'squares', a check can be made by drawing both diagonals of each 'square'. These diagonals should

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

also form continuous curves and should themselves intersect to form 'squares'. Another check which may also be used to test the accuracy of the flow net is that in each 'square' the sides are tangent to a common circle inscribed in it.

Electrical Analogy Method. It is a practical method of drawing a flow net for a particular set of boundaries, which is based on the fact that the flow of fluids and the flow of electricity through a conductor are analogous. These two systems are similar in this respect that the electric potential is analogous to the velocity potential, the electric current is analogous to the velocity of flow, and the homogeneous conductor is analogous to the homogeneous fluid.

As shown in Fig. 6.19, the fixed boundaries of the model are formed out of strips of non-conducting material mounted on a flat non-conducting surface. The end equipotential lines are formed out of conducting strips e.g. copper or brass. An electrolyte (conducting liquid) is placed at uniform depth

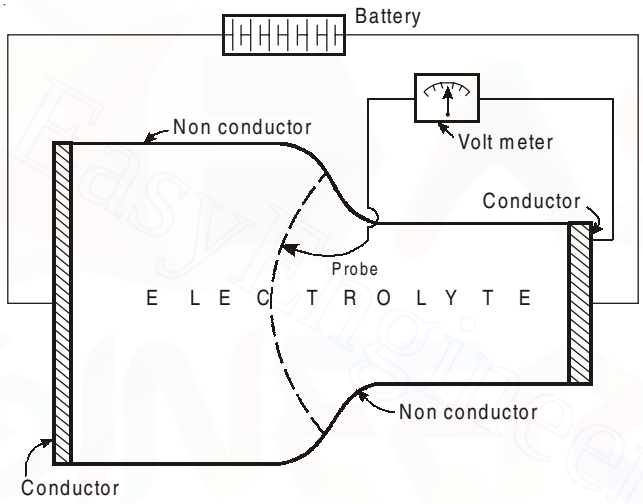


Figure 6.19 *Electrical analogy circuit*

in the flow space. A voltage potential is applied to the two ends of the conducting strips by means of a battery system or by connecting them to the electric mains. By means of a probe and a voltmeter, lines with constant drop in voltage from one end are located which may be plotted. These are equipotential lines. Once the exact pattern of equipotential lines are obtained, the streamlines can be drawn by graphical method. However, by reversing the process and making the flow boundaries out of the conducting material and the end equipotential lines from non-conducting material, the streamlines can also be sketched by this method.

6.14 USE OF THE FLOW NET

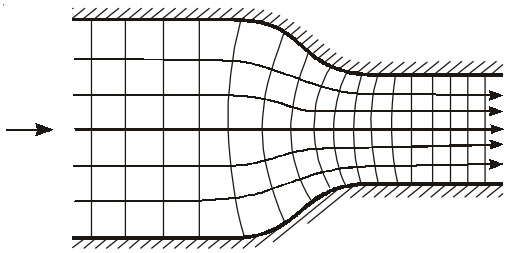
For a given set of boundary configuration there is only one possible pattern of the flow of an ideal fluid, and a correctly drawn flow net will represent this pattern. As such after a flow net for a given boundary configuration has been obtained, it may be used for all irrotational flows with geometrically similar boundaries. Once the flow net is drawn, the spacing between the adjacent streamlines is determined and the application of the continuity equation gives the velocity of flow at any point, if the

Downloaded From : www.EasyEngineering.net

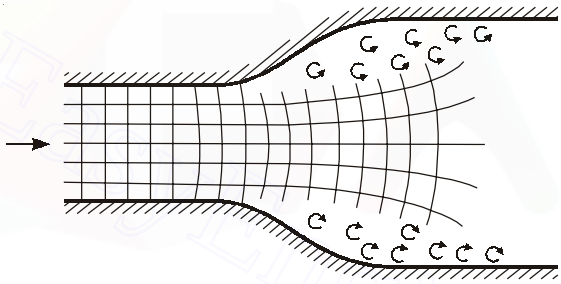
Downloaded From : www.EasyEngineering.net

velocity of flow at any reference point is known.

Further the flow net analysis assists in the determination of the efficient boundary shapes, for which the flow does not separate from the boundary surface.



(a) Flow through converging boundaries



(b) Flow through diverging boundaries

Figure 6.20 *Typical flow nets*

Although the flow net analysis is based on an ideal fluid flow concept, it may also be applied to the flow of a real fluid within certain limit. The ideal fluid theory neglects the effect of fluid friction or viscosity, which is however possessed by a real fluid. But the viscosity effects of a real fluid are most pronounced at or near a solid boundary and diminish rapidly with distance from the boundary. As such in the regions where viscosity effects are not predominant, the real fluid behaves more or less like an ideal fluid, and in these regions the flow net analysis may be applied to the real fluids with sufficient accuracy.

Figure 6.20 shows a few examples of the flow nets. It is observed in Fig. 6.20 (a) that since the boundaries are converging the streamlines also converge rapidly. In this case the accelerating flow is developed and the actual flow pattern approximates closely to that represented by the flow net. On the other hand as observed in Fig. 6.20 (b) the boundaries in the direction of flow are diverging and therefore the streamlines also tend to diverge. In the region where the streamlines diverge, a phenomenon known as separation of flow generally occurs. That is, in such cases the flowing fluid does not remain in contact with the boundary surface or it separates from the boundary, thereby developing regions of flow separation in which eddies are developed. Therefore, in such cases where the flow separation takes place, the flow net which is constructed with streamlines conforming to the boundaries does not describe the actual pattern of flow field.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

6.15 LIMITATIONS OF FLOW NET

A flow net always indicates some velocity at the boundary, but a real fluid must have zero velocity adjacent to the boundary on account of the fluid friction or viscosity. As such the flow net analysis cannot be applied in the region close to the boundary where the effects of viscosity are predominant. As stated earlier the flow net analysis can also not be applied to a sharply diverging flow, since the actual flow pattern is not represented by the flow net. Further in the case of flow of a fluid past a solid body, while the flow net gives a fairly accurate picture of the flow pattern for the upstream part of the solid body, it can give little information concerning the flow conditions at the rear because of separation and eddies. The disturbed flow in the rear of the solid body is known as wake, the formation of which is not indicated by a flow net.

ILLUSTRATIVE EXAMPLES

Example 6.1. *An airplane is observed to travel due north at a speed of 240 km per hour in a 80 km per hour wind from the north-west. What is the apparent wind velocity observed by the pilot ?*

Solution

Let V_R be the apparent wind velocity observed by the pilot. Now if V_W and V_P are the velocities of the wind and the plane respectively, then

$$V_W = V_R + \rightarrow V_P$$

$$\therefore \quad V_R = \sqrt{(240 + 80\sin 45°)^2 + (80\cos 45°)^2}$$

$$= 301.9 \text{ km/hour}$$

If the angle between V_R and V_P is θ, then

$$\tan\theta = \frac{80\cos 45°}{240 + 80\sin 45°} = 0.191$$

$$\therefore \quad \theta = 10.8°$$

Thus the apparent wind direction is N 10.8° W.

Example 6.2. *When 2500 litres of water flows per minute through a 0.3 m diameter pipe which later reduces to a 0.15 diameter pipe, calculate the velocities of flow in the two pipes.*

Solution

From Eq. 6.24 discharge

$$Q = A_1V_1 = A_2V_2$$

$$Q = \frac{2500}{60\times10^3} = 0.042 \text{ m}^3/\text{s}$$

$$A_1 = \frac{\pi}{4} \times (0.3)^2 = 0.0707 \text{ m}^2$$

$$A_2 = \frac{\pi}{4} \times (0.15)^2 = 0.0177 \text{ m}^2$$

Thus $$V_1 = \frac{Q}{A_1} = \frac{0.042}{0.0707} = 0.59 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$V_2 = \frac{Q}{A_2} = \frac{0.042}{0.0177} = 2.37 \text{ m/s}$$

Example 6.3. *Two velocity components are given in the following cases, find the third component such that they satisfy the continuity equation.*

(a) $u = x^3 + y^2 + 2z^2$; $v = -x^2y - yz - xy$;

(b) $u = \log (y^2 + z^2)$; $v = \log (x^2 + z^2)$;

(c) $u = \dfrac{-2xyz}{(x^2 + y^2)^2}$; $w = \dfrac{y}{(x^2 + y^2)}$

Solution

For an incompressible fluid the equation of continuity is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

(a) In this case

$$u = x^3 + y^2 + 2z^2$$

$\therefore$ $$\frac{\partial u}{\partial x} = 3x^2$$

and $$v = -x^2y - yz - xy$$

$$\frac{\partial v}{\partial y} = -x^2 - z - x$$

$\therefore$ By substitution in the equation of continuity, we get

$$3x^2 - x^2 - z - x + \frac{\partial w}{\partial z} = 0$$

or $$\frac{\partial w}{\partial z} = x + z - 2x^2$$

or $$\partial w = (x + z - 2x^2)\partial z$$

By integrating both sides, we get

$$w = \left(xz + \frac{z^2}{2} - 2x^2 z \right) + \text{constant of integration}$$

The constant of integration could be a function of x and y, that is $f(x, y)$. Hence the third component is

$$w = \left(xz + \frac{z^2}{2} - 2x^2 z \right) + f(x, y).$$

(b) In this case

$$u = \log (y^2 + z^2)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad \frac{\partial u}{\partial x} = 0$$

Also $v = \log (x^2 + z^2)$

$$\therefore \quad \frac{\partial v}{\partial y} = 0$$

By substitution in the equation of continuity, we get

$$\frac{\partial w}{\partial z} = 0$$

By integration, we get

$$w = f(x, y)$$

By symmetry one of the values of the third component can be

$$w = \log (x^2 + y^2)$$

(c) In this case

$$u = \frac{-2xyz}{\left(x^2 + y^2\right)^2}$$

$$\therefore \quad \frac{\partial u}{\partial x} = \frac{(x^2 + y^2)^2(-2yz) - (-2xyz) \times 2(x^2 + y^2) \times 2x}{(x^2 + y^2)^4} = \frac{6x^2yz - 2y^3z}{(x^2 + y^2)^3}$$

Also $$w = \frac{y}{(x^2 + y^2)}$$

$$\therefore \quad \frac{\partial w}{\partial z} = 0$$

By substituting in the continuity equation, we get

$$\frac{6x^2yz - 2y^3z}{(x^2 + y^2)^3} + \frac{\partial v}{\partial y} + 0 = 0$$

or $$\frac{\partial v}{\partial y} = \frac{2y^3z - 6x^2yz}{(x^2 + y^2)^3}$$

or $$\partial v = \frac{2y^3z - 6x^2yz}{(x^2 + y^2)^3} \partial y$$

By integrating both sides, we get

$$v = \frac{z(x^2 - y^2)}{(x^2 + y^2)^2} + f(x, z).$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 6.4. *The velocity components in a two-dimensional flow field for an incompressible fluid are expressed as* $u = \frac{y^3}{3} + 2x - x^2y$; $v = xy^2 - 2y - \frac{x^3}{3}$

(*a*) *Show that these functions represent a possible case of an irrotational flow.*

(*b*) *Obtain an expression for stream function* ψ.

(*c*) *Obtain an expression for velocity potential* ϕ.

Solution

(a) The partial derivatives of the given functions are

$$\frac{\partial u}{\partial x} = 2 - 2xy \; ; \; \frac{\partial v}{\partial y} = 2xy - 2$$

For a two-dimensional flow of incompressible fluid the continuity equation may be expressed as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Thus by substitution, we get

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + = 2 - 2xy + 2xy - 2 = 0$$

Therefore the functions represent a possible case of fluid flow.

The rotation component ω_z of any fluid element in the flow field is

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} \frac{\partial u}{\partial y}\right)$$

Now $$\frac{\partial v}{\partial x} = (y^2 - x^2)$$

and $$\frac{\partial u}{\partial y} = (y^2 - x^2)$$

Hence by substitution, we get

$$\omega_z = [(y^2 - x^2) - (y^2 - x^2)] = 0$$

which shows that the given functions represent an irrotational flow.

(b) From Eq. 6.41

$$\frac{\partial \psi}{\partial x} = v = xy^2 - 2y - \frac{x^3}{3} \qquad ...(i)$$

and $$\frac{\partial \psi}{\partial y} = -u = -\left(\frac{y^3}{3} + 2x - x^2y\right) \qquad ...(ii)$$

Integrating Eq. (i), we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\psi = \frac{x^2y^2}{2} - 2xy - \frac{x^4}{12} + f(y) \quad \ldots(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \psi}{\partial y} = x^2y - 2x + f'(y) \quad \ldots(iv)$$

Equating the values of $\left(\frac{\partial \psi}{\partial y}\right)$ from Eqs (ii) and (iv), we get

$$-\left(\frac{y^3}{3} + 2x - x^2y\right) = x^2y - 2x + f'(y)$$

or $$f'(y) = -\frac{y^3}{3}$$

By integrating both sides of the above expression, we get

$$f'(y) = -\frac{y^4}{12} + C$$

where C is a constant of integration which is a numerical constant only. Therefore

$$\psi = \frac{x^2y^2}{2} - 2xy - \frac{x^4}{12} - \frac{y^4}{12} + C$$

Since C is numerical constant, it may also be considered as zero, in which case

$$\psi = \frac{x^2y^2}{2} - 2xy - \frac{x^4}{12} - \frac{y^4}{12}$$

(c) From Eq. (6.39)

$$-\frac{\partial \phi}{\partial x} = u = +2x - x^2y \quad \ldots(i)$$

$$-\frac{\partial \phi}{\partial y} = v = xy^2 - 2y - \frac{x^3}{3} \quad \ldots(ii)$$

Integrating Eq. (i), we get

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3y}{3} + f(y) \quad \ldots(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \phi}{\partial y} = -xy^2 + \frac{x^3}{3} + f'(y) \quad \ldots(iv)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equating the values of $\left(\frac{\partial \phi}{\partial y}\right)$ from Eqs (ii) and (iv), we get

$$-xy^2 + 2y + \frac{x^3}{3} = -xy^2 + \frac{x^3}{3} + f'(y)$$

or $$f'(y) = 2y$$

By integrating both sides of the above expression, we get

$$f(y) = y^2 + C$$

where C is a numerical constant of integration.

Therefore

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3 y}{3} + y^2 + C$$

Again since C is a numerical constant, it may be considered as zero and hence

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3 y}{3} + y^2.$$

Example 6.5. *For a three-dimensional flow field described by*

$$V = (y^2 + z^2)\,\mathbf{i} + (x^2 + z^2)\,\mathbf{j} + (x^2 + y^2)\,\mathbf{k}$$

find at (1, 2, 3) (i) the components of acceleration, (ii) the components of rotation.

Solution

The components of velocity u, v, and w are given by

$$u = y^2 + z^2; \quad v = x^2 + z^2; \quad w = x^2 + y^2$$

$$\frac{\partial u}{\partial x} = 0 \qquad \frac{\partial v}{\partial x} = 2x \qquad \frac{\partial w}{\partial x} = 2x$$

$$\frac{du}{\partial y} = 2y \qquad \frac{\partial v}{\partial y} = 0 \qquad \frac{\partial w}{\partial y} = 2y$$

$$\frac{\partial u}{\partial z} = 2z \qquad \frac{\partial v}{\partial z} = 2z \qquad \frac{\partial w}{\partial z} = 0$$

(i) Introducting the above expressions in Eqs 6.27, 6.28 and 6.29 the components of acceleration are obtained as follows.

$$a_x = 2y(x^2 + z^2) + 2z(x^2 + y^2) = 70$$
$$a_y = 2x(y^2 + z^2) + 2z(x^2 + y^2) = 56$$
$$a_z = 2x(y^2 + z^2) + 2y(x^2 + z^2) = 66$$

(ii) Introducting the above expression in Eq. 6.33 the components of rotation are obtained as follows.

$$\omega_x = \frac{1}{2}(2y - 2z) = -1 \text{ rad/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\omega_y = \frac{1}{2}(2z - 2x) = 2 \text{ rad/s}$$

$$\omega_z = \frac{1}{2}(2x - 2y) = -1 \text{ rad/s}$$

Example 6.6. *For a two dimensional flow* $\phi = 3xy$ *and* $\psi = \frac{3}{2}(y^2 - x^2)$ *. Determine the velocity components at the points (1, 3) and (3, 3). Also find the discharge passing between the streamlines passing through the points given above.*

Solution

From Eq. 6.39, we have

$$u = -\frac{\partial \phi}{\partial x} \text{ ; and } v = -\frac{\partial \phi}{\partial y}$$

Since $\phi = 3xy$, we obtain

$$u = -3y \text{ : and } v = -3x$$

Thus at point (1, 3) velocity components are

$$u = -9 \text{ ; and } v = -3$$

and at point (3, 3), velocity components are

$$u = -9 \text{ ; and } v = -9$$

Alternatively from Eq. 6.41, we have

$$u = -\frac{\partial \psi}{\partial y} \text{ ; and } v = \frac{\partial \psi}{\partial x}$$

Since $\psi = \frac{3}{2}(y^2 - x^2)$, we obtain

$$u = -3y \text{ ; and } v = -3x$$

which are same as obtained above.

The value of ψ for streamline passing through point (1, 3) is

$$\psi_1 = 12$$

Similarly the value of ψ for streamline passing through point (3, 3) is

$$\psi_2 = 0$$

$\therefore$ Discharge passing between these two streamlines is

$$(\psi_1 - \psi_2) = (12 - 0) = 12$$

Example 6.7. *The stream function* $\psi = 4xy$ *in which* ψ *is in* cm^2 *per second and x and y are in metres describe the incompressible flow between the boundary shown below:*

Calculate

(i) Velocity at B.

(ii) Convective acceleration at B.

(iii) Flow per unit width across AB.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

The stream function is given as

$$\psi = 4xy$$

in which ψ is in cm^2/s and x and y are in m. Thus 4 is a dimensional constant having its dimensions as $\left(\frac{cm^2}{s \times m^2}\right)$.

From Eq. 6.41, we have

$$\frac{\partial \psi}{\partial x} = v = 4y$$

and

$$\frac{\partial \psi}{\partial y} = -u = 4x$$

(i) For point B, $x = 3$ m and $y = 1$ m, and hence at this point

$$u = -\frac{4 \times 3}{100} = -0.12 \text{ cm/s}$$

and

$$v = \frac{4 \times 1}{100} = 0.04 \text{ cm/s}$$

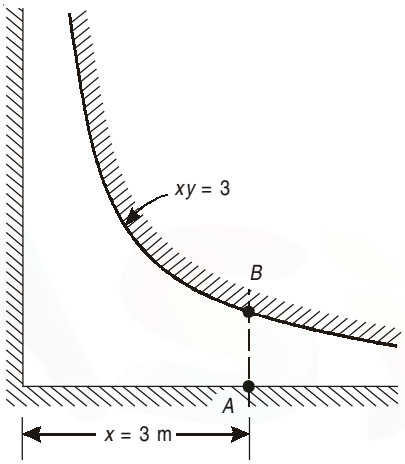


Figure Ex. 6.7

$\therefore$ Velocity at point B is

$$V = \sqrt{u^2 + v^2}$$

$$= \sqrt{(-0.12)^2 + (0.04)^2} = 0.126 \text{ cm/s}$$

(ii) For two dimensional flow the components of convective acceleration are

$$a_x = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}$$

and

$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}$$

$$\frac{\partial u}{\partial x} = -4 \; ; \frac{\partial u}{\partial y} = 0 \; ; \frac{\partial v}{\partial x} = 0 \text{ and } \frac{\partial v}{\partial y} = 4$$

Thus at point B, we have

$$a_x = -0.12 \times (-4) \times 10^{-4} \text{ cm/s}^2$$

$$= 48 \times 10^{-6} \text{ cm/s}^2$$

and

$$a_y = 0.04 \times 4 \times 10^{-4} \text{ cm/s}^2$$

$$= 16 \times 10^{-6} \text{ cm/s}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

∴ Convective acceleration at B is

$$a_x = \sqrt{a_x^2 + a_y^2}$$

$$= \sqrt{\left(48\times10^{-6}\right)^2 + \left(16\times10^{-6}\right)^2}$$

$$= 50.6\times10^{-6}\ \text{cm/s}^2$$

(iii) The flow rate per unit width across AB is given by

$$Q = u\times AB\times 1$$

$$u = -0.12\ \text{cm/s} = -0.12\times10^{-2}\text{m/s};\ \text{and}\ AB = 1\text{m}$$

$$\therefore\quad Q = -0.12\times10^{-2}\times1\times1\ \text{m}^3/\text{s}$$

$$= -0.12\times10^{-2}\text{m}^3/\text{s}$$

$$= -1200\ \text{cm}^3/\text{s}$$

Note: The negative sign simply indicates the flow in the negative x direction. However, the flow rate will be the same if it is in the positive x direction.

Alternatively the flow rate per unit width across AB is also given by

$$Q = \left(\psi_2 - \psi_1\right)\times 1$$

where ψ_2 is the value of the stream function for the curved boundary through point B; and ψ_1 is the value of the stream function for the straight horizontal boundary through point A

$$\psi_2 = 12\ \text{cm}^2/s;\ \text{and}\ \psi_1 = 0$$

$$\therefore\quad Q = 12\times1\times100\ \text{cm}^3/\text{s}$$

$$= 1200\ \text{cm}^3/\text{s}$$

Example 6.8. *The velocity components of the two-dimensional plane motion of a fluid are*

$$u = \frac{y^2 - x^2}{\left(x^2 + y^2\right)^2}\ \text{and}\ v = -\frac{2xy}{\left(x^2 + y^2\right)^2}$$

(*a*) *Show that the fluid is incompressible and flow is irrotational.*

(*b*) *Show that the points* (2, 2) *and* $\left(1, 2-\sqrt{3}\right)$ *are located on the same streamline.*

(*c*) *Determine the discharge across a line joining point (1, 1) and (2, 2) given that the thickness of the fluid stream normal to the* $x-y$ *plane is t.*

Solution

(a) For two dimensional flow of incompressible fluid the continuity equation may be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

$$u = \frac{y^2-x^2}{(x^2+y^2)^2}$$

$$\therefore \quad \frac{\partial u}{\partial x} = \frac{(-2x)(x^2+y^2)^2-(y^2-x^2)\times 2(x^2+y^2)\times 2x}{(x^2+y^2)^4}$$

$$= \frac{-2x(x^2+y^2)\ \ (x^2+y^2+2y^2-2x^2)}{(x^2+y^2)^4} = \frac{-2x(3y^2-x^2)}{(x^2+y^2)^3}$$

$$v = -\frac{2xy}{(x^2+y^2)^2}$$

$$\therefore \quad \frac{\partial v}{\partial y} = \frac{(-2x)(x^2+y^2)^2-(-2xy)\times 2(x^2+y^2)\times 2y}{(x^2+y^2)^4}$$

$$= \frac{-2x(x^2+y^2)\ \ (x^2+y^2-4y^2)}{(x^2+y^2)^4} = \frac{-2x(x^2-3y^2)}{(x^2+y^2)^3}$$

Introducing these values in the continuity equation, we get

$$\frac{-2x(3y^2-x^2+x^2-3y^2)}{(x^2+y^2)^3} = 0$$

which indicates that the fluid is incompressible

For two-dimensional flow in $x-y$ plane to be irrotational the condition to be satisfied is

$$\frac{\partial v}{\partial x} = \frac{\partial u}{\partial y}$$

$$\frac{\partial v}{\partial x} = \frac{(-2y)\ \ (x^2+y^2)^2-(-2xy)\times 2(x^2+y^2)\times 2x}{(x^2+y^2)^4}$$

$$= \frac{-2y(x^2+y^2)\ \ (x^2+y^2-4x^2)}{(x^2+y^2)^4}$$

$$= \frac{-2y(y^2-3x^2)}{(x^2+y^2)^3}$$

$$= \frac{2y(3x^2-y^2)}{(x^2+y^2)^3}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{\partial u}{\partial y} = \frac{2y(x^2+y^2)^2 - (y^2-x^2)\times 2(x^2+y^2)\times 2y}{(x^2+y^2)^4}$$

$$= \frac{2y(x^2+y^2)\quad (x^2+y^2-2y^2+2x^2)}{(x^2+y^2)^4}$$

$$= \frac{2y(3x^2-y^2)}{(x^2+y^2)^3}$$

Since in this case $\frac{\partial v}{\partial x} = \frac{\partial u}{\partial y}$, the flow is irrotational.

(b) The velocity components may be expressed in terms of stream function ψ as indicated below.

$$\frac{\partial \psi}{\partial x} = v = -\frac{2xy}{(x^2+y^2)^2} \quad ...(i)$$

and

$$\frac{\partial \psi}{\partial y} = -u = -\frac{y^2-x^2}{(x^2+y^2)^2} \quad ...(ii)$$

Integrating Eq. (i), we get

$$\psi = \frac{y}{(x^2+y^2)} + f(y) \quad ...(iii)$$

Differentating Eq. (iii) with respect to y, we get

$$\frac{\partial \psi}{\partial y} = \frac{(x^2+y^2)-y\times 2y}{(x^2+y^2)^2} + f'(y)$$

or

$$\frac{\partial \psi}{\partial y} = \frac{x^2-y^2}{(x^2+y^2)^2} + f'(y) \quad ...(iv)$$

Equating the values of $(\partial\psi / \partial y)$ given by Eqs (ii) and (iv), we get

$$f'(y) = 0$$

or $f(y) =$ constant, which may be taken as 0

$$\therefore \quad \psi = \frac{y}{(x^2+y^2)}$$

At point (2, 2) the value of ψ is obtained as

$$\psi = \frac{2}{(2)^2+(2)^2} = \frac{1}{4}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and at point $(1,\ 2-\sqrt{3})$ the value of ψ is obtained as

$$\psi = \frac{(2-\sqrt{3})}{(1)^2+(2-\sqrt{3})^2} = \frac{1}{4}$$

Since the value of ψ is same at the points (2, 2) and $(1,\ 2-\sqrt{3})$, both the points are located on the same streamline.

(c) For streamline passing through point (1, 1), the value of ψ is obtained as

$$\psi_1 = \frac{1}{2}$$

Similarly for streamline passing through point (2, 2), the value of ψ is obtained as

$$\psi_2 = \frac{1}{4}$$

∴ Discharge across a line joining points (1, 1) and (2, 2) is

$$Q = (\psi_1 - \psi_2)\times t = \left(\frac{1}{2}-\frac{1}{4}\right)\times t = \frac{t}{4}$$

Example 6.9. *A two-dimensional incompressible flow field is described by equation (a) V = (C/r); (b)V = Cr, in which V is the tangential velocity at a radius r and C is a constant. Determine for each case the circulation (i) around a circle of radius R, (ii) around a closed path formed by the arcs of two circles of radii R_1 and R_2 and the two radius vectors with an angle θ between them.*

Also calculate the vorticities of the flows described by these equations.

Solution

(i) From Eq. 6.35 circulation around a circle of radius *R* is

$$\Gamma = \int_0^{2\pi} VRd\theta = (2\pi R)V$$

(a) For $r = R$; $V = (C/R)$

∴ $\Gamma = 2\pi C$

(b) For $r = R$; $V = CR$

∴ $\Gamma = 2\pi\ CR^2$

(ii) Let *ABCD* be the closed path formed by the arcs of two circles of radii R_1 and R_2 and the two radius vectors with an angle θ between them, then $AB = R_1\theta$ and $CD = R_2\theta$. The total circulation around the path *ABCD* is given by

∴ $\Gamma = \Gamma_{AB} + \Gamma_{BC} + \Gamma_{CD} + \Gamma_{DA}$

(a) $\Gamma_{AB} = 2\pi C$; $\Gamma_{CD} = -2\pi C$; $\Gamma_{BC} = \Gamma_{DA} = 0$

∴ $\Gamma = 0$

(b) $\Gamma_{AB} = 2\pi CR_1^2$; $\Gamma_{CD} = -2\pi CR_2^2$; $\Gamma_{BC} = \Gamma_{DA} = 0$

$$\Gamma = 2\pi C\left(R_1^2 - R_2^2\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In cylindrical polar coordinates the vorticity may be expressed as

$$\zeta = \frac{\partial(rV_\theta)}{r\partial r} - \frac{\partial V_r}{r\partial\theta}$$

in which V_θ and V_r are the velocity components in the tangential and radial directions respectively.

(a) $V_\theta = V = (C/r)\ ;\ V_r = 0$

$\therefore$ $\zeta = 0$

(b) $V_\theta = V = Cr\ ;\ V_r = 0$

$\therefore$ $\zeta = 2C$

Note: The flow fields represented by case (a) and (b) correspond to free vortex motion and forced vortex motion respectively, both of which have been described in the next chapter. It may be noted that in the case of free vortex motion although the circulation is not equal to zero, the vorticity is equal to zero. This fact has been explained in the next chapter.

Example 6.10. *Derive the equation of stream function and velocity potential for a uniform stream of velocity V in a two-dimensional field, the velocity V being inclined to the x-axis at a positive angle α.*

Solution

The components of the velocity V in the x and y directions are

$$u = V\cos\alpha \text{ and } v = V\sin\alpha$$

From Eq. (6.41)

$$\frac{\partial\psi}{\partial x} = v = V\sin\alpha \qquad ...(i)$$

and

$$\frac{\partial\psi}{\partial y} = -u = -V\cos\alpha \qquad ...(ii)$$

Integrating Eq. (i), we get

$$\psi = (V\sin\alpha)\,x + f(y) \qquad ...(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial\psi}{\partial y} = f'(y) \qquad ...(iv)$$

Equating the values of $\frac{\partial\psi}{\partial y}$ from Eqs (ii) and (iv), we get

$$f'(y) = -V\cos\alpha$$

By integrating both sides of the above expression, we get

$$f(y) = (-V\cos\alpha)\,y + C$$

where C is a constant of integration.

Therefore

$$\psi = (V\sin\alpha)\,x - (V\cos\alpha)\,y + C$$

From Eq. (6.39)

$$-\frac{\partial\phi}{\partial x} = u = V\cos\alpha \qquad ...(i)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$-\frac{\partial \phi}{\partial y} = v = V \sin \alpha \quad \text{...(ii)}$$

Integrating Eq. (i), we get

$$\phi = (-V \cos \alpha)\, x + f(y) \quad \text{...(iii)}$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \phi}{\partial y} = f'(y) \quad \text{...(iv)}$$

Equating the values of $\left(\frac{\partial \phi}{\partial y}\right)$ from Eqs (ii) and (iv), we get $f'(y) = -V \sin \alpha$.

By integrating both sides of the above expression, we get

$$f(y) = (-V \sin \alpha)\, y + C$$

where C is a constant of integration.

Therefore

$$\phi = (-V \cos \alpha)\, x + (-V \sin \alpha)\, y + C.$$

Example 6.11. *A stream function is given by $\psi = 3x^2 - y^3$. Determine the magnitude of velocity components at the point (2, 1).*

Solution

From Eq. (6.41)

$$v = \frac{\partial \psi}{\partial x} = \frac{\partial}{\partial x}(3x^2 - y^3) = 6x$$

and $$u = -\frac{\partial \psi}{\partial y} = -\frac{\partial}{\partial y}(3x^2 - y^3) = 3y^2$$

At any point (2,1)

$$u = 3 \text{ and } v = 12$$

and the total velocity is the vector sum of two components. Thus

$$V = \sqrt{u^2 + v^2}$$

$$= \sqrt{(3)^2 + (12)^2} = 12.37$$

Example 6.12. *A stream function in a two-dimensional flow is $\psi = 2xy$. Show that the flow is irrotational and determine the corresponding velocity potential ϕ.*

Solution

From Eq. (6.45) a stream function ψ represents an irrotational flow if

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

By substitution

$$\left[\frac{\partial^2}{\partial x^2}(2xy) + \frac{\partial^2}{\partial y^2}(2xy)\right] = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence it represents an irrotational flow.

From Eq. 6.44

$$\frac{\partial \phi}{\partial x} = \frac{\partial \psi}{\partial y} = \frac{\partial}{\partial y}(2xy) = 2x \qquad ...(i)$$

$$\frac{\partial \phi}{\partial y} = -\frac{\partial \psi}{\partial x} = -\frac{\partial}{\partial x}(2xy) = -2y \qquad ...(ii)$$

Integrating Eq. (i), we get

$$\phi = x^2 + f(y) \qquad ...(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \phi}{\partial y} = f'(y) \qquad ...(iv)$$

Equating the values of $\left(\frac{\partial \phi}{\partial y}\right)$ from Eqs (ii) and (iv), we get

$$f'(y) = -2y$$

Integrating both the sides of the above expression, we get

$$f(y) = -y^2 + C$$

where C is the constant of integration.

Therefore

$$\phi = x^2 - y^2 + C.$$

Example 6.13. *A nozzle is so shaped that the velocity of flow along the centre line changes linearly from 1.5 m/s to 15 m/s in a distance of 0.375 m. Determine the magnitude of the convective acceleration at the beginning and end of this distance.*

Solution

The rate of change of V with respect to s is

$$\frac{\partial V}{\partial s} = \frac{15 - 1.5}{0.375} = 36 \text{ m/s per m}$$

Convective acceleration

$$a_s = V \frac{\partial V}{\partial s}$$

Therefore

$$a_{s_1} = (1.5 \times 36) = 54 \text{ m/s}^2$$

and

$$a_{s_2} = (15 \times 36) = 540 \text{ m/s}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 6.14. *The bucket of a spillway has a radius of 6 m. When the spillway is discharging 5 cubic metre of water per second per metre length of crest, the average thickness of the sheet of water over the bucket is 0.4 m. Compare the resulting normal acceleration (or centripetal acceleration) with the acceleration due to gravity.*

Solution

$$V = \frac{q}{y} = \frac{5}{0.40} = 12.5 \text{ m/s}$$

$$a_n = \frac{V^2}{r} = \frac{(12.5)^2}{6} = 26.04 \text{ m/s}^2$$

$$\therefore \quad \frac{a_n}{g} = \frac{26.04}{9.81} = 2.65.$$

Example 6.15. *If for two-dimensional flow the stream function is given by $\psi = 2xy$, calculate the velocity at a point (3, 6). Show that the velocity potential ϕ exists for this case and deduce it. Also draw the streamlines corresponding to ψ =100 and ψ = 300 and equipotential lines correspond to ϕ = 100 and ϕ = 300.*

Solution

$$\psi = 2xy$$

Since $\frac{\partial \psi}{\partial x} = v$ and $\frac{\partial \psi}{\partial y} = -u$

$$\therefore \quad v = 2y \text{ and } u = -2x$$

Thus at point (3, 6)

$$v = 12 \text{ and } u = -6$$

$$\therefore \quad V = \sqrt{u^2 + v^2}$$

$$= \sqrt{(6)^2 + (12)^2} = 13.42$$

As shown in Ex. 6.12 the velocity potential exists for this case and its value is

$$\phi = x^2 - y^2.$$

Thus considering the following equations corresponding to the given values of ψ and ϕ viz.,

$$2xy = 100$$

or $$xy = 50 \quad \text{...(i)}$$

$$2xy = 300$$

or $$xy = 150 \quad \text{...(ii)}$$

$$x^2 - y^2 = 100 \quad \text{...(iii)}$$

$$x^2 - y^2 = 300 \quad \text{...(iv)}$$

the streamlines and equipotential lines may be plotted as shown in Fig. Ex. 6.15 by means of firm lines. In the same figure the streamlines and equipotential lines shown by means of dotted lines have been drawn for symmetry by considering the negative values of the given constants.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

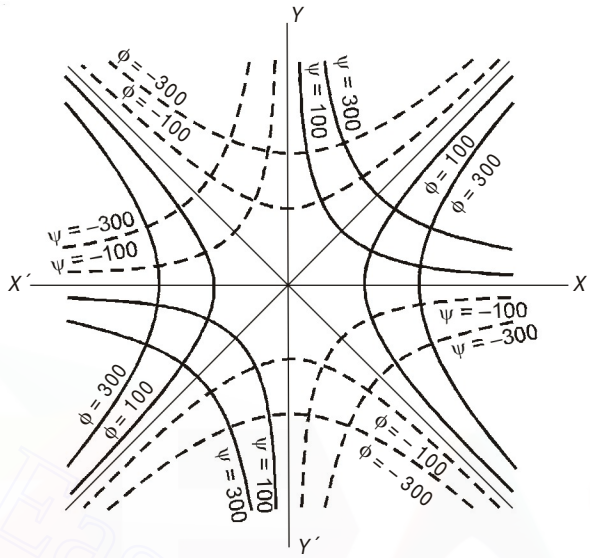


Figure Ex. 6.15

It may however be indicated that the flow-net obtained in the present case represents the flow pattern for flow around a 90° corner (or bend).

SUMMARY OF MAIN POINTS

1. The science which deals with the geometry of motion of fluids without reference to the forces causing the motion is known as 'kinematics'. Thus kinematics involves merely the description of the motion of fluids in terms of space-time relationship.
2. The science which deals with the action of the forces in producing or changing motion of fluids is known as 'kinetics'.
3. Fluid flow is said to be 'steady' if at any point in the flowing fluid various characteristics such as velocity, pressure, density, temperature, etc., which describe the behaviour of the fluid in motion, do not change with time. Mathematically it may be expressed as

$$\frac{\partial V}{\partial t} = 0 ; \frac{\partial p}{\partial t} = 0 ; \frac{\partial \rho}{\partial t} = 0 ; \text{ etc.}$$

4. Fluid flow is said to be 'unsteady' if at any point in the flowing fluid, any one or all the characteristics which describe the behaviour of the fluid in motion change with time. Mathematically it may be expressed as

$$\frac{\partial V}{\partial t} \neq 0 ; \text{ and or } \frac{\partial p}{\partial t} \neq 0 ; \text{ etc.}$$

5. When the velocity of flow of fluid does not change, both in magnitude and direction, from point to point in the flowing fluid, for any given instant of time, the flow is said to be 'uniform'. In the mathematical form a uniform flow may be expressed as

$$\frac{\partial V}{\partial s} = 0$$

6. If the velocity of flow of fluid changes from point to point in the flowing fluid at any instant, the flow is said to be non-uniform. In the mathematical form a non-uniform flow may be expressed as

$$\frac{\partial V}{\partial s} \neq 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

7. The flow is said to be 'one dimensional' if the various characteristics of the flowing fluid are the functions of only one of the three coordinate directions, and time t.
8. The flow is said to be two dimensional if the various characteristics of the flowing fluid are the function of only any two of the three coordinate directions, and time t.
9. The flow is said to be 'three dimensional' if the various characteristics of the flowing fluid are the functions of all the three coordinate directions and time t.
10. A flow is said to be 'rotational' if the fluid particles while moving in the direction of flow rotate about their mass centres.
11. A flow is said to be 'irrotational' if the fluid particles while moving in the direction of flow do not rotate about their mass centres.
12. A flow is said to be laminar when the various fluid particles move in layers (or laminae) with one layer of fluid sliding smoothly over an adjacent layer. In the development of laminar flow the viscosity of the flowing fluid plays a significant role, and hence the flow of a very viscous fluid may in general be treated as laminar flow.
13. A fluid motion is said to be turbulent when the fluid particles move in an entirely haphazard or disorderly manner, that results in a rapid and continuous mixing of the fluid leading to momentum transfer as flow occurs.
14. A 'streamline' is an imaginary curve drawn through a flowing fluid in such a way that the tangent to it at any point gives the direction of the velocity of flow at that point.

    If u and v are the components of the velocity V along x and y directions then the equation for streamlines in the xy plane may be written as

    $$udy - vdx = 0$$
15. A 'stream tube' is a tube imagined to be formed by a group of streamlines passing through a small closed curve, which may or may not be circular.
16. A'path-line' may be defined as the line traced by a single fluid particle as it moves over a period of time.

    In steady flow path-lines and streamlines are identical.
17. A streak-line may be defined as a line that is traced by a fluid particle passing through a fixed point in a flow field.

    In steady flow a streak-line a streamline and a path-line are identical. In unsteady flow a streak-line at any instant is the locus of end points of particle paths (or path-lines) that started at the instant the particle passed through the injection point.
18. The continuity equation in the differential from in cartesian coordinates in the most general from which is applicable for steady as well as unsteady flow, uniform as well as non-uniform flow, and compressible as well as incompressible fluids, is

    $$\frac{d\rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial v} + \frac{\partial(\rho z)}{\partial z} = 0$$

    where

    ρ = mass density of fluid ; and u, v, w = components of velocity V along x, y and z directions respectively. For steady flow since $\frac{d\rho}{dt} = 0$; the above equation reduces to

    $$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial v} + \frac{\partial(\rho w)}{\partial z} = 0$$

    For an incompressible fluid the mass density does not change with x, y, z, and t and hence the equation of coutinuity simplifies to

    $$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$
19. The continuity equation for two-dimensional flow in the most general form, is

    $$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial v} = 0$$

    For steady two dimensional flow the continuity equation is

    $$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial v} = 0$$

    For two dimensional flow of incompressible fluid the continuity equation is

    $$\frac{\partial \rho}{\partial t} + \frac{\partial v}{\partial y} = 0$$
20. The continuity equation for one dimensional flow in the most general from is

    $$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For steady one dimensional flow the continuity equation is

$$\frac{\partial(\rho u)}{\partial x} = 0$$

For one dimensional flow of incompressible fluid the continuity equation is

$$\frac{\partial u}{\partial x} = 0$$

21. If velocity of flow is uniform at each section of the flow passage, the continuity equation may be written as

$$Q = AV = \text{constant}$$

or

$$Q = A_1V_1 = A_2V_2 = A_3V_3 = \text{constant}$$

where Q is known as 'discharge' which is defined as the volume rate of flow of fluid or the volume of fluid flowing through any section per unit time. Discharge is usually measured in terms of cubic metre per second (or m^3/s) or liters per second. One cubic metre per second is usually written as one cumec.

22. Acceleration is defined as the rate of change of velocity with respect to time. The components of acceleration a_x, a_y, and a_z along x, y, and z, directions are as follows.

$$a_x = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}$$

$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t}$$

$$a_z = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t}$$

In the expressions for the components of acceleration each of the terms $\left(\frac{\partial u}{\partial t}\right), \left(\frac{\partial v}{\partial t}\right)$ and $\left(\frac{\partial w}{\partial t}\right)$ represents the 'local acceleration', and the remaining terms in these expressions represent the 'convective acceleration'.

23. When a fluid flows along a curved path, normal and tangential accelerations are developed. The expressions for the normal acceleration a_n and the tangential accelerations a_s are

$$a_n = \frac{V_s^2}{r} + \frac{\partial V_n}{\partial t}$$

$$a_s = V_s\frac{\partial V_s}{\partial s} + \frac{\partial V_s}{\partial t}$$

For steady flow

$$a_n = \frac{V_s^2}{r}$$

$$a_s = V_s\frac{\partial Vs}{\partial s}$$

where

V_s = tangential component of velocity ;

V_n = normal component of velocity ;

and

r = radius of curvature of the streamlines.

24. The rotation components ω_x, ω_y and ω_z about x, y and z axes respectively are given as

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)$$

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

25. The flow along a closed curve is called 'circulation'. The mathematical concept of circulation is that it is the line integral, taken completely around a closed curve, of the velocity vector.

26. The vorticity is defined as circulation per unit area. Mathmatically vorticity is equal to two times the rotation component.

27. The 'velocity potential' ϕ is defined as a scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction.

Thus

$$u = -\frac{\partial \phi}{\partial x}\ ;\ v = -\frac{\partial \phi}{\partial y}\ ;\ \text{and}\ w = -\frac{\partial \phi}{\partial z}$$

28. The 'stream function' ψ is defined as a scalar function of space and time such that its partial derivative with respect to any direction gives the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

velocity component at right angles (in the counter-clock wise direction) to this direction. Thus for the case of two dimensional flow

$$u = -\frac{\partial \psi}{\partial y} \quad v = \frac{\partial \psi}{\partial x}$$

29. The 'streamlines' are represented by ψ = constant, and ϕ = constant represents 'equipotential lines'. The streamlines and equipotential lines intersect each other orthogonally at all points of intersection.
29. A grid obtained by drawing a series of streamlines and equipotential lines is known as a 'flow net'.

PROBLEMS

6.1 Define and distinguish between (a) steady and unsteady flow: (b) uniform and non-uniform flow (c) rotational and irrotational flow.

6.2 Define and distinguish between streamline, path line and streak line.

6.3 What is meant by one-dimensional, two-dimensional and three dimensional flows?

6.4 An incompressible fluid flows steadily through two pipes of diameter 0.15 m and 0.2 m which combine to discharge in a pipe of 0.3 m diameter. If the average velocities in the 0.15 m and 0.2 m diameter pipes are 2 m/s and 3 m/s respectively, then find the average velocity in the 0.3 m diameter pipe. [*Ans.* 1.83 m/s]

6.5 Determine which of the following pairs of velocity componets u and v satisfy the continuity equation for a two-dimensional flow of an incompressible fluid.

(a) $u = Cx$; $v = -Cy$

(b) $u = (3x - y)$; $v = (2x + 3y)$

(c) $u = (x + y)$; $v = (x^2 - y)$

(d) $u = A \sin xy$; $v = -A \sin xy$

(e) $u = 2x^2 + 3y^2$; $v = -3xy$.

[*Ans.* (a) Satisfy ; (b) Does not satisfy (c) Satisfy; (d) Does not satisfy ; (e) Does not satisfy]

6.6 Calculate the unknown velocity components so that they satisfy continuity equation:

(a) $u = 2x^2$; $v = xyz$; $w = ?$

(b) $u = (2x^2 + 2xy)$; $w = (z^3 - 4xz - 2yz)$; $v = ?$

[*Ans.* (a) $-4xz - xz^2 + f(x,y)$; (b) $-3yz^2 + f(x, z)$]

6.7 Determine which of the following velocity fields represent possible example of irrotational flow:

(a) $u = Cx$; $v = -Cy$

(b) $u = -Cxy$; $v = C \log xy$

(c) $u = (Ax^2 - Bxy)$; $v = \left(-2Axy + \frac{1}{2} By^2\right)$.

[*Ans.* (a) Irrotational flow ; (b) Rotational flow; (c) Rotational flow]

6.8 Which of the following stream function ψ are possible irrotational flow fields?

(a) $\psi = Ax + By^2$; (b) $\psi = Ax^2 y^2$

(c) $\psi = A \sin xy$; (d) $\psi = A \log (x/y)$

(e) $\psi = Ay\left(1 - \frac{C}{x^2 + y^2}\right)$ (f) $\psi = (y^2 - x^2)$.

[*Ans.* (a) to (d) None ; (e) and (f) Possible cases of Irrotational Flow Fields]

6.9 In a steady flow two points A and B are 0.5 m apart on a straight streamline. If the velocity of flow varies linearly between A and B what is the acceleration at each point if the velocity at A is 2 m/s and the velocity at B is 6 m/s ?

[*Ans.* at A 16 m/s^2 ; at B 48 m/s^2]

6.10 Calculate the velocity components u and v for the following velocity potential functions ϕ :

(a) $\phi = x + y$; (b) $\phi = x^2 + y^2$

(c) $\phi = \tan^{-1}(y/x)$; (d) $\phi = \frac{Ax}{x^2 + y^2}$;

(e) $\phi = \sin x \sin y$; (f) $\phi = \log (x + y)$.

Which of these velocity potential functions satisfy the continuity equation?

[*Ans.* (a) $-1, -1$; (b) $-2x, -2y$; (c) $\frac{y}{(x^2 + y^2)}$; $\frac{-x}{(x^2 + y^2)}$; (d) $\frac{A(x^2 - y^2)}{(x^2 + y^2)^2}$; $\frac{2Axy}{(x^2 + y^2)^2}$; (e) $-\cos x, -\cos y$; (f) $\frac{-1}{(x + y)}$; $\frac{-1}{(x + y)}$. (a) , (c) (d) Satisfy ; (b) (e) (f) Do not satisfy]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

6.11 Calculate the total velocity at points (–2, 1), (2, 3) and (4, 0) for the velocity potential functions in Problem 6.10.

[*Ans.*

At Point	(a)	(b)	(c)	(d)	(e)	(f)
(–2,1)	$\sqrt{2}$;	$\sqrt{20}$;	$1/\sqrt{5}$;	$A/\sqrt{25}$	$\left[\sqrt{\cos^2(2)+\cos^2(1]};\right]$	$\sqrt{2}$
(2,3)	$\sqrt{2}$;	$\sqrt{52}$;	$1/\sqrt{13}$;	$A/\sqrt{169}$;	$\left[\sqrt{\cos^2(2)+\cos^2(3)};\right]$	$\sqrt{2}/5$
(4,0)	$\sqrt{2}$;	–8;	1/4;	$A/16$;	$[\sqrt{1+\cos^2(4)}]$	$\sqrt{2}/4$

6.12 Determine the corresponding stream function for each of the velocity potential functions in Problem 6.10

[*Ans.* (a) $y - x$ + constant ; (c) $-\frac{1}{2}\log(x^2 + y^2)$ + constant ; (d) $-\frac{Ay}{(x^2+y^2)}$ + constant]

Note: The velocity potential functions (b), (e), (f) do not represent the flow field and therefore no explicit stream functions corresponding to these can be obtained. The velocities of (b), (e), (f) determined in Problem 6.10 above have therefore no physical significance.

6.13 Describe the use and limitations of flow nets.

6.14 Discuss briefly the different methods of drawing the flow nets.

6.15 For a two dimensional flow of an incompressible fluid the velocity components V_r and V_θ are given as

$V_r = akr^n e^{[-k(n+1)\theta]}$, and $V_\theta = ar^n e^{[-k(n+1)\theta}$.

Determine the stream function ψ and show that the fluid velocity at any point is given by

$$V = \left[(n+1)\psi\sqrt{(1+k^2)}/r\right]$$

$$\left[Ans.\ \psi = \frac{1}{n+1}\{ar^{(n+1)}e^{-k(n+1)\theta}\}\right]$$

6.16 What is the irrotational velocity field associated with the potential

$\phi = 3x^2 - 3x + 3y^2 + 16t^2 + 12zt$

Does the flow field satisfy the incompressible continuity equation.

[*Ans.* $u = -3(2x - 1)$; $v = -6y$, $w = -12t$; Does not satisfy]

6.17 In two-dimensional incompressible flow show that the flow rate per unit width between two streamlines is equal to the difference between the values of the stream function corresponding to these stream lines.

6.18 The velocity components in the x– and y–directions are given as$u = (2xy^3 - (2yx^3/3)$ and $v = xy^2 - (2yx^3/3)$. Indicate whether the given velocity distribution is a possible field of flow or not.

[*Ans.* Not a possible field of flow]

6.19 If the expression for the stream function is described by $\psi = x^3 - 3xy^2$ indicate whether the flow is rotational or irrotational. If the flow is irrotational determine the value of the velocity potential.

[*Ans.* Flow is irrotational; $\phi = y^3 - 3x^2y$]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equations of Motion and Energy Equation

Chapter 7

7.1 INTRODUCTION

A fluid in motion is subjected to several forces which results in the variation of the acceleration and the energies involved in the flow phenomenon of the fluid. As such in the study of the fluid motion the forces and energies that are involved in the flow are required to be considered. This aspect of fluid motion is known as dynamics of fluid flow.

The various forces acting on a fluid mass may be classified as (i) body or volume forces, (ii) surface forces, and (iii) line forces. The body or volume forces are the forces which are proportional to the volume of the body, the examples of which are weight, centrifugal force, magnetic force, electromotive force, etc. The surface forces are the forces which are proportional to the surface area which may include pressure force, shear or tangential force, force of compressibility, force due to turbulence etc. The linear forces are the forces which are proportional to the length and the example of which is surface tension.

Alike mechanics of solids, the dynamics of fluid is also governed by Newton's second law of motion. Newton's second law of motion states that the resultant force on any fluid element must equal the product of the mass and the acceleration of the element and the acceleration vector has the direction of the resultant force vector. In the mathematical form this law may be expressed as

$$\Sigma F = Ma \qquad \text{... (7.1)}$$

where ΣF represents the resultant external force acting on the fluid element of mass M and a is the total acceleration. Obviously both the acceleration and the resultant external force must be along the same line of action. Therefore the force and the acceleration vectors can be resolved along the three reference directions x, y and z and the corresponding equations may be expressed as

$$\left.\begin{aligned} \Sigma F_x &= Ma_x \\ \Sigma F_y &= Ma_y \\ \Sigma F_z &= Ma_z \end{aligned}\right\} \qquad \text{... (7.1 a)}$$

where ΣF_x, ΣF_y and ΣF_z are the components of the resultant force in the x, y and z directions respectively and a_x, a_y and a_z are the components of the total acceleration in the x, y and z directions respectively.

Often in the study of the fluid motion the forces per unit volume of the fluid element are required to be considered. Equations 7.1 and 7.1 (a) may then be expressed as

$$\Sigma f = \rho a \qquad \text{... (7.2)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and
$$\left.\begin{aligned}\Sigma f_x &= \rho a_x \\ \Sigma f_y &= \rho a_y \\ \Sigma f_z &= \rho a_z\end{aligned}\right\} \quad \text{... (7.2 a)}$$

in which f is the resultant force per unit volume acting on the fluid mass, ρ is the mass density of the fluid, and f_x, f_y and f_z are the components of the resultant force per unit volume, f, along x, y and z directions.

7.2 FORCES ACTING ON FLUID IN MOTION

The various forces that may influence the motion of a fluid are due to gravity, pressure, viscosity, turbulence, surface tension and compressibility. The *gravity force* F_g is due to the weight of the fluid and it is equal to Mg. The gravity force per unit volume is equal to ρg. The *pressure force* F_p is exerted on the fluid mass if there exists a pressure gradient between the two points in the direction of flow. The *viscous force* F_v is due to the viscosity of the flowing fluid and thus exists in the case of all real fluids. The turbulent force F_t is due to the turbulence of the flow. In the turbulent flow the fluid particles move from one layer to other and therefore, there is a continuous momentum transfer between adjacent layers, which results in developing additional stresses (called Reynolds stresses) for the flowing fluid. The *surface tension force* F_s is due to the cohesive property of the fluid mass. It is, however, important only when the depth of flow is extremely small. The *compressibility force* F_e is due to the elastic property of the fluid and it is important only either for compressible fluids or in the cases of flowing fluids in which the elastic properties of fluids are significant.

If a certain mass of fluid in the motion is influenced by all the above mentioned forces, then according to Newton's second law of motion the following equation of motion may be written as

$$Ma = F_g + F_p + F_v + F_t + F_s + F_e \quad \text{... (7.3)}$$

Further by resolving the various forces and the acceleration along the x, y and z directions the following equations of motion may be obtained.

$$\left.\begin{aligned}Ma_x &= F_{gx} + F_{px} + F_{vx} + F_{tx} + F_{sx} + F_{ex} \\ Ma_y &= F_{gy} + F_{py} + F_{vy} + F_{ty} + F_{sy} + F_{ey} \\ Ma_z &= F_{gz} + F_{pz} + F_{vz} + F_{tz} + F_{sz} + F_{ez}\end{aligned}\right\} \quad \text{... (7.3 a)}$$

The subscripts x, y, z are introduced to represent the components of each of the forces and the acceleration in the respective directions.

In most of the problems of the fluids in motion the surface tension forces and the compressibility forces are not significant. Hence these forces may be neglected. Then Eq. 7.3 and 7.3(a) become

$$Ma = F_g + F_p + F_v + F_t \quad \text{... (7.4)}$$

and
$$\left.\begin{aligned}Ma_x &= F_{gx} + F_{px} + F_{vx} + F_{tx}, \\ Ma_y &= F_{gy} + F_{py} + F_{vy} + F_{ty}, \\ Ma_z &= F_{gz} + F_{pz} + F_{vz} + F_{tz}\end{aligned}\right\} \quad \text{... (7.4 a)}$$

Equation 7.4 (a) are known as *Reynolds' equations of motion* which are useful in the analysis of the turbulent flows.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further for laminar or viscous flows the turbulent forces also become less significant and hence these may be neglected. The Eqs. 7.4 and 7.4 (a) may then be modified as

$$Ma = F_g + F_p + F_v \quad \text{...(7.5)}$$

and

$$\left.\begin{aligned} Ma_x &= F_{gx} + F_{px} + F_{vx} \\ Ma_y &= F_{gy} + F_{py} + F_{vy} \\ Ma_z &= F_{gz} + F_{pz} + F_{vz} \end{aligned}\right\} \quad \text{... (7.5a)}$$

Equation 7.5 (a) are known as Navier–Stokes equations which are useful in the analysis of viscous flow.

Further if the viscous forces are also of little significance in the problems of fluid flows, then these forces may also be neglected. The viscous forces will become insignificant if the flowing fluid is an ideal fluid. However, in the case of real fluids also the viscous forces may be considered to be insignificant if the viscosity of the flowing fluid is small. In such cases the Eqs 7.5 and 7.5(a) may be further modified as

$$M_a = F_g + F_p \quad \text{... (7.6)}$$

and

$$\left.\begin{aligned} Ma_x &= F_{gx} + F_{px} \\ Ma_y &= F_{gy} + F_{py} \\ Ma_z &= F_{gz} + F_{pz} \end{aligned}\right\} \quad \text{... (7.6 a)}$$

Equations 7.6 (a) are known as *Euler's equations of motion.*

A detailed study of the Reynolds equations of motion and the Navier–Stokes equations are beyond the scope of this book. The Euler's equations of motion are however dealt with in detail in the next section.

7.3 EULER'S EQUATION OF MOTION

As stated above in the Euler's equations of motion only two forces, namely the pressure force and the fluid weight or in general the body force, are assumed to be acting on the mass of fluid in motion. Further in the derivation of these equations it is assumed that the fluid is non-viscous, so that the pressure forces may be considered to be acting in the direction normal to the surface.

Consider a point P (x, y, z) in a flowing mass of fluid at which let u, v and w be the velocity components in the directions x, y and z respectively ; ρ be the mass density of the fluid and p be the pressure intensity. Further let X, Y and Z be the components of the body force per unit mass at the same point.

A parallelopiped of fluid with its edges of length δx, δy and δz parallel to x, y and z axes is considered in the fluid mass with the point P (x, y, z) as one of its corners, as shown in Fig. 7.1.

The various external forces acting on the parallelopiped may now be evaluated. The mass of the fluid contained in the parallelopiped is $(\rho \delta x\ \delta y\ \delta z)$. Therefore the total component of the body force acting on the parallelopiped in the x direction is equal to $X\ (\rho \delta x\ \delta y\ \delta z)$.

Similarly the components of the body force acting on the parallelopiped in the y and z directions are obtained as $Y\ (\rho \delta x\ \delta y\ \delta z)$ and $Z\ (\rho \delta x\ \delta y\ \delta z)$ respectively.

The pressure intensity at the point P is p. Since the length of the edges of the parallelopiped are extremely small, it may be assumed that the pressure intensity on the face $PQR'S$ is uniform and equal to p. Therefore the total pressure force acting on the face $PQR'S$ is in the direction normal to the surface

$$= (p\ \delta y\ \delta z)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since the pressure p varies with x, y and z, the pressure intensity on the face $RS'P'Q'$ will be

$$= \left(p + \frac{\partial p}{\partial x} \delta x \right)$$

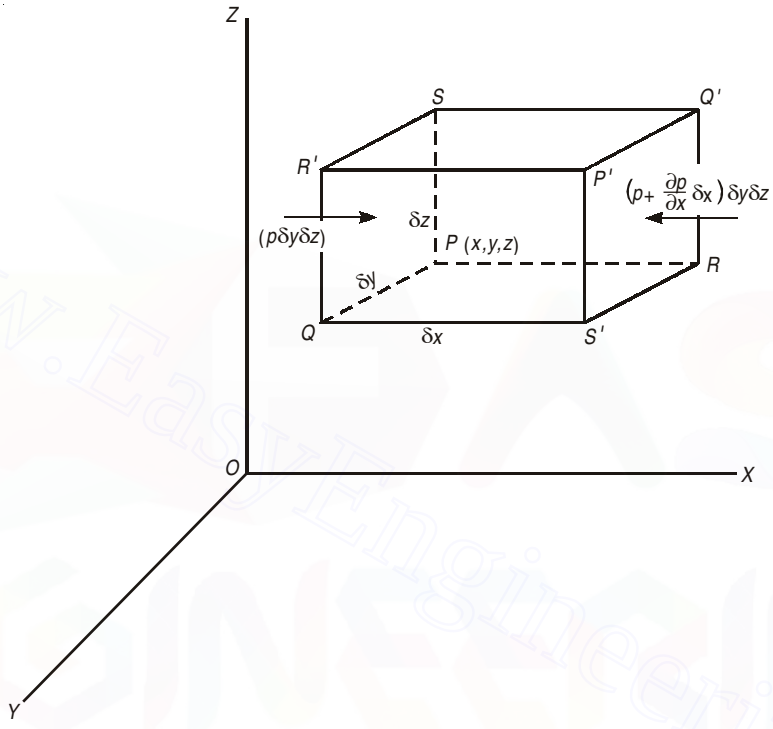


Figure 7.1 *Elementary parallelopiped subjected to the pressure and body forces*

Therefore the total pressure force acting on the face $RS'P'Q'$ in the direction normal to the force

$$= \left(p + \frac{\partial p}{\partial x} \delta x \right) \delta y \, \delta z$$

Since there is a difference of pressure between the two end faces of the parallelopiped, a net pressure force F_{px} acts on the fluid mass in the x direction, the magnitude of which is obtained as

$$F_{px} = p\delta y \, \delta z - \left(p + \frac{\partial p}{\partial x} \delta x \right) \delta y \, \delta z$$

or

$$F_{px} = -\frac{\partial p}{\partial x} \delta x \, \delta y \, \delta z \qquad ...(7.7\ a)$$

In the same manner the components of the pressure force in the y and z directions may also be obtained as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F_{py} = \left(-\frac{\partial p}{\partial y}\delta x\ \delta y\ \delta z\right) \quad \text{... (7.7 b)}$$

and

$$F_{pz} = \left(-\frac{\partial p}{\partial z}\delta x\ \delta y\ \delta z\right) \quad \text{... (7.7 c)}$$

The components of the pressure force per unit volume may be obtained by dividing both sides of Eq. 7.7 by the volume of the parallelopiped ($\delta x\ \delta y\ \delta z$) as

$$\left.\begin{aligned} f_{px} &= -\frac{\partial p}{\partial x} \\ f_{py} &= -\frac{\partial p}{\partial y} \\ f_{pz} &= -\frac{\partial p}{\partial z} \end{aligned}\right\} \quad \text{... (7.8)}$$

Now adding the body force and the pressure forces and equating to the product of the mass and the acceleration in the x direction, in accordance with the Newton's second law of motion, we get

$$X(\rho\ \delta x\ \delta y\ \delta z) - \frac{\partial p}{\partial x}\ \delta x\ \delta y\ \delta z = (\rho\ \delta x\ \delta y\ \delta z)\ a_x$$

where a_x is the component of the total acceleration in the x direction.

Dividing both sides by the mass of the fluid ($\rho\ \delta x\ \delta y\ \delta z$) in the parallelopiped and taking the limit so that the parallelopiped tends to point P, we obtain the Euler's equation of motion in the x direction at any point in the flowing mass of fluid as

$$X - \frac{1}{\rho}\frac{\partial p}{\partial x} = a_x \quad \text{..(7.9 a)}$$

In the same manner the equations for the y and z directions may be obtained as

$$Y - \frac{1}{\rho}\frac{\partial p}{\partial y} = a_y \quad \text{...(7.9 b)}$$

and

$$Z - \frac{1}{\rho}\frac{\partial p}{\partial z} = a_z \quad \text{...(7.9 c)}$$

Equations 7.9 (a), 7.9 (b) and 7.9 (c) are known as Euler's equations of motion.

As indicated in Chapter 6 the acceleration components a_x, a_y and a_z may be expressed in terms of the velocity components u, v and w as

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}$$

$$a_y = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$a_z = \frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}$$

Substituting these expressions for the accelerations in Eq. 7.9, the Euler's equations of motion are obtained as

$$\left.\begin{aligned} X - \frac{1}{\rho}\frac{\partial p}{\partial x} &= \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \\ Y - \frac{1}{\rho}\frac{\partial p}{\partial y} &= \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \\ Z - \frac{1}{\rho}\frac{\partial p}{\partial z} &= \frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \end{aligned}\right\} \quad \text{... (7.10)}$$

In the derivation of these equations, no assumption has been made that the mass density ρ is a constant. Hence these equations of motion are applicable to compressible or incompressible, non viscous fluids in steady or unsteady state of flow.

7.4 INTEGRATION OF EULER'S EQUATIONS

The Euler's equations of motion can be integrated to yield the energy equation under the following assumptions.

(i) There exists a *force potential* which is defined as that whose negative derivative with respect to any direction gives the component of the body force per unit mass in that direction. It is usually denoted by Ω and according to the definition given

and

$$\left.\begin{aligned} X &= -\frac{\partial \Omega}{\partial x} \\ Y &= -\frac{\partial \Omega}{\partial y} \\ Z &= -\frac{\partial \Omega}{\partial z} \end{aligned}\right\} \quad \text{... (7.11)}$$

(ii) The flow is irrotational i.e., the velocity potential exists or the flow may be rotational, but it is steady.

(a) **When the flow is irrotational.** Considering the Euler's equation of motion in one of the directions, we have

$$X - \frac{1}{\rho}\frac{\partial p}{\partial x} = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}$$

If the flow is irrotational then as indicated in Chapter 6

$$\frac{\partial u}{\partial y} = \frac{\partial v}{\partial x};\ \frac{\partial u}{\partial z} = \frac{\partial w}{\partial x}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $u = -\dfrac{\partial \phi}{\partial x}$, since the velocity potential ϕ exists for an irrotational flow.

Then
$$\frac{\partial u}{\partial t} = \frac{\partial}{\partial t}\left(-\frac{\partial \phi}{\partial x}\right)$$
$$= -\frac{\partial^2 \phi}{\partial t\, \partial x} = -\frac{\partial^2 \phi}{\partial x\, \partial t}$$

The order of differentiation has been changed meaning thereby that $\dfrac{\partial^2 \phi}{\partial t\, \partial x}$ is same as $\dfrac{\partial^2 \phi}{\partial x\, \partial t}$, which is however true only if ϕ is a continuous function and its second derivative exists.

By substituting these values the equation of motion in the x direction becomes

$$-\frac{\partial \Omega}{\partial x} - \frac{1}{\rho}\frac{\partial p}{\partial x} = -\frac{\partial^2 \phi}{\partial t\, \partial x} + u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial x} + w\frac{\partial w}{\partial x}$$

or
$$-\frac{\partial \Omega}{\partial x} - \frac{1}{\rho}\frac{\partial p}{\partial x} = -\frac{\partial^2 \phi}{\partial t\, \partial x} + \frac{1}{2}\frac{\partial u^2}{\partial x} + \frac{1}{2}\frac{\partial v^2}{\partial x} + \frac{1}{2}\frac{\partial w^2}{\partial x}$$

or
$$\frac{\partial}{\partial x}\left[\Omega + \frac{p}{\rho} + \frac{u^2 + v^2 + w^2}{2} - \frac{\partial \phi}{\partial t}\right] = 0$$

Now if it is assumed that the fluid is incompressible so that the mass density ρ is independent of pressure, then integrating the above equation with respect to x, we get

$$\Omega + \frac{p}{\rho} + \frac{u^2 + v^2 + w^2}{2} - \frac{\partial \phi}{\partial t} = F_1\ (y, z, t)$$

where F_1 is an arbitrary function resulting from integration.

However if the fluid is compressible, so that the mass density ρ depends on the pressure p, then integrating the above equation with respect to x, we get

$$\Omega + \int \frac{dp}{\rho} + \frac{u^2 + v^2 + w^2}{2} - \frac{\partial \phi}{\partial t} = F_1(y, z, t)$$

The above equation is thus a more general equation which is applicable to both compressible as well as incompressible fluids.

If V represents the resultant velocity at any point whose components in the x, y and z directions are u, v and w, then

$$V^2 = (u^2 + v^2 + w^2)$$

and the integrated equation becomes

$$\Omega + \int \frac{dp}{\rho} + \frac{V^2}{2} - \frac{\partial \phi}{\partial t} = F_1(y, z, t) \qquad \text{... (7.12 a)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By adopting the same procedure the other two Euler's equations of motion for the y and z directions may also be integrated which may be written as

$$\Omega + \int \frac{dp}{\rho} + \frac{V^2}{2} - \frac{\partial \phi}{\partial t} = F_2(x, z, t) \quad \text{... (7.12 b)}$$

and

$$\Omega + \int \frac{\partial p}{\rho} + \frac{V^2}{2} - \frac{\partial \phi}{\partial t} = F_3(x, y, t) \quad \text{... (7.12 c)}$$

In Eqs. 7.12 (a), 7.12 (b) and 7.12 (c) it is observed that the left hand side of all these equations is same and therefore

$$F_1(y, z, t) = F_2(x, z, t) = F_3(x, y, t)$$

Since x, y and z are independent variables, the above equation will hold good only if these variables disappear from the functional terms, or in other words the arbitrary functions F_1, F_2 and F_3 are the functions of the time alone or are mere constants.

Thus the final integrated form of the Euler's equations of motion for all the three directions becomes one equation as

$$\Omega + \int \frac{dp}{\rho} + \frac{V^2}{2} - \frac{\partial \phi}{\partial t} = F(t) \quad \text{... (7.13)}$$

which contains an arbitrary function of time $F(t)$.

Equation 7.13 is known as *pressure equation.* In steady flow where there is no change in the conditions with respect to time Eq. 7.13 reduces to

$$\Omega + \int \frac{dp}{\rho} + \frac{V^2}{2} = C \quad \text{... (7.14)}$$

where C is an arbitrary constant which is to be determined by the known conditions of velocity, pressure and body force potential at some point in the flow.

If the body force exerted on the flowing fluid is only due to gravity and the z -axis is so oriented that z is measured in the vertical upward direction with reference to a datum, then we have

$$-\frac{\partial \Omega}{\partial x} = 0 \; ; -\frac{\partial \Omega}{\partial y} = 0 \; ; -\frac{\partial \Omega}{\partial z} = -g$$

since the acceleration due to gravity g is force per unit mass and it is considered to be positive when acting in the vertical downward direction. By integrating the above equation, we get

$$\Omega = gz + C_1$$

where C_1 is a constant of integration. Since at $z = 0$, $\Omega = 0$, so that $C_1 = 0$ and hence $\Omega = gz$.

Thus by substituting the value of Ω in Eq. 7.14 it becomes

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = C \quad \text{... (7.15)}$$

Equation 7.15 is known as Bernoulli's equation which is applicable for steady irrotational flow of compressible fluids. If the flowing fluid is incompressible then since the mass density ρ is independent of pressure, Eq. 7.15 becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p}{\rho}+\frac{V^2}{2}+gz = C \qquad \ldots (7.16)$$

Equation 7.16 is also known as Bernoulli's equation which is applicable for steady irrotational flow of incompressible fluids. Since the gravitational acceleration g is constant by dividing both sides of euation 7.16 by g, we get

$$\frac{p}{\rho g}+\frac{V^2}{2g}+z = C'$$

or

$$\frac{p}{w}+\frac{V^2}{2g}+z = C' \qquad \ldots (7.17)$$

since specific weight $w = \rho g$ and C' is another constant.

Equations 7.15, 7.16 and 7.17 are the energy equations, since each term in these equations represents the energy possessed by the flowing fluid. Each term in Eq. 7.15 or 7.16 represents the energy per unit mass of the flowing fluid and in Eq. 7.17 each term represents the energy per unit weight of the flowing fluid. The energy per unit weight of the fluid is expressed as N.m/N or kg(f) - m /kg(f) , that is, it has a dimension of length and therefore it is known as head. Thus in Eq. 7.17 the term $\frac{p}{w}$ is known as *pressure head* or *static head;* $\frac{V^2}{2g}$ is known as *velocity head* or *kinetic head* and z is known as *potential head* or *datum head*. The sum of the pressure head, the velocity head and the potential head is known as the total head or the total energy per unit weight of the fluid. The Bernoulli's equation 7.17 thus states that in a steady, irrotational flow of an incompressible fluid the total energy at any point is constant. In other words, if the Bernoulli's equation is applied between any two points in a steady irrotational flow of an incompressible fluid then, we get

$$\frac{p_1}{w}+\frac{V_1^2}{2g}+z_1 = \frac{p_2}{w}+\frac{V_2^2}{2g}+z_2 \qquad \ldots (7.18)$$

where the different terms with subscripts 1 and 2 correspond to the two points considered.

The sum of the pressure head and the potential head i.e., $\left(\frac{p}{w}+z\right)$ is also termed as *piezometric head.*

It is observed from Eq. 7.18 that if V_2 is greater than V_1 then the piezometric head $\left(\frac{p_2}{w}+z_2\right)$ must be less than the piezometric head $\left(\frac{p_1}{w}+z_1\right)$. However, if the two points considered lie along the same horizontal plane $z_1 = z_2$, in which case the changes in velocity cause corresponding change in the pressure.

Further, Eq. 7.18 has been derived for an ideal fluid which is non-viscous and hence there is no loss of energy. However, for the flow of real fluids since there is always some energy of the flowing fluid converted into heat due to the viscous and turbulent shear and consequently there is a certain

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

amount of energy loss. Hence for the flow of real fluids, Eq. 7.18 may be modified as

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_L \qquad ...(7.19)$$

where h_L is the loss of energy (or head) between the points under consideration.

Equation 7.19 can be applied, between any two sections of a pipe or a channel or any other passage through which a real fluid is flowing, by considering the average values of the pressure intensity and the velocity of flow at each of these sections.

(b) When the flow is steady but may not be irrotational. If the flow is rotational then also the Euler's equation of motion can be integrated along a streamline provided the flow is steady. For this Euler's equation along a streamline may first be obtained as described below.

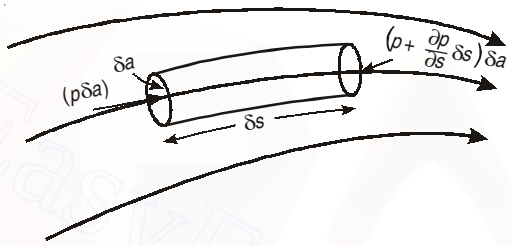


Figure 7.2 *Elementary stream filament subjected to pressure and body forces*

Consider an element of stream filament of cross-sectional area δa and length δs along a streamline as shown in Fig. 7.2. The various forces acting on the element are the body force and the pressure forces on the two ends. Let S be the component of the body force per unit mass in the direction s along the streamline. Further if ρ is the mass density of the fluid then the mass of the fluid contained in the element of the stream filament is ($\rho\, \delta a\, \delta s$). Therefore, the total component of the body force acting on the element in the direction s is equal to S ($\rho\, \delta a\, \delta s$).

If p is the pressure intensity on the left end of the element then the total pressure force acting on this end is ($p\, \delta a$).

Since the pressure p varies with the distance in the direction of flow the pressure intensity on the right end will be $\left(p + \frac{\partial p}{\partial s}\delta s\right)$.

Therefore the total pressure force on the right end of the element will be $\left(p + \frac{\partial p}{\partial s}\delta s\right)\delta a$.

The net pressure force acting on the element is therefore

$$F_{ps} = (p\delta a) - \left(p + \frac{\partial p}{\partial s}\delta s\right)\delta a$$

or

$$F_{ps} = -\frac{\partial p}{\partial s}\delta s\, \delta a$$

Now if V is the velocity of flowing fluid at the element in the direction tangential to the streamline then as explained in Chapter 6 for steady flow the acceleration in the direction tangential to the streamline is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$a_s = V\frac{\partial V}{\partial s} = \frac{1}{2}\frac{\partial V^2}{\partial s}$$

Now according to the Newton's second law of motion, in the direction of the streamline, we have

$$S(\rho\,\delta s\;\delta a) - \frac{\partial p}{\partial s}\,\delta s\;\delta a = (\rho\;\delta s\;\delta a)\left(\frac{1}{2}\frac{\partial v^2}{\partial s}\right)$$

Dividing both sides by the mass of the fluid ($\rho\,\delta s\;\delta a$) in the element and taking the limit so that the element is reduced to a point, we obtain the Euler's equation of motion in the s direction at any point on the streamline in the flowing mass of fluid as

$$S - \frac{1}{\rho}\frac{\partial p}{\partial s} = \frac{1}{2}\frac{\partial V^2}{\partial s} \quad \ldots (7.20)$$

The body force may be expressed in terms of the force potential as

$$S = -\frac{\partial \Omega}{\partial s}$$

then Eq. 7.20 becomes

$$\frac{\partial \Omega}{\partial s} + \frac{1}{\rho}\frac{\partial p}{\partial s} + \frac{1}{2}\frac{\partial V^2}{\partial s} = 0$$

Now if it is assumed that the fluid is incompressible, then integrating the above equation with respect to s, we get

$$\Omega + \frac{p}{\rho} + \frac{V^2}{2} = C \quad \ldots (7.21)$$

where C is a constant of integration, the value of which depends on the particular streamline chosen and so the constant C will in general be different for different streamlines.

However if the fluid is compressible then integrating the above equation with respect to s, we get

$$\Omega + \int\frac{\partial p}{\rho} + \frac{V^2}{2} = C \quad \ldots (7.22)$$

As stated earlier if the body force exerted on the flowing fluid is due to gravity only, then we have

$$\Omega = gz$$

Thus by substituting the value of Ω in Eq. 7.22 it becomes

$$\int\frac{\partial p}{\rho} + \frac{V^2}{2} + gz = C \quad \ldots (7.23)$$

For an incompressible fluid Eq. 7.23 becomes

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = C \quad \ldots (7.24)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equations 7.23 and 7.24 are again Bernoulli's equations for compressible and incompressible fluids respectively which are exactly same as Eqs. 7.15 and 7.16.

Thus on the basis of the above analysis it may be concluded that if the flow is irrotational then the same Bernoulli's equation is applicable to all the points in the flow field, that is for all the streamlines the value of the constant is same. However, for the rotational flow, the Bernoulli's equation is applicable only to particular streamline that is the value of the constant is different for different streamlines.

7.5 BERNOULLI'S EQUATION FROM THE PRINCIPLE OF CONSERVATION OF ENERGY

As stated earlier each term in the Bernoulli's Eq. 7.17 represents the energy possessed by the flowing fluid per unit weight of the fluid. Therefore the Bernoulli's equation may be considered as an energy equation applied to a flowing fluid. The general energy equation for the flow of fluids may be derived on the basis of the principle of conservation of energy and is essentially a complete accounting of the work done on the fluid and the resulting change of energies of the flowing fluid. Therefore, Bernoulli's equation may also be derived on the basis of the principle of conservation of energy as indicated below.

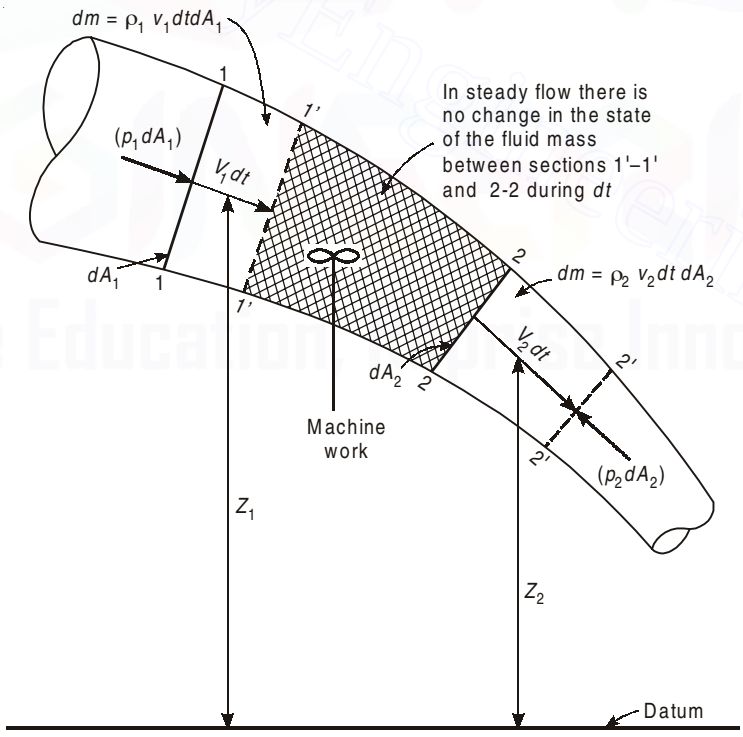


Figure 7.3 *Change of energy of flowing fluid in a stream tube*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider a free-body of flowing fluid occupying a portion of a stream tube between two arbitrarily chosen sections 1–1 and 2–2 at any instant of time t_1 as shown in Fig. 7.3. In a small interval of time dt the fluid moves a short distance to a new position 1'–1' and 2'–2' at another instant of time t_2. Therefore $(t_2 - t_1)$ equals dt.

The general energy equation describing the motion of fluid in the stream tube during dt can be expressed as follows:

$$\begin{bmatrix}\text{Work done on the fluid by}\\ \text{external forces during } dt\end{bmatrix} \pm \begin{bmatrix}\text{Mechanical work per}\\ \text{formed on the fluid}\end{bmatrix}$$

$$= \begin{bmatrix}\text{Total energy of fluid}\\ \text{between}\\ 1'-1' \text{ and } 2'-2' \text{ at } t_2\end{bmatrix} - \begin{bmatrix}\text{Total energy of fluid}\\ \text{between}\\ 1-1 \text{and } 2-2 \text{ at } t_1\end{bmatrix}$$

But

$$\begin{bmatrix}\text{Total energy of fluid}\\ \text{between}\\ 1'-1' \text{ and } 2'-2' \text{ at } t_2\end{bmatrix}$$

$$= \begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 1'-1' \text{ and } 2-2 \text{ at } t_2\end{bmatrix} + \begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 2-2 \text{ and } 2'-2' \text{ at } t_2\end{bmatrix}$$

and

$$\begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 1-1 \text{ and } 2-2 \text{ at } t_1\end{bmatrix}$$

$$= \begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 1-1 \text{ and } 1'-1' \text{ at } t_1\end{bmatrix} + \begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 1'-1' \text{ and } 2-2 \text{ at } t_1\end{bmatrix}$$

For steady flow the state of the flowing fluid in the stream tube within the region bounded between sections 1' – 1' and 2 – 2 remains unchanged with respect to time. Thus

$$\begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 1'-1' \text{ and } 2-2 \text{ at } t_2\end{bmatrix} = \begin{bmatrix}\text{Total energy of}\\ \text{fluid between}\\ 1'-1' \text{ and } 2-2 \text{ at } t_1\end{bmatrix}$$

Hence the general energy equation for the steady flow of fluid is reduced to the following form:

$$\begin{bmatrix}\text{Work done on the}\\ \text{fluid by external}\\ \text{forces during } dt\end{bmatrix} \pm \begin{bmatrix}\text{Mechanical work}\\ \text{performed on the fluid}\end{bmatrix}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \begin{bmatrix} \text{Total energy of} \\ \text{fluid between} \\ 2-2 \text{ and } 2'-2' \text{ at } t_2 \end{bmatrix} - \begin{bmatrix} \text{Total energy of} \\ \text{fluid between} \\ 1-1 \text{ and } 1'-1' \text{ at } t_1 \end{bmatrix}$$

Further when the flow of fluid is steady, the mass flow at sections 1–1 and 2–2 during time *dt* being the same, the following continuity equation holds:

$$\begin{bmatrix} \text{Fluid mass between} \\ \text{sections} \\ 1-1 \text{ and } 1'-1' \end{bmatrix} = \begin{bmatrix} \text{Fluid mass} \\ \text{between sections} \\ 2-2 \text{ and } 2'-2' \end{bmatrix}$$

Let p_1 and p_2 be the pressure intensities and V_1 and V_2 be the velocities of flow at sections 1–1 and 2–2 of the stream tube where the cross-sectional areas of the tube are dA_1 and dA_2 respectively. Then the pressure forces $(p_1 dA_1)$ and $(p_2 dA_2)$ act at sections 1–1 and 2–2 respectively. During the time interval *dt* the pressure force $(p_1 dA_1)$ acting at section 1–1 causes the liquid mass to move through a distance $(V_1 dt)$. Thus the work done by the pressure force $(p_1 dA_1)$ on the free body of the fluid in the stream tube during the time *dt* is $+ (p_1 d A_1)(V_1 dt)$. The positive sign indicates that both the pressure force and the distance through which it acts are of the same direction. Similarly the work done by the pressure force $(p_2 dA_2)$ at section 2–2 on the free-body of flowing fluid in the stream tube is $- (p_2 dA_1)(V_2 dt)$. The work done is negative because the pressure force $(p_2 d A_2)$ and the distance $(V_2 dt)$ through which it acts are of opposite direction. Therefore the net work performed by the pressure forces on the free-body of the fluid in the stream tube during the interval of time *dt* is $[(p_1 dA_1)(V_1 dt) - (p_2 dA_1)(V_2 dt)]$. Now if *dm* is the total mass of fluid flowing across section 1–1 during the time interval *dt*, or *dm* is actually the mass of fluid in the stream tube within the region between sections 1–1 and 1'–1', or between sections 2–2 and 2'–2', as shown in Fig. 7.3, and if ρ_1 and ρ_2 are the mass densities of the flowing fluid at sections 1–1 and 2–2, then

$$(V_1 dA_1)\, dt = \frac{dm}{\rho_1}; \text{ and } (V_2 dA_2)\, dt = \frac{dm}{\rho_2}$$

By substituting these values the net work done by the pressure forces on the free-body of the fluid in the stream tube becomes $[(p_1/\rho_1) - (p_2/\rho_2)]dm$. The pressure force on the boundaries of the stream tube from the adjacent fluids are everywhere normal to the surface and therefore the work done by them is zero.

In between sections, 1–1 and 2–2 there might be some mechanical device such as a pump which performs work on the fluid and puts mechanical energy into it, or there might be a machine such as a turbine on which the fluid performs the work and hence some of the energy possessed by the flowing fluid is utilized. If h_m represents the work done on the fluid by the machine or the work done by the fluid on the machine, per unit weight of the flowing fluid then the total amount of work performed on the fluid or by the fluid may be denoted by $(+h_m g dm)$ or $(-h_m g dm)$ respectively.

As shown in Fig. 7.3, with reference to an arbitrarily chosen datum, the gravitational potential energies of the fluid in the stream tube in the regions bounded by sections 1–1 and 1′–1′ and 2 – 2 and 2′–2′ are $(z_1 g dm)$ and $(z_2 g dm)$ respectively, where z_1 and z_2 are the elevations in metres of these two regions above the chosen datum. The change in gravitational potential energy of the flowing fluid in the stream tube during time *dt* is, therefore, $(z_2 - z_1)\, g dm$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The kinetic energies of the fluid in these two regions are respectively $\left(V_1^2/2\right)dm$ and $\left(V_2^2/2\right)dm$. Therefore the change in kinetic energy of the flowing fluid in the stream tube during dt is $dm\left(V_2^2 - V_1^2\right)/2$.

Thus for the steady flow of the fluid in the stream tube of Fig. 7.3, the general energy equation may be expressed in the mathematical form as follows:

$$\left(\frac{p_1}{\rho_1} - \frac{p_2}{\rho_2}\right)dm \pm h_m g dm = (z_2 - z_1)\, g dm + \frac{dm}{2}\left(V_2^2 - V_1^2\right) \quad \text{... (7.25)}$$

Dividing both sides by the common factor (gdm) and rearranging the various terms, the general energy Eq. 7.25 takes the following form:

$$\frac{p_1}{\rho_1 g} + \frac{V_1^2}{2g} + z_1 \pm h_m = \frac{p_2}{\rho_2 g} + \frac{V_2^2}{2g} + z_2 \quad \text{... (7.26)}$$

Equation 7.26 is valid for the flow of both compressible and incompressible fluids. However, if the flowing fluid is incompressible then $\rho_1 = \rho_2 = \rho$ = constant, then Eq. 7.26 becomes

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 \pm h_m = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 \quad \text{... (7.27)}$$

since specific weight $w = \rho g$.

Now if there is no mechanical device in between the sections 1–1 and 2–2, then neither any work is done on the fluid nor any work is done by the fluid and hence $h_m = 0$. Equation 7.27, then becomes

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 \quad \text{... (7.28)}$$

Since both the sections 1–1 and 2–2 are chosen arbitrarily, it follows that for any section

$$\frac{p}{w} + \frac{V^2}{2g} + z = \text{constant} \quad \text{... (7.29)}$$

which is Bernoulli's equation for the steady flow of an incompressible non-viscous fluid in a stream tube, as derived earlier. However, for the steady flow of an incompressible real fluid there is certain loss of useful energy caused by the viscous and turbulent friction. Thus if h_L represents the loss of energy per unit weight of fluid between the sections 1–1 and 2–2, Eq. 7.28 may be modified as

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_L \quad \text{... (7.30)}$$

which states that in steady flow of real fluid the total head (or total energy per N or per kg(f) of flowing fluid) at any section is equal to that at any subsequent section, plus the loss of head (or loss of energy per N or per kg(f) of flowing fluid) occurring between the two sections.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

7.6 KINETIC ENERGY CORRECTION FACTOR

In the Bernoulli's equation derived above the velocity head or the kinetic energy per unit weight of the fluid has been computed on the basis of the assumption that the velocity is uniform over the entire cross-section of the stream tube. But in the case of flow of real fluids the velocity distribution across any cross-sectional area of the flow passage is not uniform. Therefore along any cross-section the velocities of flow will be different at different points and the total kinetic energy possessed by flowing fluid at any section will be obtained by integrating the kinetic energies possessed by different fluid particles. If v is the velocity of flowing fluid at any point through any elementary area dA of the cross-section, then the mass of fluid flowing per unit time will be ($\rho v dA$) and the kinetic energy of this mass of fluid will be ($\rho v dA$) $\frac{v^2}{2}$. The total kinetic energy possessed by the flowing fluid across the entire cross-section A is

$$\int_A \rho \frac{v^3}{2} dA = \frac{w}{2g} \int_A v^3 dA \quad \text{... (7.31)}$$

By knowing the velocity distribution across any cross-section the actual kinetic energy possessed by the flowing fluid can be determined by using Eq. 7.31.

It is however more convenient to express the kinetic energy of the flowing fluid in terms of the mean velocity of flow. But the actual kinetic energy possessed by the flowing fluid is greater than that computed by using the mean velocity. Hence a factor called *kinetic energy correction factor* represented by α (Greek 'alpha') is introduced, so that the kinetic energy computed by using the mean velocity V may be expressed as $\left(\alpha \frac{w}{2g} AV^3\right)$ and it is equal to the actual total kinetic energy possessed by the flowing fluid. Thus equating the two, the value of the kinetic energy correction factor α may be obtained as

$$\alpha \frac{w}{2g} AV^3 = \frac{w}{2g} \int_A v^3 dA$$

Therefore

$$\alpha = \frac{1}{AV^3} \int_A v^3 dA \quad \text{... (7.32)}$$

Mathematically, the cube of the average is less than the average of the cubes, that is $V^3 < \frac{1}{A} \int_A v^3 dA$; the numerical value of α will always be greater than 1. The actual value of α depends on the velocity distribution at the flow section. The value of α for turbulent flow in pipes lies between 1.03 to 1.06, which is very close to 1, because in turbulent flow the velocity distribution is very close to uniform velocity distribution. However, for laminar flow in pipes the value of α is 2.

In the application of the Bernoulli's equation between any two sections if the velocity distribution is non-uniform then the kinetic energy correction factors will be required to be introduced if the kinetic energy is expressed in terms of the mean velocity at each section. Thus Eq. 7.30 is modified as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_1}{w} + \alpha_1 \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_L \quad \text{...(7.33)}$$

in which α_1 and α_2 are the energy correction factors at sections 1 and 2 respectively. However, in most of the problems of turbulent flow the value of α is nearly equal to 1, and therefore it may be assumed as one without any appreciable error being introduced.

7.7 BERNOULLI'S EQUATION FOR A COMPRESSIBLE FLUID

As explained earlier for a steady irrotational flow of a compressible fluid, Bernoulli's equation is expressed by Eq. 7.15 as

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = C$$

in which it is assumed that the kinetic energy correction factor α is equal to unity. The flow of a compressible fluid may take place with either isothermal or adiabatic changes of volume, and therefore the pressure term in the above equation may be integrated by using the perfect gas laws as indicated below.

(a) Isothermal change. For isothermal change, we have

$$\frac{p}{\rho} = K; \text{ or } \frac{p}{\rho g} = K'$$

where K and K' are constants.

By substituting the value of ρ in the above equation and integrating the pressure term, we get

$$K \log_e p + \frac{V^2}{2} + gz = C \quad \text{... (7.34 a)}$$

or $$gK' \log_e p + \frac{V^2}{2} + gz = C \quad \text{... (7.34 b)}$$

Applying the above equation between any two sections 1–1 and 2–2, we get

$$K \log_e p_1 + \frac{V_1^2}{2} + gz_1 = K \log_e p_2 + \frac{V_2^2}{2} + gz_2 \quad \text{... (7.35 a)}$$

or $$gK' \log_e p_1 + \frac{V_1^2}{2} + gz_1 = gK' \log_e p_2 + \frac{V_2^2}{2} + gz_2 \quad \text{... (7.35 b)}$$

Equation 7.35 may also be expressed as

$$K \log_e (p_1/p_2) = \frac{V_2^2}{2} - \frac{V_1^2}{2} + g(z_2 - z_1) \quad \text{... (7.35 c)}$$

or $$K' \log_e (p_1/p_2) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{... (7.35 d)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For a gas, the potential head terms z_1 and z_2 are usually small as compared with the pressure head terms and hence the same may be neglected. Equation 7.35 then simplifies to

$$K \log_e (p_1 / p_2) = \frac{V_2^2}{2} - \frac{V_1^2}{2} \quad \text{... (7.36a)}$$

or $$K' \log_e (p_1/p_2) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} \quad \text{... (7.36 b)}$$

Equation 7.35 holds good for a frictionless isothermal flow of compressible fluid for which there is no loss of energy as the fluid flows. However, if there is some loss of energy between the sections 1–1 and 2–2, represented by head loss h_L, then Eq. 7.35 may be modified as

$$K \log_e p_1 + \frac{V_1^2}{2} + gz_1 = K \log_e p_2 + \frac{V_2^2}{2} + gz_2 + gh_L \quad \text{... (7.37a)}$$

or $$K' \log_e p_1 + \frac{V_1^2}{2g} + z_1 = K' \log_e p_2 + \frac{V_2^2}{2g} + z_2 + h_L \quad \text{... (7.37b)}$$

(b) **Adiabatic change.** For adiabatic change, we have

$$\frac{p}{\rho^k} = C_1 \text{ ; or } \frac{p}{(\rho g)^k} = C'_1$$

where C_1 and C'_1 are constants and k is adiabatic exponent or adiabatic constant.

Differentiating the above equation, we get

$$\frac{dp}{d\rho} = C_1 k \rho^{k-1}; \quad \text{or} \quad \frac{dp}{\rho} = C_1 k \rho^{k-2} d\rho$$

and $$\frac{dp}{d\rho} = C'_1 g^k k \rho^{k-1}; \quad \text{or} \quad \frac{dp}{\rho} = C'_1 g^k k \rho^{k-2} d\rho$$

By substituting the value of (dp/ρ) in Eq. 7.15 and integrating the pressure term, we get

$$C_1 k \frac{\rho^{k-1}}{(k-1)} + \frac{V^2}{2} + gz = C$$

or $$C_1' g^k k \frac{\rho^{k-1}}{(k-1)} + \frac{V^2}{2} + gz = C$$

which may be expressed as

$$\frac{k}{k-1} \frac{p}{\rho} + \frac{V^2}{2} + gz = C \quad \text{... (7.38 a)}$$

or $$\frac{k}{k-1} \frac{p}{w} + \frac{V^2}{2g} + z = C' \quad \text{... (7.38 b)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Applying the above equation between any two sections 1–1 and 2–2, we get

$$\frac{k}{k-1}\frac{p_1}{\rho_1} + \frac{V_1^2}{2} + gz_1 = \frac{k}{k-1}\frac{p_2}{\rho_2} + \frac{V_2^2}{2} + gz_2 \quad \text{... (7.39 a)}$$

and

$$\frac{k}{k-1}\frac{p_1}{w_1} + \frac{V_1^2}{2g} + z_1 = \frac{k}{k-1}\frac{p_2}{w_2} + \frac{V_2^2}{2g} + z_2 \quad \text{... (7.39 b)}$$

Again for a gas the potential head terms z_1 and z_2 are usually small as compared with the pressure head terms and the same may be neglected so that Eq. 7.39 simplifies to

$$\frac{k}{k-1}\frac{p_1}{\rho_1} + \frac{V_1^2}{2} + = \frac{k}{k-1}\frac{p_2}{\rho_2} + \frac{V_2^2}{2} \quad \text{... (7.40 a)}$$

or

$$\frac{k}{k-1}\frac{p_1}{w_1} + \frac{V_1^2}{2g} = \frac{k}{k-1}\frac{p_2}{w_2} + \frac{V_2^2}{2g} \quad \text{... (7.40 b)}$$

Equation 7.39 is applicable for a frictionless adiabatic flow of compressible fluid for which there is no loss of energy as the fluid flows. However, if there is some loss of energy between the sections 1–1 and 2–2, represented by head loss h_L, then Eq. 7.39 may be modified as

$$\frac{k}{k-1}\frac{p_1}{\rho_1} + \frac{V_1^2}{2} + gz_1 = \frac{k}{k-1}\frac{p_2}{\rho_2} + \frac{V_2^2}{2} + gz_2 + gh_L \quad \text{... (7.41 a)}$$

or

$$\frac{k}{k-1}\frac{p_1}{w_1} + \frac{V_1^2}{2g} + z_1 = \frac{k}{k-1}\frac{p_2}{w_2} + \frac{V_2^2}{2g} + z_2 + h_L \quad \text{... (7.41b)}$$

7.8 PRESSURE VELOCITY REALATIONSHIP

If the two sections in a flowing fluid are lying at the same height above the assumed datum then $z_1 = z_2$ and the Bernoulli's equation applied between these sections reduces to

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g} + h_L \quad \text{... (7.42)}$$

which gives the pressure velocity relationship. Equation 7.42 is also applicable to the problems of fluid flow in which the effect of the fluid weight may be neglected. These include the problems of gas flow and the flow of liquid in horizontal pipes. If the energy loss between the two sections can be neglected then Eq. 7.42 becomes

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g} \quad \text{... (7.43)}$$

It is indicated from Eq. 7.43 that if the flow passage is such that V_2 the velocity of flow at section 2 is greater than V_1 the velocity of flow at section 1, then p_1 the pressure at section 1 will be greater than p_2 the pressure at section 2 so that Eq. 7.43 will be satisfied. In actual practice also it is observed that if at any section of flow passage due to the considerations of continuity, the velocity of flow increases then the pressure at this section reduces and vice versa.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

7.9 APPLICATIONS OF BERNOULLI'S EQUATION

Bernoulli's equation finds wide application in the solution of many problems of fluid flow. The other equation which is commonly used in the solution of the problems of fluid flow is the continuity equation. Both these equations combinedly form a very useful tool in the solution of many problems of fluid flow. Some of the simple applications of Bernoulli's equation are discussed in the following sections.

7.10 VENTURI METER

A venturi meter is a device which is used for measuring the rate of flow of fluid through a pipe. The principle of the venturi meter was first demonstrated in 1797 by Italian physicist G.B. Venturi (1746–1822), but the principle was first applied, by C. Herschel (1842–1930) in 1887, to develop the device in its present form for measuring the discharge or the rate of flow of fluid through pipes. The basic principle on which a venturi meter works is that by reducing the cross-sectional area of the flow passage, a pressure difference is created and the measurement of the pressure difference enables the determination of the discharge through the pipe.

As shown in Fig. 7.4 a venturi meter consists of (1) an *inlet section* followed by a *convergent cone*, (2) a cylindrical *throat*, and (3) a gradually *divergent cone.* The inlet section of the venturi meter is of the same diameter as that of the pipe which is followed by a convergent cone. The convergent cone is a short pipe which tapers from the original size of the pipe to that of the throat of the venturi meter. The throat of the venturi meter is a short parallel-sided tube having its cross-sectional area smaller than that of the pipe. The divergent cone of the venturi meter is a gradually diverging pipe with its cross-sectional area increasing from that of the throat to the original size of the pipe. At the inlet section and the throat i.e., sections 1 and 2 of the venturi meter, pressure taps are provided through pressure rings as shown in Fig. 7.4.

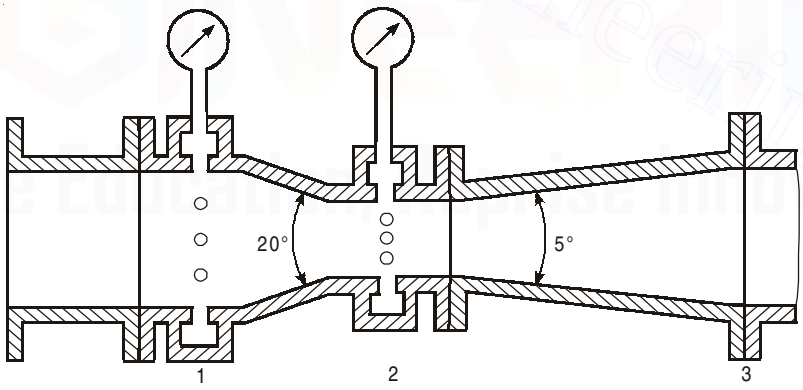


Figure 7.4 *Venturi meter*

The convergent cone of a venturi meter has a total included angle of 21° ± 1° and its length parallel to the axis is approximately equal to 2.7 ($D - d$), where D is the diameter of the inlet section and d is the diameter of the throat.

The length of the throat is equal to d. The divergent cone has a total included angle lying between

* IS : 4477–1975 provides detailed specifications for venturi meter

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

5° to 15°, (preferably about 6°). This results in the convergent cone of the venturi meter to be of smaller length than its divergent cone. This is so because from the consideration of the continuity equation it is obvious that in the convergent cone the fluid is being accelerated from the inlet section 1 to the throat section 2, but in the divergent cone the fluid is retarded from the throat section 2 to the end section 3 of the venturi meter. The acceleration of the flowing fluid may be allowed to take place rapidly in a relatively small length, without resulting in appreciable loss of energy. However, if the retardation of flow is allowed to take place rapidly in small length, then the flowing fluid will not remain in contact with the boundary of the diverging flow passage or in other words the flow separates from the walls, and eddies are formed which in turn result in excessive energy loss. Therefore, in order to avoid the possibility of flow separation and the consequent energy loss, the divergent cone of the venturi meter is made longer with a gradual divergence. Since the separation of flow may occur in the divergent cone of the venturi meter, this portion is not used for discharge measurement.

Since the cross-sectional area of the throat is smaller than the cross-sectional area of the inlet section, the velocity of flow at the throat will become greater than that at the inlet section, according to the continuity equation. The increase in the velocity of flow at the throat results in the decrease in the pressure at this section as explained earlier. As such a pressure difference is developed between the inlet section and the throat of the venturi meter. The pressure difference between these sections can be determined either by connecting a differential manometer between the pressure taps provided at these sections or by connecting a separate pressure gage at each of the pressure taps. The measurement of the pressure difference between these sections enables the rate of flow of fluid to be calculated as indicated below. For a greater accuracy in the measurement of the pressure difference the cross-sectional area of the throat should be reduced considerably, so that the pressure at the throat is very much reduced. But if the cross-sectional area of the throat of a venturi meter is reduced so much that the pressure at this section drops below the vapour pressure of the flowing liquid, then the flowing liquid may vapourise and vapour pockets or bubbles may be formed in the liquid at this section. Further liquids ordinarily contain some dissolved air which is released as the pressure is reduced and it too may form air pockets in the liquid. The formation of the vapour and air pockets in the liquid ultimately results in a phenomenon called *cavitation**, which is not desirable. Therefore, in order to avoid the phenomenon of caviation to occur, the diameter of the throat can be reduced only upto a certain limited value which is restricted on account of the above noted factors. In general, the diameter of the throat may vary from $\frac{1}{3}$ to $\frac{3}{4}$ of the pipe diameter and more commonly the diameter of the throat is kept equal to 1/2 of the pipe diameter.

Let a_1 and a_2 be the cross-sectional areas at the inlet section and the throat (i.e., sections 1 and 2) of the venturi meter respectively, at which let the pressures be p_1 and p_2 and the velocities be V_1 and V_2 respectively. Assuming that the flowing fluid is incompressible and there is no loss of energy between the sections 1 and 2 of the venturi meter, then applying Bernoulli's equation between the sections 1 and 2, we get

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 \qquad ...(i)$$

where w is the specific weight of the flowing fluid.

* For a description of cavitation phenoma, see Appendix V.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As shown in Fig. 7.5 if the venturi meter is connected in a horizontal pipe then $z_1 = z_2$ (or in this case the datum may be assumed to be passing through the axis of the venturi meter so that $(z_1 = z_2 = 0)$. The above equation then reduced to

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g}$$

or
$$\frac{p_1}{w} - \frac{p_2}{w} = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} \quad \text{... (ii)}$$

In the above expression $\frac{p_1}{w} - \frac{p_2}{w}$ is the difference between the pressure heads at sections 1 and 2 which is known as venturi head and is denoted by h. That is

$$h = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} \quad \text{... (iii)}$$

Further if Q_{th} represents the discharge through the pipe, then by continuity equation

$$Q_{th} = a_1 V_1 = a_2 V_2$$

or
$$V_1 = \frac{Q_{th}}{a_1} \text{ ; and } V_2 = \frac{Q_{th}}{a_2} \quad \ldots \text{ (iv)}$$

By substituting the values of V_1 and V_2 from Eq. (iv) in Eq. (iii), we get

$$h = \frac{Q_{th}^2}{2g}\left[\frac{1}{a_2^2} - \frac{1}{a_1^2}\right]$$

or
$$Q_{th} = \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}} \quad \text{... (7.44)}$$

Equation 7.44 gives only the theoretical discharge because the loss of energy has not been considered. But in actual practice there is always some loss of energy as the fluid flows through the venturi meter, on account of which the actual discharge will be less than the theoretical discharge given by Eq. 7.44. The actual discharge may therefore be obtained by multiplying the theoretical discharge by a factor C_d (or K) called coefficient of discharge of the venturi meter which is defined as the ratio between the actual discharge and the theoretical discharge of the venturi meter. That is

$$C_d \text{ (or } K) = \frac{Q}{Q_{th}} \text{ ; or } Q = C_d \text{ (or } K)\, Q_{th}$$

where Q represents the actual discharge. Therefore the actual discharge through the venturi meter is given by

$$\left.\begin{aligned} Q &= C_d \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}} \\ \text{or} \quad Q &= K \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}} \end{aligned}\right\} \quad \ldots \text{ (7.45)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since for a given venturi meter the cross-sectional areas of the inlet section and the throat i.e., a_1 and a_2 are fixed and therefore we may introduce constant of the venturi meter C, expressed as

$$C = \frac{a_1 a_2 \sqrt{2g}}{\sqrt{a_1^2 - a_2^2}} \qquad ...(7.46)$$

Introducing Eq. 7.46 in Eq. 7.45, we get

$$\left.\begin{aligned} Q &= C_d C\sqrt{h} \\ Q &= KC\sqrt{h} \end{aligned}\right\} \qquad \text{or} \qquad \ldots (7.47)$$

The coefficient of discharge of the venturi meter also accounts for the effects of non-uniformity of velocity distribution at sections 1 and 2. The coefficient of discharge of the venturi meter varies somewhat with the rate of flow, the viscosity of the fluid and the surface roughness, but in general for the fluids of low viscosity a value of about 0.98 is usually adopted for C_d (or K) of the venturi meter.

As shown in Fig. 7.5 if a U-tube manometer is used for measuring the difference between the pressure heads at sections 1 and 2, then for a difference in the levels of the manometric liquid in the two limbs equal to x, we have

$$\frac{p_1}{w} - \frac{p_2}{w} = h = x\left(\frac{w_m}{w} - 1\right)$$

where w_m and w are the specific weights of the manometric liquid and the liquid flowing in the venturi meter respectively. If S_m and S are respectively the specific gravities of the manometeric liquid and the liquid flowing in the venturi meter, then the expression for the venturi head becomes

$$\left(\frac{p_1}{w} - \frac{p_2}{w}\right) = h = x\left(\frac{S_m}{S} - 1\right)$$

On the other hand if an inverted U-tube manometer is used for measuring the difference between the pressure heads at sections 1 and 2, then since $S_m < S$, we have

$$\frac{p_1}{w} - \frac{p_2}{w} = h = x\left(1 - \frac{S_m}{S}\right)$$

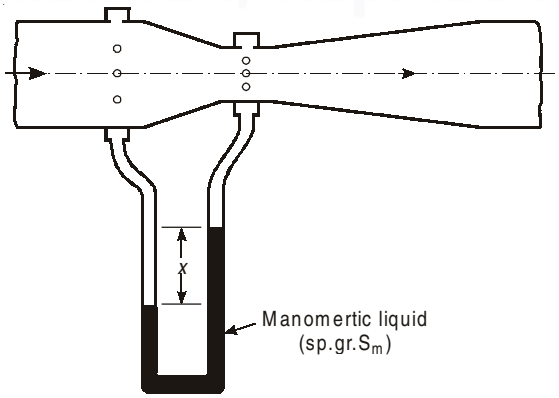


Figure 7.5 *Horizontal venturi meter with U-tube differential manometer*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Venturi meter can also be used for measuring the discharge through a pipe which is laid either in an inclined or in vertical position. Consider a venturi meter connected in an inclined pipe as shown in Fig. 7.6. Applying Bernoulli's equation between sections 1 and 2 for no loss of energy, we get

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2$$

or $$\left(\frac{p_1}{w} + z_1\right) - \left(\frac{p_2}{w} + z_2\right) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g}$$

or $$h = \frac{V_1^2}{2g} - \frac{V_1^2}{2g}$$

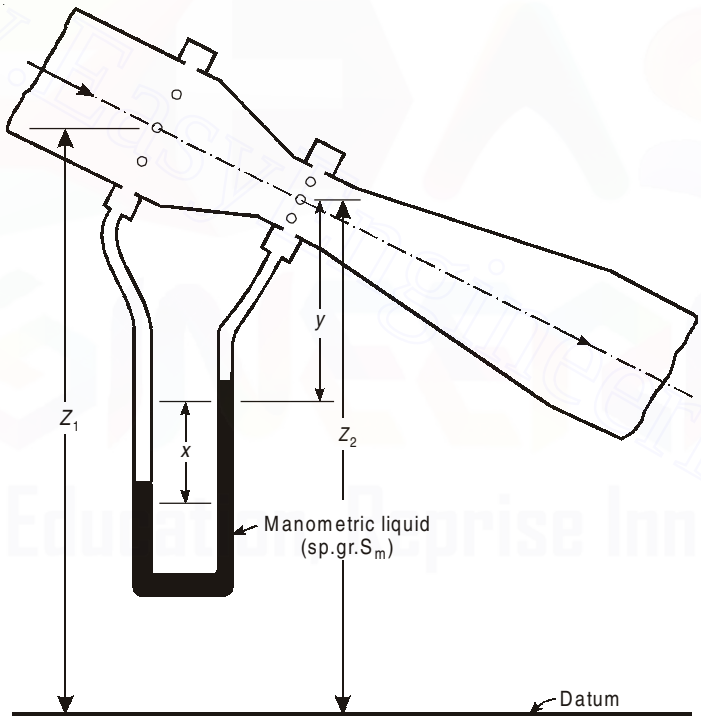


Figure 7.6 *Inclined venturi meter with U-tube manometer*

where h is again the venturi head which in this case is the difference between the piezometric heads at sections 1 and 2. Again by considering the continuity equation along with the above expression, we may obtain an expression for the discharge Q through an inclined venturi meter which will be same as Eq. 7.45 or 7.47. Thus it may be stated that even when a venturi meter is connected in an inclined pipe the discharge Q is given by Eq. 7.45 or 7.47 with the only difference that the venturi head in this case is equal to the difference between the piezometric heads at sections 1 and 2.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Furthermore as shown in Fig. 7.6 if a U-tube manometer is connected between the pressure taps at sections 1 and 2, then for a difference in the levels of the manometric liquid in the two limbs equal to x, the following manometric equation may be formed

$$\frac{p_1}{w} + (z_1 - z_2) + y + x = \frac{p_2}{w} + y + x\frac{S_m}{S}$$

or $$\left(\frac{p_1}{w} + z_1\right) - \left(\frac{p_2}{w} + z_2\right) = h = x\left(\frac{S_m}{S} - 1\right)$$

Therefore it may be stated that even in the case of venturi meter laid in an inclined position the venturi head h may be determined simply by noting the difference in the levels of the manometeric liquid in the limbs of the manometer.

In the same manner it can also be shown that Eq. 7.45 or 7.47 may also be used for determining the discharge through a venturi meter laid in a vertical position, again by considering the venturi head to be equal to the difference between the piezometric heads at the inlet section and the throat of the venturi meter.

7.11 ORIFICE METER*

An orifice meter is another simple device used for measuring the discharge through pipes. Orifice meter also works on the same principle as that of venturi meter i.e., by reducing the cross-sectional area of the flow passage a pressure difference between the two sections is developed and the measurement of the pressure difference enables the determination of the discharge through the pipe. However, an orifice meter is a cheaper arrangement for discharge measurement through pipes and its installation requires a smaller length as compared with venturi meter. As such where the space is limited, the orifice meter may be used for the measurement of discharge through pipes.

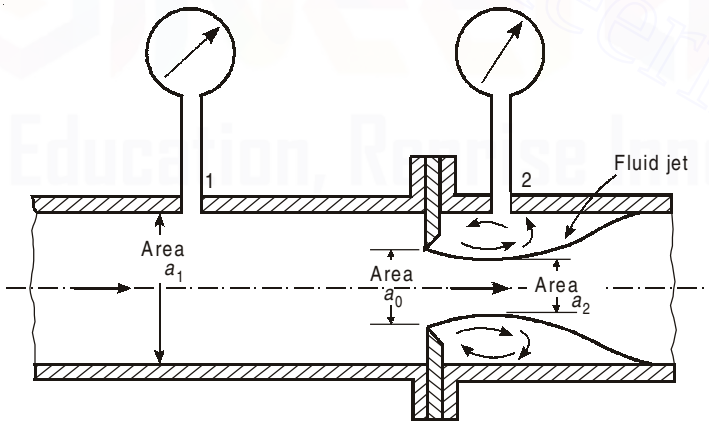


Figure 7.7 *Orifice meter*

An orifice meter consists of a flat circular plate with a circular hole called orifice, which is concentric with the pipe axis. The thickness of the plate t is less than or equal to 0.05 times the diameter of the pipe. From the upstream face of the plate the edge of the orifice is made flat for a thickness t_1 less

IS : 2952-1964 prvides detailed specification for orific meter and nozzle meter.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

then or equal to 0.02 times the diameter of the pipe and for the remaining thickness of the plate it is bevelled with the bevel angle lying between 30° to 45° (preferably 45°).

However, if the plate thickness t is equal to t_1, then no bevelling is done for the edge of the orifice. The plate is clamped between the two pipe flanges with the bevelled surface facing downstream as shown in Fig. 7.7. The diameter of the orifice may vary from 0.2 to 0.85 times the pipe diameter, but generally the orifice diamter is kept as 0.5 times the pipe diameter. Two pressure taps are provided, one at section 1 on the upstream side of the orifice plate and the other at section 2 on the downstream side of the orifice plate. The upstream pressure tap is located at a distance of 0.9 to 1.1 times the pipe diameter from the orifice plate. The position of the downstream pressure tap, however, depends on the ratio of the orifice diameter and the pipe diameter. Since the orifice diameter is less than the pipediameter as the fluid flows through the orifice the flowing stream converges which results in the acceleration of the flowing fluid in accordance with the considerations of continuity. The effect of the convergence of flowing stream extends upto a certain distance upstream from the orifice plate and therefore the pressure tap on the upstream side is provided away from the orifice plate at a section where this effect is non-existent. However, on the downstream side the pressure tap is provided quite close to the orifice plate at the section where the converging jet of fluid has almost the smallest cross-sectional area (which is known as *vena contracta*) resulting in almost the maximum velocity of flow and consequently the minimum pressure at this section. Therefore a maximum possible pressure difference exists between the sections 1 and 2, which is measured by connecting a differential manometer between the pressure taps at these sections, or by connecting a separate pressure gage at each of the pressure taps. The jet of fluid coming out of the orifice gradually expands from the vena contracta to again fill the pipe. Since in the case of an orifice meter an abrupt change in the cross-sectional area of the flow passage is provided and there being no gradual change in the cross-sectional area of the flow passage as in the case of a venturi meter, there is a greater loss of energy in an orifice meter than in a venturi meter.

Let p_1, p_2 and V_1,V_2 be the pressures and velocities at sections 1 and 2 respectively. Then for an incompressible fluid, applying Bernoulli's equation between the sections 1 and 2 and neglecting the losses, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 \qquad ...(i)$$

or $$\left(\frac{p_1}{w} + z_1\right) - \left(\frac{p_2}{w} + z_2\right) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g}$$

or $$h = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} \qquad ...(ii)$$

where h is the difference between the piezometric heads at sections 1 and 2. However, if the orifice meter is connected in a horizontal pipe then $z_1 = z_2$, in which case h will represent the difference between the pressure heads at sections 1 and 2.

From Eq. (ii) above, we obtain

$$V_2 = (2gh + V_1^2)^{1/2} \qquad ...(iii)$$

Since in deriving the above expression the losses have not been considered this expression gives the theoretical velocity of flow at Section 2. In order to obtain the actual velocity at Section 2 it must

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

be multiplied by a factor C_v called coefficient of velocity which is defined as the ratio between the actual velocity and the theoretical velocity. Thus the actual velocity of flow at section 2 is obtained as

$$V_2 = C_v (2gh + V_1^2)^{1/2} \quad ...(iv)$$

Further if a_1 and a_2 are the cross-sectional area of the pipe at section 1 and that of the jet at Section 2 respectively and Q represents the actual discharge through the pipe, then by continuity equation

$$Q = a_1 V_1 = a_2 V_2 \quad ...(v)$$

The area of the jet a_2 at section 2 (i.e., at vena contracta) may be related to the area of the orifice a_0 by the following expression

$$a_2 = C_c a_0$$

where C_c is known as the coefficient of contraction which is defined as the ratio between the area of the jet at vena contracta and the area of the orifice. Thus introducing the value of a_2 in Eq. (v), we get

$$V_1 = V_2 C_c \frac{a_0}{a_1}$$

By substituting the value of V_1 in Eq. (iv), we get

$$V_2 = C_v \left(2gh + V_2^2 C_c^2 \frac{a_0^2}{a_1} \right)^{\frac{1}{2}}$$

Solving for V_2, we get

$$V_2 = C_v \left\{ \frac{2gh}{1 - C_v^2 C_c^2 (a_0^2 / a_1^2)} \right\}^{\frac{1}{2}}$$

Now $Q = a_2 V_2 = C_c a_0 V_2$

and $C_c C_v = C_d$

$$\therefore \quad Q = \frac{C_d a_0 (2gh)^{\frac{1}{2}}}{\left\{1 - C_d^2 \left(a_0^2 / a_1^2\right)\right\}^{1/2}}$$

where C_d is the coefficient of discharge of the orifice. It is usual to simplify the above expression for the discharge through the orifice meter by using a coefficient C expressed as

$$C = C_d \frac{\left\{1 - (a_0^2 / a_1^2)\right\}^{1/2}}{\left\{1 - C_d^2 (a_0^2 / a_1^2)\right\}^{1/2}}$$

so that

$$\left.\begin{aligned} Q &= \frac{C a_0 (2gh)^{1/2}}{\{1 - (a_0^2 / a_1^2)\}^{1/2}} \\ \text{or} \quad Q &= \frac{C a_0 a_1 (2gh)^{1/2}}{\{a_1^2 - a_0^2\}^{1/2}} \end{aligned}\right\} \quad ...(7.48)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 7.48 gives the discharge through an orifice meter which is similar to Eq. 7.45 which gives the discharge through venturi meter. The coefficient C introduced in Eq. 7.48 may be considered as the coefficient of discharge of an orifice meter. The coefficient of discharge for an orifice meter is much smaller than that for a venturi meter. This is because in the case of an orifice meter there are no gradual converging and diverging flow passages as in the case of a venturi meter, which results in a greater loss of energy and consequent reduction of the coefficient of discharge for an orifice meter.

7.12 NOZZLE METER OR FLOW NOZZLE

A nozzle meter or flow nozzle is also a device used for measuring the discharge through pipes. As shown in Fig. 7.8 a nozzle meter (or flow nozzle) consists of a streamlined convergent nozzle through which the fluid is gradually accelerated. Therefore a nozzle meter is essentially a venturi meter with the divergent part omitted, and hence the basic equations are the same as those for venturi meter. On the downstream of the throat of the nozzle, there being no divergent cone, there is a greater dissipation of energy than for a venturi meter. But since the coefficient of discharge usually does not depend on what happens beyond the throat, it is almost the same as that for a venturi meter.

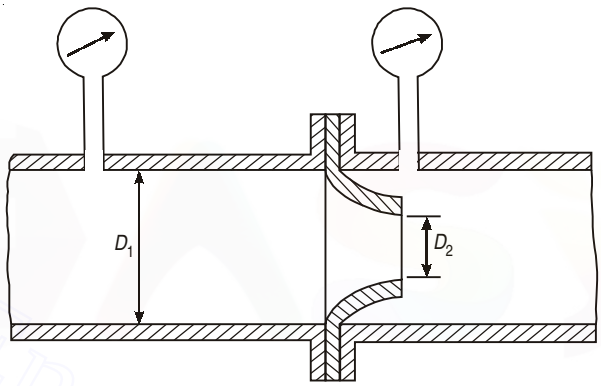


Figure 7.8 *Nozzle meter* (*or Flow nozzle*)

7.13 OTHER FLOW MEASUREMENT DEVICES

Besides the above described devices there are some more devices used for measuring discharge through pipes. These devices are *Rotameter* and *Elbow Meter* (or *Pipe-Bend Meter*) which are described below.

Rotameter. The rotameter also known as variable-area meter is shown in Fig. 7.9. It consists of a vertical transparent conical tube in which there is a rotor or float having a sharp circular upper edge. The rotor has grooves on its head which ensure that as liquid flows past, it causes the rotor to rotate about its axis. The rotor is heavier than the liquid and hence it will sink to the bottom of the tube when the liquid is at rest. But as the liquid begins to flow through the meter, it lifts the rotor until it reaches a steady level corresponding to the discharge. This rate of flow of liquid can then be read from graduations engraved on the tube by prior calibration, the sharp edge of the float serving as a pointer. The rotating motion of the float helps to keep it steady. In this condition of equilibrium, the hydrostatic and dynamic thrusts of the liquid on the under side of the rotor will be equal to the hydrostatic thrust on the upper side, plus the apparent weight of the rotor.

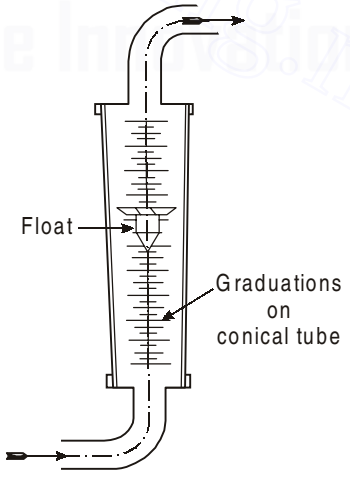


Figure 7.9 *Rotameter*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Elbow Meter (or Pipe-bend Meter). An elbow meter (or pipe-bend meter) consists of a simple 90° pipe bend provided with two pressure taps, one each at the inside and the outside of the bend, as shown in Fig. 7.10. Its operation is based on the fact that as liquid flows round a pipe bend its pressure increases with the radius, due to approximately free vortex conditions being developed in the bend. As such a pressure difference is produced on the inside and outside of the bend which is used as a measure of the discharge. The pressure taps are connected to a differential manometer to measure the differential pressure head h. The discharge Q may then be computed by the following expression

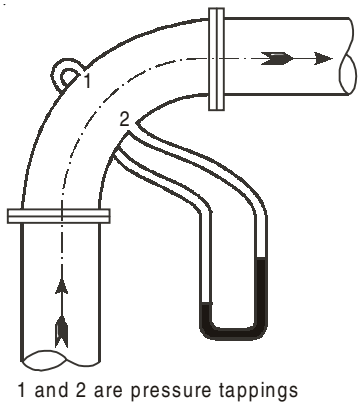


Figure 7.10 *Elbow meter*

$$Q = C_d A \sqrt{2gh}$$

where C_d is the coefficient of discharge of the elbow meter and A is its cross-sectional area. The coefficient of discharge C_d depends mainly on the ratio R/c (where R is the radius of the axis of the bend and c is the radius of the pipe), and its value can be obtained by calibration. The main advantage associated with an elbow meter is that it entails no additions or alterations to an existing pipe system, except for the drilling of pressure taps, and if suitably calibrated it can be used for precision measurements.

7.14 PITOT TUBE

A pitot tube is a simple device used for measuring the velocity of flow. The basic principle used in this device is that if the velocity of flow at a particular point is reduced to zero, which is known as *stagnation point*, the pressure there is increased due to the conversion of the kinetic energy into pressure energy, and by measuring the increase in the pressure energy at this point the velocity of flow may be determined. It is named in honour of its originator Henri de Pitot (1695–1771), a French engineer who in 1732 adopted this principle for measuring the velocities in the River Seine.

In its simplest form a pitot tube consists of a glass tube, large enough for capillary effects to be negligible, and bent at right angles. A single tube of this type may be used for measuring the velocity of flow in an open channel. The tube is dipped vertically in the flowing stream of fluid with its open end A, directed to face the flow, and the other open end projecting above the fluid surface in the stream as shown in Fig. 7.11. The fluid enters the tube and the level of the fluid in the tube exceeds that of the fluid surface in the surrounding stream. This is so because the end A of the tube is a stagnation point where the fluid is at rest, and the fluid approaching the end A divides at this point and passes around the tube. Since at the stagnation point the kinetic energy is converted into the pressure energy, the fluid in the tube rises above the surrounding fluid surface by a height which corresponds to the velocity of flow of fluid approaching the end A of pitot tube. The pressure at the stagnation point is known as *stagnation pressure.*

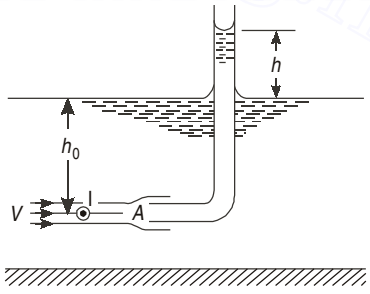


Figure 7.11 *Simple pitot tube*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider a point 1 slightly upstream of end A and lying along the same horizontal plane in the flowing stream where the velocity of flow is V. Now if the points 1 and A are at a vertical depth of h_0 below the free surface of fluid in the stream and h is the height of the fluid raised in the pitot tube above the free surface, then applying Bernoulli's equation between the points 1 and A and neglecting the loss of energy, we get

$$h_0 + \frac{V^2}{2g} = h_0 + h$$

In the above expression $(h_0 + h)$, is the stagnation pressure head at point A, which consists of two parts viz., the static pressure head h_0 and the dynamic pressure head h. By simplifying the expression, we get

$$\frac{V^2}{2g} = h;$$

or
$$V = \sqrt{2gh} \qquad ...(7.49)$$

Equation 7.49 indicates that the dynamic pressure head h is proportional to the square of the velocity of flow in the stream at the point close to the end A of the pitot tube. Thus the velocity of flow at any point in the flowing stream may be determined by dipping the pitot tube to the required point and measuring the height h of the fluid raised in the tube above the free surface. However, the velocity of flow given by Eq. 7.49 is somewhat more than the actual velocity of flow, because in deriving the above equation no loss of energy has been considered. Moreover, when the flow is highly turbulent the pitot tube records a value of h which is higher than that corresponding to the mean velocity of flow in the direction of the tube axis. As such in order to take into account the errors which may creep in due to the above noted factors the actual velocity of flow may be obtained by introducing a coefficient C (or C_v) called pitot tube coefficient, so that the actual velocity of flow is given by

$$V = C\sqrt{2gh} \qquad ...(7.50)$$

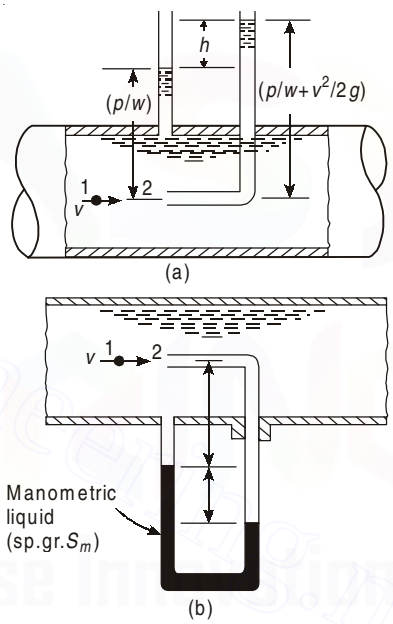


Figure 7.12 *Pitot tube used for measuring velocity in pipes*

A probable value for the coefficient of the pitot tube, C is 0.98. However, the actual value of the coefficient C for a pitot tube may be determined by calibration.

When a pitot tube is used for measuring the velocity of flow in a pipe or any other closed conduit then the pitot tube may be inserted in the pipe as shown in Fig. 7.12. Since a pitot tube measures the stagnation pressure head (or the total head) at its dipped end, the static pressure head is also required to be measured at the same section where the tip of the pitot tube is held, in order to determine the dynamic pressure head h. For measuring the static pressure head a pressure tap (or a static orifice) is provided at this section to which a piezometer may be connected as shown in Fig. 7.12 (a). Alternatively the dynamic pressure head may also be determined directly by connecting a suitable differential manometer between the pitot tube and the pressure tap meant for measuring the static pressure, as shown in Fig. 7.12 (b)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider point 1 slightly upstream of the stagnation point 2 as shown in Fig. 7.12 (b). Applying Bernoulli's equation between the points 1 and 2, we get

$$\frac{p_1}{w} + \frac{V^2}{2g} = \frac{p_2}{w} \qquad ...(i)$$

where p_1 and p_2 are the pressure intensities at points 1 and 2, V is the velocity of flow at point 1 and w is the specific weight of the fluid flowing through the pipe. Obviously p_1 is the static pressure and p_2 is the stagnation pressure. The equation for the pressure through the manometer in metres of water may be written as

$$\frac{p_1}{w} S + yS + xS_m = (y + x)S + \frac{p_2}{w} S$$

where S and S_m are the specific gravities of the fluid flowing in the pipe and the manometric liquid respectively. By simplifying

$$\frac{p_2}{w} - \frac{p_1}{w} = x\left(\frac{S_m}{S} - 1\right) \qquad ....(ii)$$

After substituting for $[(p_2/w) - (p_1/w)]$ in Eq. (i) and solving for V,

$$V = \sqrt{2gx\left(\frac{S_m}{S} - 1\right)} \qquad ...(7.51)$$

Again introducing the coefficient of the pitot tube C, the actual velocity of flow is given by

$$V = C\sqrt{2gx\left(\frac{S_m}{S} - 1\right)} \qquad ...(7.52)$$

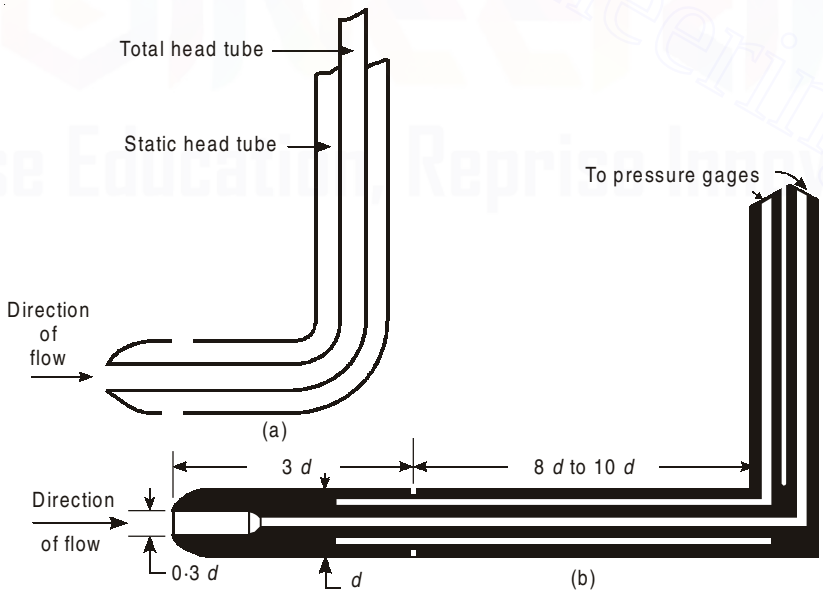


Figure 7.13 (*a*) *Pitot-static tube,* (*b*) *Prandtl pitot tube*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The tubes recording static pressure and stagnation pressure are frequently combined into one instrument known as a *pitot-static tube*, which is shown in Fig. 7.13. In this the 'static' tube surrounds the 'total head' tube, and two or more small holes are drilled radially through the outer wall into the annular space. For a pitot-static tube also the same equations noted above may be used to determine the velocity of flow.

Pitot-static tubes with standard proportions have been developed which give very accurate results. A particular form of a pitot-static tube is Prandtl pitot tube shown in Fig. 7.13 (b) which has a blunt nose and it has been so designed that the disturbances due to nose and leg cancel thereby having a value of C equal to unity. However, for the pitot static tubes of other proportions, the coefficient C may be determined by calibration.

7.15 FREE LIQUID JET

A jet of liquid issuing from a nozzle in atmosphere is known as free liquid jet. Under the action of gravity the liquid jet traverses a parabolic path known as trajectory. A free liquid jet is a particular case of steady curvilinear flow of a liquid with a free surface in which at all points the pressure is atmospheric. Accordingly, Bernoulli's equation may be applied to the whole trajectory of the jet if the air resistance is neglected. Since the pressure head is equal to zero at every point along the trajectory, the term for pressure head disappears from the Bernoulli's equation and therefore the sum of velocity head and potential head will be constant at all points along the trajectory.

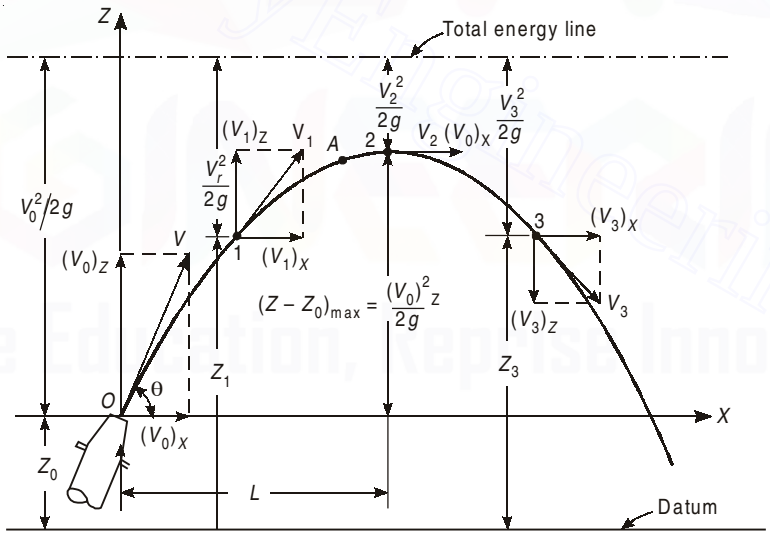


Figure 7.14 *Free liquid jet*

Consider a jet issuing from a nozzle as shown in Fig. 7.14. Let V_0 be the initial velocity of issuing jet at O and θ be the angle, which the issuing jet makes with the horizontal. As the trajectory of a jet is actually a streamline, the velocity at any point on the trajectory of the jet will be tangential to the trajectory. As such the components of V_0 along x and z directions $(V_0)_x$ and $(V_0)_z$ respectively may be expressed as

$$(V_0)_x = V_0 \cos \theta$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$(V_0)_z = V_0 \sin\theta$$

Now if z_0 is the height of point 0 above the assumed datum then according to Bernoulli's equation, we have

$$\frac{V_0^2}{2g} + z_0 = \text{constant}$$

Further if V_1, V_2 and V_3 are the velocities at points 1, 2 and 3 along the jet, which are at heights z_1, z_2 and z_3 respectively above the assumed datum, then according to Bernoulli's equation for no losses, we get

$$\frac{V_0^2}{2g} + z_0 = \frac{V_1^2}{2g} + z_1 = \frac{V_2^2}{2g} + z_2 = \frac{V_3^2}{2g} + z_3$$

The velocities V_1, V_2 and V_3 etc., can also be resolved along x and z directions. At the topmost point 2 of the trajectory the velocity V_2 will be acting parallel to the x direction and therefore its components in the z direction will be equal to zero. Moreover, there is no acceleration in x direction, the velocity component in the x direction is same at all the points along the jet, that is,

$$(V_0)_x = (V_1)_x = V_2 = (V_3)_x \text{ and so on.}$$

Further it is observed that from point 0 to 2, the z component of the velocity decreases, at point 2 it is equal to zero and from point 2 to 3 it increases.

The basic equations of projectile motion may be used to determine the velocity components at any point along the trajectory. Thus if V is the velocity and V_x and V_z are its components in the x and z directions respectively at any point A on the trajectory which is at a height z above the assumed datum, then we have

$$V_x = (V_0)_x \quad \text{...(i)}$$

and $$V_z = (V_0)_z - gt \quad \text{...(ii)}$$

Further the coordinates of the trajectory are expressed as follows:

$$x = (V_0)_x t \quad \text{...(iii)}$$

$$(z - z_0) = (V_0)_z t - \frac{1}{2} g t^2 \quad \text{...(iv)}$$

where t is the time elapsed after the liquid jet leaves the nozzle. By combining Eqs (ii) and (iv), we get

$$V_z^2 = (V_0)_z^2 - 2g(z - z_0) \quad \text{...(v)}$$

For point 2 since z component of velocity is zero, from Eq. (v) the maximum vertical elevation of the jet profile is obtained as

$$(z - z_0)_{\max} = \frac{(V_0)_z^2}{2g} \quad \text{...(7.53)}$$

By eliminating t between equation (iii) and (iv) and combining the resulting equation with Eq. 7.53 it can be shown that the horizontal distance of point 2 from point 0 is obtained as

$$L = \frac{(V_0)_x (V_0)_z}{g} = \frac{V_0^2 \sin 2\theta}{2g} \quad \text{...(7.54)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As can be seen from Eq. 7.54 the distance L will be maximum for the given velocity when angle θ is equal to 45°.

In Fig. 7.14 the total energy (or head) line is also shown which is horizontal since there are no losses. Obviously the total energy line is at a distance of $\frac{V_2^2}{2g}$ $\left[\text{which is also equal to } \frac{(V_0)_x^2}{2g}, \frac{(V_1)_x^2}{2g} \text{ and } \frac{(V_3)_x^2}{2g}\right]$ above point 2.

7.16 VORTEX MOTION

Bernoulli's equation may also be applied to the problems of vortex motion. A rotating mass of fluid is known as *vortex* and the motion of rotating mass of fluid is known as *vortex motion*. The vortex motion is of two types viz., free vortex motion and forced vortex motion.

A *free vortex motion* is that in which the fluid mass rotates without any external force being impressed on it. In this motion the whole of the moving mass of fluid rotates either by virtue of some internal

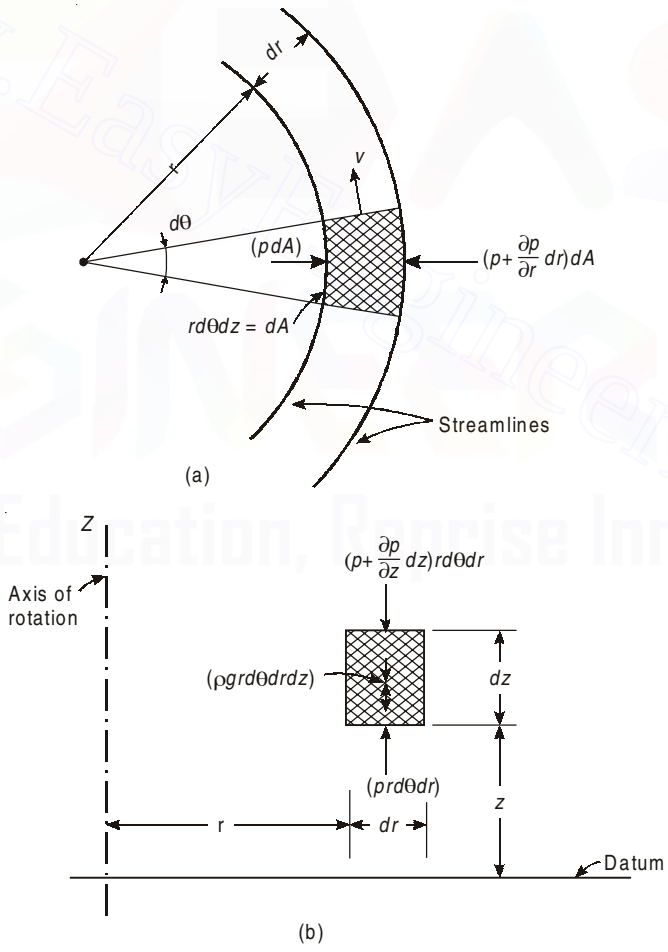


Figure 7.15 *Vortex motion: (a) plan view of a vortex motion (b) elevation of a vortex motion*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

action or due to rotation previously imparted to it. Hence in a free vortex motion no expenditure of energy from any external source takes place. Some of the examples of a free vortex motion are: a whirlpool in a river; the flow of a liquid drained through an outlet provided in the bottom of a shallow vessel such as a wash basin or a bath tub draining water through an outlet at a bottom; flow around a circular bend in a pipe system ; flow of liquid in a centrifugal pump casing after it has left the impeller: flow of water in a turbine casing before it enters the guide vanes, etc.

A *forced vortex motion* is that in which the fluid mass is made to rotate by means of some external source of power, which exerts a constant torque on the fluid mass, thereby causing the whole mass of fluid to rotate at constant angular velocity ω. As such in a forced vortex motion there is always a constant external torque required to be applied to the fluid mass resulting in an expenditure of energy. A most common example of a forced vortex motion is that of a vertical cylinder containing liquid rotated about its central axis with a constant angular velocity. Some of the other examples of forced vortex motion are the flow of liquid inside the impeller of a centifugal pump; flow of water in the runner of a turbine etc.

A vortex motion may also be characterised as cylindrical vortex motion and spiral vortex motion. A *cylindrical vortex motion,* is that in which the fluid mass rotates in concentric circles i.e., this motion may be described by concentric circular streamlines. A *spiral vortex motion* is a combination of cylindrical vortex motion and radial flow that is, when a cylindrical vortex motion is superimposed over the radial flow, then the resulting vortex motion is known as spiral vortex motion. All these types of vortex motions can exist independent of each other so that any of the four types of the combination of vortex motions is possible, viz., (a) Cylindrical free vortex; (b) Cylindrical forced vortex; (c) Spiral free vortex; and (d) Spiral forced vortex. In the following paragraphs different types of vortex motions have been analysed.

Fundamental differential equations representing the variation of pressure in a cylindrical free and forced vortex motion may first be developed. In a cylindrical vortex motion the fluid particles rotate along circular path at a constant speed. When the fluid particles move at a constant speed along a curved path, the pressure within the fluid mass on a horizontal plane varies in the radial direction only. This variation in the fluid pressure may be determined by considering the dynamic equilibrium of an element of fluid mass within a stream tube moving in a curved path on a horizontal plane as shown in Fig. 7.15. The fluid element of mass ($\rho dA dr$)is located at a distance r from the centre of its curved path where its tangential velocity is v. Since the magnitude of the tangential velocity is constant, there exists no variation of pressure in the tangential direction. However, on the two radial faces there exists a difference in the pressure forces (pdA) and $\left[\left(p+\frac{\partial p}{\partial r}dr\right)dA\right]$ due to the centrifugal force on the fluid element. Therefore

$$\left(p+\frac{\partial p}{\partial r}dr\right)dA - pdA = \rho dA\, dr\, \frac{v^2}{r}$$

or
$$\frac{\partial p}{\partial r} = \rho\frac{v^2}{r} \qquad \ldots (7.55)$$

If there is no acceleration (other than gravity) in the vertical direction then considering the equilibrium of fluid element in the vertical direction we have from Fig. 7.15 (b)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$prd\,\theta\,dr - \left(P + \frac{\partial p}{\partial r}dr\right) rd\,\theta\,dr = \rho g\, rd\theta\, drdz$$

or
$$\frac{\partial p}{\partial z} = -\rho g \qquad \ldots (7.56)$$

Equations 7.55 and 7.56 are the fundamental relationships which may be used for studying the flow characteristics in vortex motion. It may be noted that Eq. 7.55 and 7.56 are same as equations 5.17 and 5.18 derived in Chapter 5.

(a) Free vortex motion. As stated earlier in free vortex motion no external torque is required to be exerted on the fluid mass. Therefore the rate of change of angular momentum of the flow must be zero. Thus again referring to the fluid element in Fig. 7.15, if m is the mass of the fluid element then following equation of angular momentum of flow may be obtained,

$$\frac{\partial (mvr)}{\partial t} = 0$$

which upon integrating, becomes

$$vr = \text{constant} = C \qquad \ldots (7.57)$$

It is thus seen from Eq. 7.57 that the velocity of flow in a free vortex motion varies inversely with the radial distance from the centre of vortex motion; that is when $r \to \infty$, $v \to 0$, and for $r \to 0$, $v \to \infty$. . The point where the velocity v becomes infinite is called *singular point*. This condition for the velocity of flow to approach infinitely large magnitude at the centre of a free vortex motion is however not developed in actual practice as explained below.

For a free vortex motion the circulation around any streamline may be calculated as $\Gamma = (2\pi r)v = 2\pi C$, which is independent of r. Hence the circulation around all the streamlines of a free vortex motion is constant. Further it may be shown that the flow-field of a free vortex motion is everywhere irrotational except at the axis and therefore the free vortex motion is also called irrotational vortex motion. It is thus observed that for a free vortex motion although the circulation around the various streamlines is not equal to zero yet the motion is irrotational. This paradoxical situation may however be made clear as follows. If the axis which is a singular point in this case is separated from the rest of the flow field by enclosing it within an infinitesimal area surrounded by a streamline considered at an infinitesimal distance from the axis then since the circulation around the various streamlines is constant, it may be readily shown that the net circulation for the entire flow-field outside the infinitesimal area is equal to zero which thus satisfies the condition for an irrotational motion. Within the infinitesimal area close to the axis the flow is rotational. The foregoing analysis of a free vortex motion is valid only for ideal fluids. In the case of a real fluid, in the central region of a free vortex motion the viscous effects become quite predominant, due to which a central core of fluid with rotational motion is developed in which the fluid tends to rotate as a solid body with velocity proportional to the radius as in the case of a forced vortex as indicated later. Farther away from the central region where the viscous effects are negligible irrotational motion is developed. Equation 7.57 then applies only to the region farther away from the central region of a free vortex motion. As such in the case of real fluids a combination of free and forced vortex motion frequently occurs which is known as *Rankine vortex motion*. The motion of air mass in a tornado is found to approximate this type of combined vortex motion.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

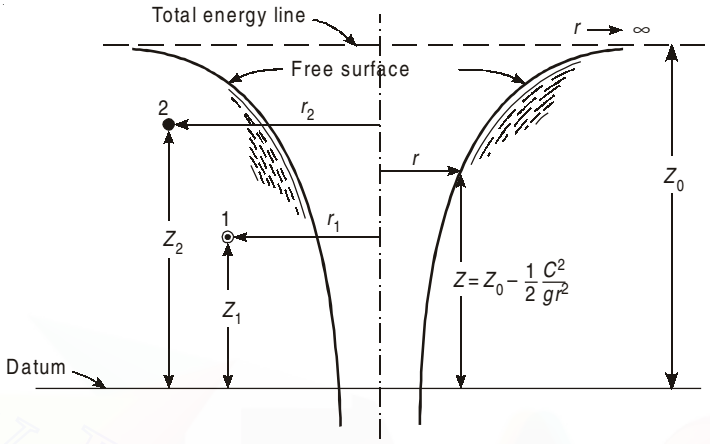


Figure 7.16 *Free vortex motion*

The irrotationality of a free-vortex motion may also be visualized by developing a free vortex motion with a free surface on which small particles of some light-weight material capable of floating are placed. It will be seen that as these particles move with the fluid they do not rotate about their axes, which is indicated by a line marked on the particle remaining parallel to itself during the different positions of the particle.

By introducing Eq. 7.57 in Eq. 7.55, the radial pressure gradient for a free vortex motion becomes

$$\frac{\partial p}{\partial r} = \rho \frac{C^2}{r^3} \qquad \text{... (7.58)}$$

The distribution of the fluid pressure in a free vortex motion is obtained by integrating both Eqs. 7.56 and 7.58 between any two points 1 and 2 in the fluid (Fig. 7.16) and their results combined in the following form :

$$\frac{p_2 - p_1}{\rho} = \frac{1}{2} C^2 \left(\frac{1}{r_1^2} - \frac{1}{r_2^2} \right) + g (z_1 - z_2) \qquad \ldots \text{ (7.59)}$$

Again from Eq. 7.57, $vr = v_1 r_1 = v_2 r_2 = C$, the above expression can be rearranged and written as

$$\frac{p_1}{\rho} + \frac{v_1^2}{2} + g z_1 = \frac{p_2}{\rho} + \frac{v_2^2}{2} + g z_2$$

which indicates that Bernoulli's equation for the flow of an ideal fluid also applies to the free vortex motion. Since the points 1 and 2 have been chosen arbitrarily, this equation thus provides an additional proof of the irrotationality of a free vortex motion.

The profile of the free surface of a free vortex motion is obtained by considering $p_1 = p_2 = p_a = 0$ in Eq. 7.59. Thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(z_2 - z_1) = \frac{1}{2}\frac{C^2}{g}\left(\frac{1}{r_1^2} - \frac{1}{r_2^2}\right) \quad \text{... (7.60)}$$

or
$$z = z_0 - \frac{1}{2}\frac{C^2}{gr^2} \quad \text{... (7.61)}$$

as shown in Fig. 7.16 ; here p_a denotes atmospheric pressure.

(b) Forced vortex motion. As stated earlier forced vortex motion occurs when a constant torque is applied to the fluid mass. This type of motion may be developed if a cylinder containing some liquid is rotated about its vertical axis at a constant angular velocity ω. The liquid inside the cylinder is forced to rotate with the cylinder. In this case also (which has already been dealt with in Chapter 5, Sec. 5.5) the streamlines are concentric circles and the velocity v of any liquid particle at a distance r from the axis of rotation may be expressed as

$$v = r\omega \quad \text{...(7.62)}$$

Equation 7.62 therefore indicates that in a forced vortex motion the velocity of flow is directly porportional to its radial distance from the axis of rotation.

By substituting the value of v in Eq. 7.55, the pressure gradient in the radial direction for a forced vortex motion becomes

$$\frac{\partial p}{\partial r} = \rho r\omega^2 \quad \text{...(7.63)}$$

Again the distribution of the fluid pressure in a forced vortex motion is obtained by integrating both the Eqs. 7.56 and 7.63 between any two points 1 and 2 in the fluid and their results combined in the following form:

$$\frac{p_2 - p_1}{\rho} = \frac{1}{2}\omega^2\left(r_2^2 - r_1^2\right) + g\left(z_1 - z_2\right) \quad \text{...(7.64)}$$

Equation 7.64 shows that, in a forced vortex motion, (1) on any horizontal plane (i.e., $z_1 = z_2$) the fluid pressure increases with the radial distance from the centre of vortex motion; (2) surfaces of constant pressure (i.e., $p_1 = p_2$) are paraboloids of revolution; and (3) the free surface (i.e., $p_1 = p_2 = p_a$) is a special surface of constant pressure and is also a paraboloid of revolution.

The forced vortex motion is essentially a rotational motion because every fluid particle in such a motion is also found to rotate about its own axis as it moves along the curved path. Mathematically also it can be shown that a forced vortex motion is rotational. Therefore in a forced vortex motion the Bernoulli's equation may although be applied to the same streamline, it cannot be applied between points on different streamlines, which is also indicated by Eq. 7.64.

As stated earlier a *spiral vortex motion* is a combination of a cylindrical vortex motion and radial flow. Hence before analysing spiral vortex motion it is essential to discuss the radial flow, which is described in the next section.

7.17 RADIAL FLOW OR RADIAL MOTION

When a fluid flows in radial direction, so that the pressure and velocity at any point in the flow varies with respect to the radial distance of that point from the central axis, then the flow is designated as radial flow. For producing radial flow consider two horizontal circular plates of same diameter

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

placed one above the other at a small distance B apart, as shown in Fig. 7.17. The bottom plate is provided with an opening at its centre, to which is attached, a vertical pipe of the same diameter as that of the opening. Through the central pipe let some liquid be allowed to flow up continuously,

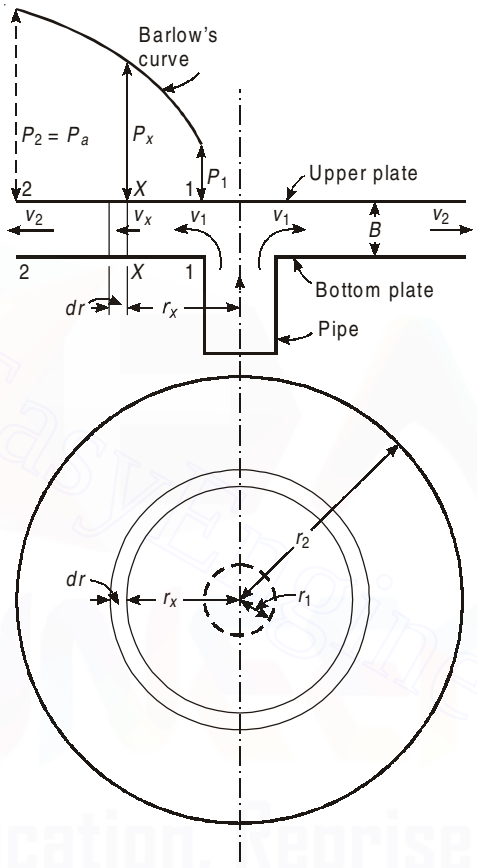


Figure 7.17 *Radial flow*

which will flow radially outward between the plates, thereby developing the radial flow of liquid. Consider sections 1–1 and 2–2 in the radial flow as shown in Fig. 7.17. Section 1-1 is assumed to be located where the radial flow of liquid just commences i.e., at this section the liquid just enters from the vertical pipe into the passage between the plates; and section 2–2 is just at the outer periphery of the plates where the liquid is discharged into atmospheric air. Let r_1 and r_2 be the radial distances of sections 1–1 and 2–2 respectively from the central axis; and v_1 and v_2 be the radial velocities of flow of liquid at sections 1-1 and 2–2 respectively. Now if Q is discharge of liquid through any section then from continuity equation the following relation between the velocity of flow and the radial distance may be obtained.

$$Q = (2\pi r_1 B \times v_1) = (2\pi r_2 B \times v_2)$$
$$= (2\pi r_x B \times v_x)$$

or
$$v_1 r_1 = v_2 r_2 = v_x r_x = \text{constant} \quad \ldots (7.65)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where r_x is the radial distance of any section X–X shown in Fig. 7.17 and v_x is the radial velocity of flow at section X–X.

Now if p_1, p_2 and p_x are the pressure intensities at sections 1–1, 2–2 and X–X respectively then neglecting the loss of head and applying Bernoulli's equation between these sections which are located at the same elevation above the datum, the following expression may be obtained

$$\frac{p_1}{w} + \frac{v_1^2}{2g} = \frac{p_2}{w} + \frac{v_2^2}{2g} = \frac{p_x}{w} + \frac{v_x^2}{2g} = H \quad \ldots (7.66)$$

where w is specific weight of the flowing liquid and H is constant, the value of which may be computed by the values of either p_1 and v_1 or p_2 and v_2 being known.

Since section 2–2 is considered to be located just at the outer periphery of the plates, the pressure p_2 = atmospheric pressure p_a. Thus from Eq. 7.66

$$\frac{p_1}{w} = \frac{p_a}{w} + \frac{v_2^2}{2g} - \frac{v_1^2}{2g} \quad \ldots (7.67)$$

Further from Eq. 7.65 as $r_1 < r_2$, $v_1 > v_2$ and hence it may be seen from Eq. 7.67 that the pressure p_1 at section 1–1 is less than the atmospheric pressure.

Again from Eq. 7.65 and 7.66

$$\frac{p_x}{w} = H - \frac{v_x^2}{2g} = H - \frac{1}{2g}\left(\frac{v_2^2 r_2^2}{r_x^2}\right) \quad \ldots (7.68)$$

Equation 7.68 represents the pressure distribution in a radial flow, from which it may be seen that the pressure at any section in the radial flow varies with the square of the radial distance of the section from the central axis. Further the pressure increases with the increase in the radial distance following a parabolic law. The pressure distribution curve is also shown in Fig. 7.17 which is known as Barlow's curve.

Total Pressure on the Upper Plate. The total pressure (or thrust) exerted on the circular plates due to radial flow of liquid may be obtained with the help of Eq. 7.68. For this consider a small portion of the upper plate in the form of an elementary ring at a radial distance r_x and thickness d_r as shown in Fig. 7.17. The total upward pressure on the ring

$$= (2\pi r_x \times dr) \times p_x$$

$$= (2\pi r_x \times dr) \times w \left[H - \frac{v_2^2}{2g}\left(\frac{r_2^2}{r_x^2}\right)\right]$$

Thus the upward thrust on the upper plate in the portion between r_1 and r_2

$$\int_{r_1}^{r_2} 2\pi r_x \times w \left[H - \frac{v_2^2}{2g}\left(\frac{r_2^2}{r_x^2}\right)\right] dr$$

$$= 2\pi w \left[\frac{H(r_2^2 - r_1^2)}{2} - r_2^2 \frac{v_2^2}{2g} \log_e (r_2 / r_1)\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further if p_0 is the pressure of the liquid in the central vertical pipe, then assuming this pressure to be constant over the entire cross-section of the pipe, the upward thrust on the area $\left(\pi r_1^2\right)$ of the upper plate = $p_0 \times \left(\pi r_1^2\right)$.

Thus total upward pressure (or thrust) on the upper plate becomes

$$F_u = 2\pi w + \left[\frac{H(r_2^2 - r_1^2)}{2} - r_2^2 \frac{v_2^2}{2g} \log_e (r_2 / r_1)\right] + (p_0 \times \pi r_1^2) \quad \text{... (7.69)}$$

The top portion of the upper plate will be subjected to a downward force which may be either due to atmospheric pressure or it may be due to hydrostatic pressure exerted by some column of liquid lying above the plate. Thus downward force on the top portion of the upper plate may be obtained as follows.

(i) If only the atmospheric pressure p_a acts, then the downward force

$$F_d = (p_a \times \pi r_2^2)$$

(ii) If there is a liquid column of height h and specific weight w over the entire top portion of the upper plate then the downward force

$$F_d = (wh + p_a)\ \pi r_2^2$$

The resultant force F_R on the upper plate may then be obtained as

$$F_R = (F_u - F_d) \text{ or } (F_d - F_u)$$

Total Pressure on the Bottom Plate. Due to radial flow of liquid the total pressure (or thrust) exerted on the top portion of the bottom plate will be acting in the downward direction, which may also be determined by adopting the same procedure as indicated in the case of the upper plate and it is obtained as

$$F_d = 2\pi w \left[\frac{H(r_2^2 - r_1^2)}{2} - r_2^2 \frac{v_2^2}{2g} log_e (r_2 / r_1)\right] \quad \text{... (7.70)}$$

Further the lower portion of the bottom plate will also be subjected to an upward force F_u either due to atmospheric pressure or due to hydrostatic pressure exerted by some column of liquid lying above the plate as indicated in the case of the upper plate.

The resultant force F_R on the bottom plate may also be obtained as

$$F_R = (F_d - F_u) \text{ or } (F_u - F_d).$$

7.18 SPIRAL VORTEX MOTION

A spiral vortex motion may be developed when a cylindrical vortex motion is superimposed over the radial flow. In other words a spiral vortex motion is a combination of cylindrical vortex motion and radial flow and hence the pressure difference between any two points lying on a horizontal plane in a spiral vortex motion may be obtained by combining the corresponding equations for the cylindrical vortex motion and the radial flow. Accordingly the following expressions may be obtained for a spiral vortex motion.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) *Spiral Free Vortex Motion*

$$\frac{p_2}{w} - \frac{p_1}{w} = \frac{C^2}{2g}\left(\frac{1}{r_1^2} - \frac{1}{r_2^2}\right) + \left(\frac{v_1^2}{2g} - \frac{v_2^2}{2g}\right) \quad \text{... (7.71)}$$

(ii) *Spiral Forced Vortex Motion*

$$\frac{p_2}{w} - \frac{p_1}{w} = \frac{\omega^2}{2g}\left(r_2^2 - r_1^2\right) + \left(\frac{v_1^2}{2g} - \frac{v_2^2}{2g}\right) \quad \text{... (7.72)}$$

ILLUSTRATED EXAMPLES

Example 7.1. *A 0.25 m diameter pipe carries oil of specific gravity 0.8 at the rate of 120 litres per second and the pressure at a point A is 19.62 kN/m² (gage). If the point A is 3.5 m above the datum line, calculate the total energy at point A in metres of oil.*

Solution

Total energy in terms of oil is given by

$$\frac{p}{w} + \frac{V^2}{2g} + z$$

$$\frac{p}{w} = \frac{19.62 \times 10^3}{9\,810 \times 0.8} = 2.5 \text{ m of oil}$$

By continuity

$$Q = AV$$

$$Q = 120 \times 10^{-3} = 0.12 \text{ m}^3/\text{s}$$

$$A = \frac{\pi}{4} \times (0.25)^2 = 0.049 \text{ m}^2$$

Therefore $$V = \frac{Q}{A} = \frac{0.12}{0.049} = 2.45 \text{ m/s}$$

and $$\frac{V^2}{2g} = \frac{(2.45)^2}{2 \times 9.81} = 0.31 \text{ m of oil}$$

$$z = 3.5$$

$\therefore$ Total energy = (2.5 + 0.31+3.5)

= 6.31 m of oil.

Example 7.2. *A 0.3 m pipe carries water at a velocity of 24.4 m/s. At points A and B measurements of pressure and elevation were respectively 361 kN/m² and 288 kN/m² and 30.5 m and 33.5 m. For steady flow, find the loss of head between A and B.*

Solution

Total energy in terms of metres of water is given by

$$\frac{p}{w} + \frac{V^2}{2g} + z$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

At point A

$$\frac{p}{w} = \frac{361\times10^3}{9\ 810} = 36.80 \text{ m of water}$$

$$\frac{V^2}{2g} = \frac{(24.4)^2}{2\times9.81} = 30.34 \text{ m of water}$$

$$z = 30.50 \text{ m}$$

$\therefore$ Total energy at A

$$= (36.80 + 30.34 + 30.50) = 97.64 \text{ m}$$

At point B

$$\frac{p}{w} = \frac{288\times10^3}{9\ 810} = 29.36 \text{ m of water}$$

$$\frac{V^2}{2g} = \frac{(24.4)^2}{2\times9.81} = 30.34 \text{ m of water}$$

$$z = 33.5 \text{ m}$$

$\therefore$ Total energy at B

$$= (29.36 + 30.34 + 33.50) = 93.20 \text{ m}$$

$\therefore$ Loss of head $= (97.64 - 93.20) = 4.44$ m

Example 7.3. *A pipe 300 m long has a slope of 1 in 100 and tapers from 1.2 m diameter at the high end to 0.6 m diameter at the low end. Quantity of water flowing is 5 400 litres per minute. If the pressure at the high end is 68.67 k Pa [0.7 kg (f) /cm^2], find the pressure at the low end. Neglect losses.*

Solution

Discharge $Q = \frac{5\ 400}{60\times10^3} = 0.09 \text{ m}^3/\text{s}$

Area of flow section at higher end

$$= \frac{\pi}{4}(1.2)^2 = 1.131 \text{ m}^2$$

$\therefore$ Velocity at higher end

$$= \frac{0.09}{1.131} = 0.079\ 6 \text{ m/s}$$

Area of flow section at lower end

$$= \frac{\pi}{4}(0.6)^2 = 0.282\ 7 \text{ m}^2$$

$\therefore$ Velocity at lower end

$$= \frac{0.09}{0.282\ 7} = 0.318\ 4 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Applying Bernoulli's equation between the higher and the lower ends of the pipe we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2$$

Assume datum to be passing through the lower end of the pipe, then

$$z_2 = 0 \text{ ; and } z_1 = \frac{300}{100} = 3.0 \text{ m}$$

Thus by substitution

In SI Units

$$\frac{68.67\times10^3}{9\,810} + \frac{(0.079\,6)^2}{2\times9.81} + 3.0 = \frac{p_2}{w} + \frac{(0.3184)^2}{2\times9.81} + 0$$

or $$\frac{p_2}{w} = 9.995 \text{ m of water}$$

$$\therefore \quad P_2 = 98.051 \text{ kN/m}_2 = 98.051 \text{ kPa}$$

In Metric Units

$$\frac{0.7\times10^4}{1000} + \frac{(0.079\,6)^2}{2\times9.81} = 3.0 = \frac{p_2}{w} + \frac{(0.3184)^2}{2\times9.81} + 0$$

or $$\frac{p_2}{w} = 9.995 \text{ m of water}$$

$$\therefore \quad p_2 = 0.999\,5 \text{ kg(f)/cm}^2$$

Example 7.4. *A conical tube is fixed vertically with its smaller end upwards. The velocity of flow down the tube is 4.5 m/s at the upper end and 1.5 m/s at the lower end. The tube is 1.5 m long and the pressure head at the upper end is 3.1 m of the liquid. The loss in the tube expressed as a head is* $\frac{0.3(V_1 - V_2)^2}{2g}$ *where* V_1 *and* V_2 *are the velocities at the upper and lower ends respectively. What is the pressure head at the lower end?*

Solution

Applying Bernoulli's equation between the upper and the lower ends, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + \frac{0.3(V_1 - V_2)^2}{2g}$$

or $$\frac{p_2}{w} = \frac{p_1}{w} + (z_1 - z_2) + \frac{V_1^2}{2g} - \frac{V_2^2}{2g} - \frac{0.3(V_1 - V_2)^2}{2g}$$

$$\frac{p_1}{w} = 3.1 \text{ m}, (z_1 - z_2) = 1.5 \text{ m}$$

$$V_1 = 4.5 \text{ m/s}, V_2 = 1.5 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by substitution, we get

$$\frac{p_2}{w} = 3.1 + 1.5 + \frac{(4.5)^2}{2\times 9.81} - \frac{(1.5)^2}{2\times 9.81} - \frac{0.3(4.5-1.5)^2}{2\times 9.81}$$

or $$\frac{p_2}{w} = 5.38 \text{ m}$$

$\therefore$ Pressure head at the lower end,

$$\frac{p_2}{w} = 5.38 \text{ m of liquid flowing in the tube.}$$

Example 7.5. *A 15 kW pump with 80% efficiency is discharging oil of specific gravity 0.85 to the overhead tank as shown in Fig. Ex. 7.5. If losses in the whole system are 1.75 m of flowing fluid, find the discharge.*

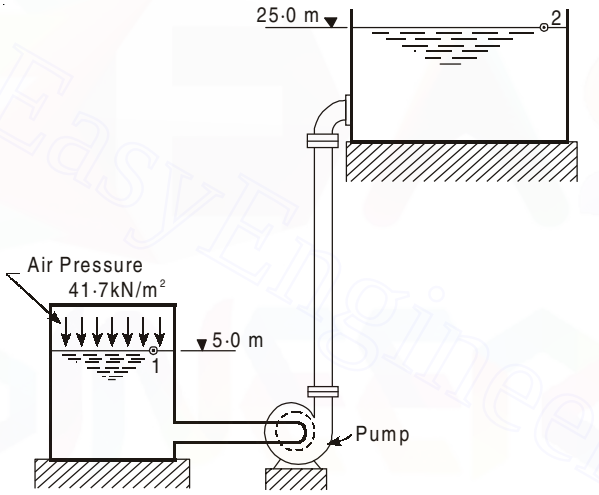


Figure Ex. 7.5

Solution

Applying Bernoulli's equation between 1 and 2, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1 + H_m = \frac{p_2}{w} + \frac{V_2^2}{2g} + Z_2 + h_L$$

$$\frac{p_2}{w} = 0, \text{ since pressure at 2 is atmospheric;}$$

V_1 and V_2 are negligible and hence the velocity heads are neglected; and

$$h_L = 1.75 \text{ m}$$

Thus by substitution, we get

$$\frac{4.17\times 10^3}{(0.85\times 9\ \ 810)} + 5.0 + 0 + H_m = 0 + 0 + 25 + 1.75$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $H_m = 16.75$ m of oil

Efficiency of pump

$$\eta = \frac{wQH_m}{P}$$

where P is power in watts.

Thus

$$0.80 = \frac{(0.85\times 9\ 810)\times Q\times 16.75}{15\times 10^3}$$

$$\therefore \quad Q = 0.086\ \text{m}^3/\text{s}$$

Discharge = 0.086 m^3/s or 86 litres/second

Example 7.6. *A portion of a pipe for conveying water is vertical and the diameter of the upper part of the pipe is 50 mm, and the section is gradually reduced to 25 mm diameter at the lower part. A pressure gage is inserted where the diameter is 50 mm and a second gage 2 m below the first and where pipe is 25 mm diameter. When the quantity of water flowing up through the pipe is 0.205 m^3 per minute, the gages show a pressure difference of 31 kN/m^2 (or 31 kPa). Assuming that the head loss varies as the square of the velocity determine the quantity of water passing through the pipe when the two gages show no pressure difference and the water is flowing downwards.*

Solution

The head loss varies as the square of velocity, as such if v is the velocity of flow at any section CC at a distance x from the lower end as shown in the figure then the head loss for a small length dx of the pipe is

$$dh_f = kv^2$$

where k is a constant of proportionality. By integrating the above expression the total loss of head for the entire pipe between the sections 1–1 and 2–2 is obtained as

$$h_f = \int_0^2 kv^2 dx \qquad \text{...(i)}$$

The diameter of the pipe at section CC is obtained as

$$d = 0.025\left(\frac{x+2}{2}\right)\text{m}$$

If Q is the discharge flowing through the pipe then

$$v = \frac{Q}{(\pi/4)d^2} = \frac{Q}{(\pi/4)(0.025)^2\left(\frac{x+2}{2}\right)^2}$$

By substituting the value of v in Eq. (i), we get

$$h_f = \int_0^2 \frac{kQ^2}{(\pi/4)^2(0.025)^4}\frac{(2)^4}{(x+2)^4}\,dx$$

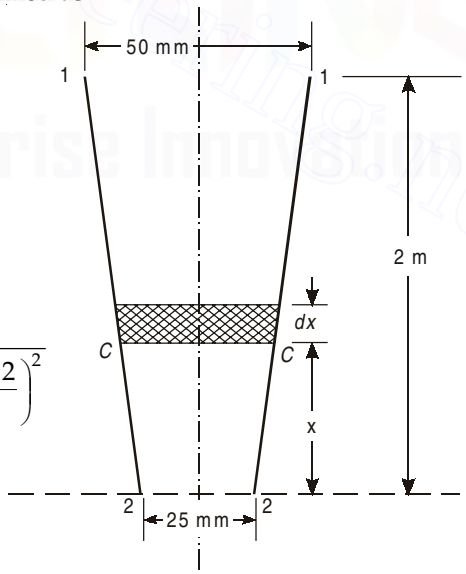


Figure Ex. 7.6

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$h_f = \frac{16\,kQ^2}{(\pi/4)^2(0.025)^4} \times \frac{1}{3}\left[\frac{1}{8} - \frac{1}{64}\right]$$

$$= \frac{7kQ^2}{(\pi/4)^2(0.025)^4(12)}$$

When water is flowing in the upward direction, then applying Bernoulli's equation between sections 1–1 and 2–2, we get

$$\frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 = \frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 + h_f$$

$$\left(\frac{p_1}{w} - \frac{p_2}{w}\right) = \frac{31\times10^3}{9\,810} = 3.16 \text{ m}$$

$$Q = \frac{0.205}{60} = 0.003\,42 \text{ m}^3/\text{s}$$

$$V_2 = \frac{Q}{A_2} = \frac{0.0003\,42}{(\pi/4)(0.05)^2} = 6.97 \text{ m/s}$$

$$V_1 = \frac{Q}{A_1} = \frac{0.0003\,42}{(\pi/4)(0.05)^2} = 1.74 \text{ m/s}$$

$$z_2 = 0 \text{ and } z_1 = 2 \text{ m}$$

Thus by substitution, we get

$$3.16 + \frac{(6.97)^2}{2\times9.81} - \frac{(1.74)^2}{2\times9.81} - 2 = \frac{7k(0.003\,42)^2}{(\pi/4)^2(0.025)^4(12)}$$

or $$k = 0.123$$

Now when the water is flowing in the downward direction then again applying the Bernoulli's equation between the sections 1–1 and 2–2, we get

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_f$$

$$\frac{p_1}{w} + \frac{p_2}{w} = 0; z_1 = 2 \text{ m}; z_2 = 0$$

If Q is the discharge flowing in this case then

$$Q = \frac{\pi}{4}(0.025)^2\ V_2 = \frac{\pi}{4}(0.05)^2\ V_1$$

$$\therefore \quad V_2 = 4\,V_1$$

Thus by substitution, we get

$$\frac{V_1^2}{2g} + 2 = \frac{16V_1^2}{2g} + 0\ \frac{7(0.123)(\pi/4)^2(0.05)^4V_1^2}{(\pi/4)^2(0.025)^4\times12}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $V_1 = 1.02 \text{ m/s}$

$$Q = A_1 V_1$$

$$= \left[\frac{\pi}{4}(0.05)^2 \times 1.02\right] = 0.002 \text{ m}^3/\text{s}$$

Discharge $= (0.002 \times 60) = 0.12 \text{ m}^3$ per minute.

Example 7.7. *The velocity distribution in a pipe is given by*

$$v = V_{max}\left(1 - \frac{r}{R}\right)^n$$

where R is the radius of the pipe, r is any radius at which the velocity is v and n is a constant index. Find the energy correction factor α *for this case. Also determine the value of* α *when* $n = \frac{1}{7}$.

Solution

The mean velocity of flow V is given by

$$Q = \pi R^2 V = \int_0^R 2\pi r \, V_{\max}\left(1 - \frac{r}{R}\right)^n dr$$

$$= \frac{2\pi R^2 V_{\max}}{(n+1)(n+2)}$$

$$\therefore \quad V = \frac{2V_{\max}}{(n+1)(n+2)}$$

and $$\frac{v}{V} = \frac{(n+1)(n+2)}{2}\left(1 - \frac{r}{R}\right)^n$$

From Eq. 7.32, we have

$$\alpha = \frac{1}{A}\int_A \left(\frac{v}{V}\right)^3 dA$$

or $$\alpha = \frac{1}{\pi R^2}\int_0^R \left[\frac{(n+1)(n+2)}{2}\right]^3 \left(1 - \frac{r}{R}\right)^{3n} 2\pi r \, dr$$

$$\therefore \quad \alpha = \frac{(n+1)^3(n+2)^3}{4(3n+1)(3n+2)}$$

Substituting $n = \frac{1}{7}$, we get after simplification

$$\alpha = 1.06$$

Example 7.8. *A venturi meter having a diameter of 75 mm at the throat and 150 mm diameter at the enlarged end is installed in a horizontal pipeline 150 mm in diameter carrying an oil of specific gravity 0.9. The difference of pressure head between the enlarged end and the throat recorded by a U-tube is 175 mm of mercury. Determine the discharge through the pipe. Assume the coefficient of discharge of the meter as 0.97.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

The discharge through the venturi meter is given by

$$Q = C_d \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$$

$$C_d = 0.97$$

$$a_1 = \frac{\pi}{4} \times \left(\frac{150}{1000}\right)^2 = 0.017\,7 \text{ m}^2$$

$$a_2 = \frac{\pi}{4} \times \left(\frac{75}{1000}\right)^2 = 0.004\,4 \text{ m}^2$$

$$h = x\left(\frac{S_m}{S} - 1\right)$$

$$= \frac{175}{1000}\left(\frac{13.6}{0.9} - 1\right) = 2.469 \text{ m}$$

Thus by substitution, we get

$$Q = \frac{0.97 \times (0.017\ \ 7 \times 0.004\ \ 4)\sqrt{2 \times 9.81 \times 2.469}}{\sqrt{(0.0177)^2 - (0.0044)^2}}$$

or $$Q = 0.030\,67 \text{ m}^3/\text{s}$$; or 30.67 litres/second

Example 7.9. *A venturi meter has its axis vertical, the inlet and throat diameters being 150 mm and 75 mm respectively. The throat is 225 mm above inlet and K = 0.96. Petrol of specific gravity 0.78 flows up through the meter at a rate of 0.029 m^3/s. Find the pressure difference between the inlet and the throat.*

Solution

The discharge through a venturi meter is given by

$$Q = \frac{K a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}} \quad K = 0.96$$

$$a_1 = \frac{\pi}{4} \times \left(\frac{150}{1000}\right)^2 = 0.017\,7 \text{ m}^2$$

$$a_2 = \frac{\pi}{4} \times \left(\frac{75}{1000}\right)^2 = 0.004\,4 \text{ m}^2$$

$$Q = 0.029 \text{ m}^3/\text{s}$$

By substitution, we have

$$0.029 = \frac{0.96 \times (0.0177 \times 0.004\ \ 4)\sqrt{2 \times 9.81 \times h}}{\sqrt{(0.0177)^2 - (0.0044)^2}}$$

$\therefore$ $$h = 2.254 \text{ m of oil}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $h = \left(\frac{p_1}{w} + z_1\right) - \left(\frac{p_2}{w} + z_2\right)$

$\therefore \quad 2.254 = -\left(\frac{p_1}{w} - \frac{p_2}{w}\right)(z_1 - z_2)$

or $2.254 = -\left(\frac{p_1}{w} - \frac{p_2}{w}\right) - \left(\frac{225}{1000}\right)$

or $\left(\frac{p_1}{w} - \frac{p_2}{w}\right) = 2.479$

$\therefore \quad (p_1 - p_2) = (2.479 \times 0.78 \times 9\,810)$

$= 18\,969 \text{ N/m}^2 = 18.969 \text{ kN/m}^2$

$= 18\,969 \text{ Pa} = 18.969 \text{ kPa}$

Example 7.10. *215 litres of gasoline (specific gravity 0.82) flow per second upwards in an inclined venturi meter fitted to a 300 mm diameter pipe. The venturi meter is inclined at 60° to the vertical and its 150 mm diameter throat is 1.2 m from the entrance along its length. Pressure gages inserted at entrance and throat show pressures of 0.141 N/mm² and 0.077 N/mm² respectively. Calculate the discharge coefficient of venturi meter.*

If instead of pressure gages the entrance and the throat of the venturi meter are connected to the two limbs of a U-tube mercury manometer, determine its reading in mm of differential mercury column.

Solution

The discharge through the venturi meter is given by

$$Q = \frac{Ka_1a_2\sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$$

$$Q = 215 \times 10^{-3} = 0.215 \text{ m}^3/\text{s}$$

$$a_1 = \frac{\pi}{4} \times \left(\frac{300}{1000}\right)^2 = 0.070\,7 \text{ m}^2$$

$$a_2 = \frac{\pi}{4} \times \left(\frac{150}{1000}\right)^2 = 0.017\,7 \text{ m}^2$$

$$h = \left(\frac{p_1}{w} + z_1\right) - \left(\frac{p_2}{w} + z_2\right)$$

$$\frac{p_1}{w} = \frac{0.141 \times 10^6}{9\,810 \times 0.82} = 17.528 \text{ m of gasoline}$$

$$\frac{p_2}{w} = \frac{0.077 \times 10^6}{9\,810 \times 0.82} = 9.572 \text{ m of gasoline}$$

$$z_1 = 0 \; ; \; z_2 = (1.2 \sin 30°) = 0.60 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad h = (17.528 + 0) - (9.572 + 0.60) = 7.356 \text{ m}$$

Thus by substitution, we get

$$0.215 = \frac{K \times (0.070\ 7 \times 0.017\ 7)\sqrt{2 \times 9.81 \times 7.356}}{\sqrt{(0.070\ 7)^2 - (0.017\ 7)^2}}$$

or $$K = 0.979$$

If a U-tube manometer is connected then

$$h = x\left(\frac{S_m}{S} - 1\right)$$

or $$7.356 = x\left(\frac{13.6}{0.82} - 1\right)$$

$$\therefore \quad x = 0.472 \text{ m or } 472 \text{ mm}$$

Example 7.11. *A venturi meter is to be fitted in a pipe 0.25 m diameter where the pressure head is 7.6 m of flowing liquid and the maximum flow is 8.1 m^3 per minute. Find the least diameter of the throat to ensure that the pressure head does not become negative. Take K = 0.96.*

Solution

The discharge through the venturi meter is given by

$$Q = \frac{Ka_1a_2\sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$$

$$Q = \frac{8.1}{60} = 0.135 \text{ m}^3/\text{s}$$

$$K = 0.96$$

$$a_1 = \frac{\pi}{4} \times (0.25)^2 = 0.049 \text{ m}^2$$

$$h = 7.6 \text{ m}$$

By substitution, we have

$$0.135 = \frac{0.96 \times (0.049) \times a_2\sqrt{2 \times 9.81 \times 7.6}}{\sqrt{(0.049)^2 - a_2^2}}$$

or $$a_2 = 0.0112 \text{ m}^2$$

$$\therefore \quad d_2 = \sqrt{\frac{0.0112 \times 4}{\pi}} = 0.119\ 5 \text{ m} = 119.5 \text{ mm}$$

Example 7.12. *The coefficient of discharge for a venturimeter used for measuring the flow of an incompressible fluid was found to be constant when the rate of flow Q exceeded a certain value. Show that under these conditions, the loss of head h_f in the convergent portion of the venturimeter can be expressed as K_1Q^2, where K_1 is a constant.*

Solution

Applying Bernoulli's equation between the inlet and throat sections of the venturimeter, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_1}{w}+\frac{V_1^2}{2g} = \frac{p_2}{w}+\frac{V_2^2}{2g}+h_f$$

If h is the venturi head then, we have

$$\frac{p_1}{w}-\frac{p_2}{w} = h$$

and hence

$$\left(h-h_f\right) = h'=\frac{V_2^2}{2g}-\frac{V_1^2}{2g}$$

Again if a_1 and a_2 are the cross-sectional areas at the inlet and throat sections of the venturimeter then, we have

$$h = \frac{Q^2}{2g}\left[\frac{1}{a_2^2}-\frac{1}{a_1^2}\right]$$

or
$$Q = \frac{a_1a_2\sqrt{2gh'}}{\sqrt{a_1^2-a_2^2}}$$

or
$$Q = C\sqrt{h'} \quad \ldots \text{(i)}$$

where $C = \dfrac{a_1a_2\sqrt{2g}}{\sqrt{a_1^2-a_2^2}}$, is the constant of the venturimeter.

Further from Eq. 7.47, we have

$$Q = KC\sqrt{h} \quad \ldots \text{(ii)}$$

in which K is the coefficient of discharge of the venturimeter.

Thus from Eqs. (i) and (ii), we obtain

$$\left(h-h'\right) = h_f = \frac{Q^2}{C^2}\left[\frac{1}{K^2}-1\right]$$

$$\therefore \quad h_f = K_1Q^2$$

where $K_1=\dfrac{1}{C^2}\left[\dfrac{1}{K^2}-1\right]$ is a constant.

Example 7.13. *Water flows at the rate of 0.147 m^3/s through a 150 mm diameter orifice inserted in a 300 mm diameter pipe. If the pressure gages fitted upstream and downstream of the orifice plate have shown readings of 176.58 kN/m^2* and *88.29 kN/m^2 respectively, find the coefficient of discharge C of the orifice meter.*

Solution

The discharge through an orifice meter is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = \frac{Ca_0a_1\sqrt{2gh}}{\sqrt{a_1^2 - a_0^2}}$$

$$Q = 0.147 \text{ m}^3/\text{s}$$

$$h = \left(\frac{p_1}{w} - \frac{p_2}{w}\right)$$

$$= \frac{(176.58 - 88.29) \times 10^3}{9810} = 9 \text{ m of water}$$

$$a_1 = \frac{\pi}{4} \times \left(\frac{300}{1000}\right)^2 = 0.070\,7 \text{ m}^2$$

$$a_0 = \frac{\pi}{4} \times 2 = 0.017\,7 \text{ m}^2$$

Thus by substitution, we get

$$0.147 = \frac{C \times (0.0707 \times 0.0177)\sqrt{2 \times 9.81 \times 9}}{\sqrt{(0.0707)^2 - (0.0177)^2}}$$

$$\therefore \quad C = 0.605$$

Example 7.14. *A submarine moves horizontally in a sea and has its axis below the water surface. A prandtl pitot tube placed in front of the submarine and along its axis is connected to the two limbs of a U-tube containing mercury. The difference in mercury level is found to be 170 mm. Find the the speed of submarine knowing that specific gravity of sea water is 1.025.*

Solution

Let the submarine along with pitot tube be moving with a velocity V.

By superimposing a velocity equal in magnitude to that of the submarine but in opposite direction, i.e., $-V$, to the whole system, the submarine including pitot tube is brought to rest and sea water attains a velocity V.

Thus from Eq. 7.25, we have

$$V = C\sqrt{2gx\left(\frac{S_m}{S} - 1\right)}$$

$$x = \frac{170}{1000} = 0.17 \text{ m}; \; S_m = 13.6; \; S = 1.025 \text{ ; and } C \text{ is taken as 1.}$$

Thus by substitution, we get

$$V = \sqrt{2 \times 9.81 \times 0.17\left(\frac{13.6}{1.025} - 1\right)}$$

or $$V = 6.4 \text{ m/s}$$

i.e., submarine is moving with a speed of 6.4 m/s

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 7.15. *Water flows in a 300 mm pipe. Two pitot tubes are installed in the pipe, one on the centreline and the other 75 mm from the centreline. If the velocities at the two points are 3 m/s and 2 m/s respectively calculate the reading on the differential mercury manometer connected to the two tubes.*

Solution

As shown in the Figure Ex. 7.15 let A and B be the stagnation points at the tips of the two pitot tubes. Considering points 1 and 2 slightly upstream of A and B and applying Bernoulli's equation, we get

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_A}{w} \qquad ...(i)$$

and

$$\frac{p_2}{w} + \frac{V_2^2}{2g} = \frac{p_B}{w} \qquad ...(ii)$$

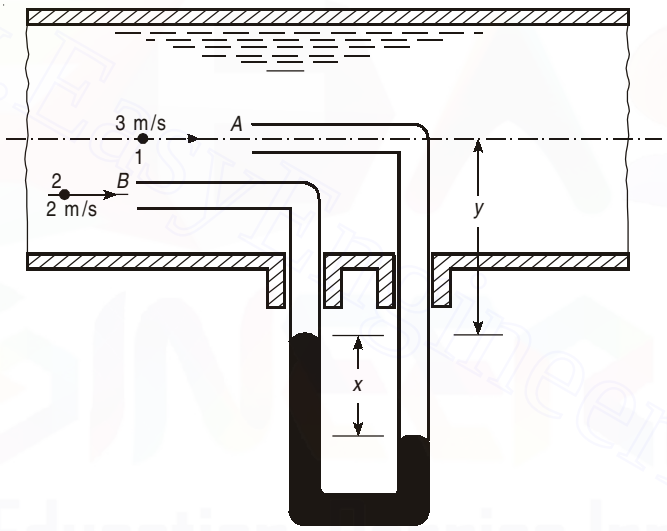


Figure Ex 7.15

where p_1 and p_2 are the static pressure intensities and V_1 and V_2 are the velocities at points 1 and 2 respectively and p_A and p_B are the stagnation pressure intensities at points A and B respectively. Also the following relation between p_1 and p_2 may be written

$$\frac{p_1}{w} + 0.075 = \frac{p_2}{w} \qquad ...(iii)$$

Thus from Eqs. (*i*), (*ii*) and (*iii*), we get

$$\frac{p_B}{w} - \frac{p_A}{w} = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + 0.075 \qquad ...(iv)$$

Further if x is the difference in the levels of mercury columns in the two limbs of the manometer and y is the depth of the higher mercury column below the centreline of the pipe, then the following manometric equation may be developed

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_A}{w} + y + x = \frac{p_B}{w} + y - 0.075 + 13.6\,x$$

or $$\frac{p_B}{w} - \frac{p_A}{w} = -12.6x + 0.075 \quad \text{...(v)}$$

Solving Eqs. (iv) and (*v*), we get

$$\frac{V_2^2}{2g} - \frac{V_1^2}{2g} + 0.075 = -12.6\,x + 0.075$$

or $$\frac{V_2^2 - V_1^2}{2g} = -12.6x$$

or $$\frac{(2)^2 - (3)^2}{2\times 9.81} = -12.6\,x$$

$$x = 0.020\,2 \text{ m or } 20.2 \text{ mm}$$

Example 7.16. *A pitot-static tube placed in the centre of a 200 mm pipe line, has one orifice pointing upstream and the other perpendicular to it. If the pressure difference between the two orifices is 40 mm of water when the discharge through the pipe is 1 365 litres per minute, calculate the coefficient of the pitot tube. Take the mean velocity in the pipe to be 0.83 of the central velocity.*

Solution

$$Q = \frac{1365\times 10^3}{60} = 0.022\,75 \text{ m}^3/\text{s}$$

The mean velocity of flow is

$$V = \frac{Q}{A} = \frac{0.02275}{\frac{\pi}{4}\left(\frac{200}{1000}\right)^2} = 0.724 \text{ m/s}$$

Central velocity = $\frac{V}{0.83} = \frac{0.724}{0.83} = 0.872$ m/s

Pressure difference $h = 40 \times 10^{-3}$ m

Thus from Eq. 7.50

$$0.872 = C\sqrt{2\times 9.81\times(40\times 10^{-3})}$$

$$\therefore \quad C = 0.984$$

Example 7.17. *Water is discharged from a 50 mm diameter nozzle which is inclined at a 30° angle above the horizontal. If the jet strikes the ground at a horizontal distance of 3.6 m and a vertical distance of 0.6 m from the nozzle as shown in Fig. Ex. 7.17 what is the rate of flow?*

Solution

Let V_0 be the velocity of the jet then

$$(V_0)_x = V_0 \cos 30° = 0.866\,V_0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(V_0)_z = V_0 \sin 30° = 0.5\, V_0$$

Therefore the coordinate equations for the trajectory are

$$x = 0.866\, V_0\, t$$

and

$$z = 0.5\, V_0\, t - \frac{1}{2} g t^2$$

By eliminating t and solving for V_0 from these two equations, we get

$$V_0 = \frac{x}{0.866}\left[\frac{g}{2(0.577x - z)}\right]^{1/2}$$

By substitution, we get

$$V_0 = \frac{3.6}{0.866}\left[\frac{9.81}{2\{(3.6\times 0.577) - (0.6)\}}\right]^{1/2}$$

or

$$V_0 = 5.63 \text{ m/s}$$

Hence

$$Q = AV_0$$

$$= \left[\frac{\pi}{4}\times\left(\frac{50}{1000}\right)^2 \times 5.63\right] = 0.011 \text{ m}^3/\text{s}$$

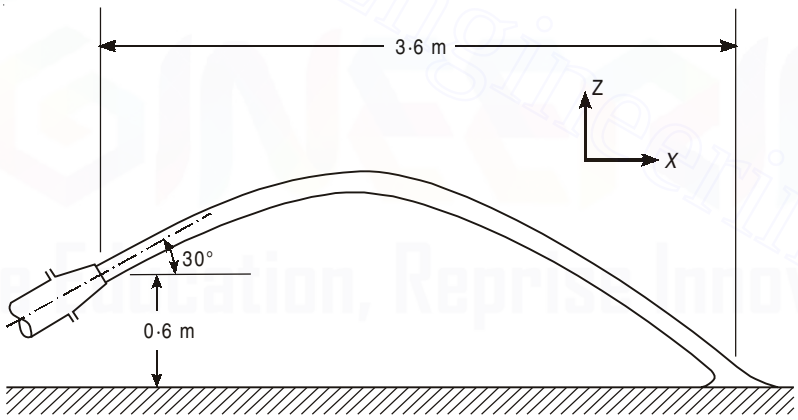


Figure Ex 7.17

Example 7.18. *A fireman intends to reach a window 25.5 m above the ground with a fire stream from a nozzle having a cylindrical tip 30 mm in diameter and discharging 1140 litres per minute. Neglecting air resistance and assuming a nozzle height of 1.5 m, determine the greatest distance from the building at which fireman can stand and still play the stream upon the window.*

Solution

$$Q = \frac{1140\times 10^{-3}}{60} = 0.019 \text{ m}^3/\text{s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_0 = \frac{Q}{A} = \frac{0.019}{\frac{\pi}{4}\left(\frac{30}{1000}\right)^2} = 26.88 \text{ m/s}$$

$$x = (V_0 \cos\theta)t \quad \text{...(i)}$$

and $$z = 24 = (V_0 \sin\theta)t - \frac{1}{2}gt^2 \quad \text{...(ii)}$$

From Eq.(i)

$$\cos\theta = \frac{x}{V_0 t}$$

$$\therefore \quad \sin\theta = \sqrt{1-\left(\frac{x}{V_0 t}\right)^2}$$

Substituting this value of sin θ in Eq. (ii), we get

$$24 = \left(\sqrt{V_0^2 t^2 - x^2}\right) - \frac{1}{2}gt^2$$

Solving for x, we get

$$x = \sqrt{V_0^2 t^2 - (24 + \frac{1}{2}gt^2)^2}$$

and $$\frac{dx}{dt} = \frac{2V_0^2 t - 2\left(24 + \frac{1}{2}gt^2\right)gt}{2\sqrt{V_0^2 t^2 - \left(24 + \frac{1}{2}gt^2\right)^2}}$$

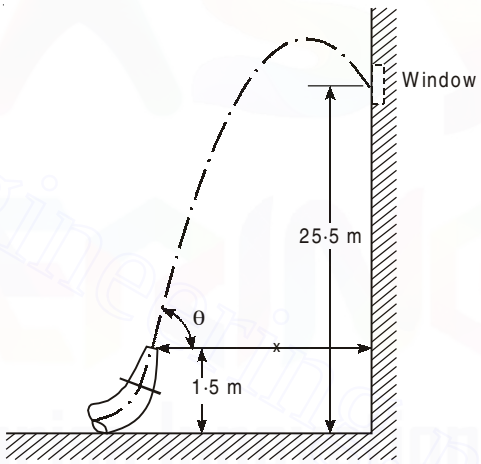


Figure Ex. 7.18

For maximum x, $\frac{dx}{dt} = 0$

Hence $$V_0^2 t = 24\,gt + \frac{1}{2}g^2 t^3$$

Solving for t, we get

$$t = \sqrt{\frac{2(26.88)^2 - 48g}{g^2}} = 3.18 \text{ s}$$

$$\therefore \quad x = \sqrt{(26.88)^2)(3.18)^2 - \left[24 + \frac{1}{2}\times 9.81 \times (3.18)^2\right]^2}$$

$$= 43.47 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 7.19. *In a free cylindrical vortex of water the tangential velocity at a radius of 0.12 m from the axis of rotation is found to be 7.2 m/s and the intensity of pressure is 245.25 kN/m² [2.5 kg(f)/ cm²]. Find the intensity of pressure at a radius of 0.24 m from the axis.*

Solution

For a free vortex

$$v_1 r_1 = v_2 r_2$$

or $$7.2 \times 0.12 = v_2 \times 0.24$$

$$\therefore \quad v_2 = 3.6 \text{ m/s}$$

Also $$\frac{p_1}{w} + \frac{v_1^2}{2g} = \frac{p_2}{w} + \frac{v_2^2}{2g}$$

Thus by substitution

In SI Units

$$\frac{245.25 \times 10^3}{9\,810} + \frac{(7.2)^2}{2 \times 9.81} = \frac{p_2}{w} + \frac{(3.6)^2}{2 \times 9.81}$$

$$p_2 = 264.69 \text{ kN/m}^2, \text{ or } 264.69 \text{ kPa}$$

In Metric Units

$$\frac{2.5 \times 10^4}{1000} + \frac{(7.2)^2}{2 \times 9.81} = \frac{p_2}{w} + \frac{(3.6)^2}{2 \times 9.81}$$

$$\therefore \quad p_2 = 2.698 \text{ kg (f)/cm}^2$$

Example 7.20. *Water flows radially between the two flanges at the end of a 0.15 m diameter pipe as shown in Fig. Ex. 7.20. Neglecting losses, if the pressure head at A is –0.3 m, find the pressure head at B* and *the flow in m³/s.*

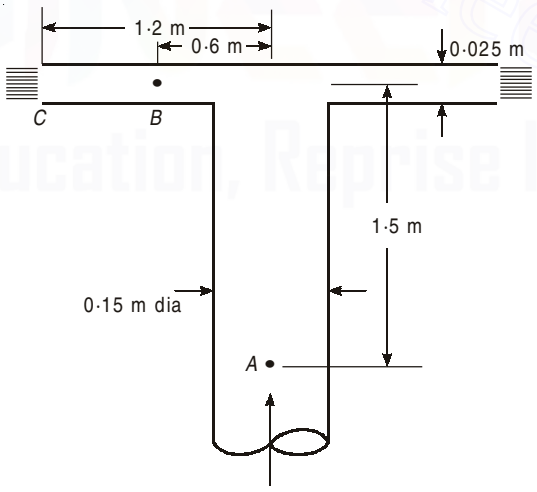


Figure Ex. 7.20

Solution

By continuity equation

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = \left(\frac{\pi}{4}\times 0.15^2\right) V_A = (2\pi \times 0.6 \times 0.025) V_B$$
$$= (2\pi \times 1.2 \times 0.025)\, V_C$$

or $$V_A = \frac{16}{3} V_B \text{ ; and } V_B = 2V_C$$

By applying Bernoulli's equation between A and C, we have

$$-0.3 + \frac{V_A^2}{2g} = \frac{V_c^2}{2g} + 1.5$$

or $$\frac{V_A^2}{2g}\left[1 - \left(\frac{3}{32}\right)^2\right] = 1.8$$

$\therefore$ $$V_A = 5.97 \text{ m/s}$$

Thus $$Q = \left(\frac{\pi}{4}\times 0.15^2 \times 5.97\right) = 0.105 \text{ m}^3/\text{s}$$

Again applying Bernoulli's equation between points B and C, we have

$$\frac{p_B}{w} + \frac{V_B^2}{2g} = \frac{p_C}{w} + \frac{V_C^2}{2g}$$

$$V_B = \left(\frac{3}{16}\times 5.97\right) = 1.12 \text{ m/s}$$

$$V_C = 0.56 \text{ m/s}$$

Thus by substitution, we get

$$\frac{p_B}{w} + \frac{(1.12)^2}{2\times 9.81} = 0 + \frac{(0.56)^2}{2\times 9.81}$$

$\therefore$ $$\frac{p_B}{w} = -0.048 \text{ m}$$

Example 7.21. *Assume a steady isothermal flow of air considered as a perfect gas with R = 29.3 metres per ℃ absolute. The velocity field is u = 3x, v= –3y, w = 20, where u, v, w are the velocity components in m/s in the coordinate directions x, y, z respectively. The coordinates are in metres. What is the ratio of pressures between the positions (10, 3, 0) and (0,– 5, 0) ? The temperature is 38℃.*

Solution

At point (10, 3, 0)

$$u = 30 \text{ m/s} ; v = -9 \text{ m/s} ; w = 20 \text{ m/s}$$

$\therefore$ $$V_1 = \sqrt{u^2 + v^2 + w^2}$$
$$= \sqrt{(30)^2 + (9)^2 + (20)^2}$$
$$= 37.16 \text{ m}$$

Similarly at point (0, –5, 0)

$$u = 0 ; v = 15 \text{ m/s} ; w = 20 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad V_2 = \sqrt{(0)^2 + (15)^2 + (20)^2}$$
$$= 25 \text{ m/s}$$

From Eq. 7.35 for isothermal flow, we have

$$K \log_e p_1 + \frac{V_1^2}{2} + gz_1 = K \log_e p_2 + \frac{V_2^2}{2} + gz_2$$

where K is a constant given by

$$\frac{p}{\rho} = K = RT$$

Thus $K = 29.3\ (273 + 38) = 9112.3$

Considering $z_1 = z_2$, and substituting in the above equation, we get

$$9\ 112.3 \log_e (p_2 / p_1) = \frac{(37.16)^2}{2} - \frac{(25)^2}{2}$$

or $$\log_e (p_2/p_1) = 0.041\ 5$$

$$\therefore \quad \frac{p_2}{p_1} = 1.042$$

Example 7.22. *The pressure leads from a Pitot-static tube mounted on an aircraft are connected to a pressure gage in the cockpit. The dial of the pressure gage is calibrated to read the aircraft speed in m/s.*

The calibration is done on the ground by applying a known pressure across the gage and calculating the equivalent velocity using the incompressible Bernoulli equation and assuming that the density is 1.224 kg/m^3.

The gage having been calibrated in this way the aircraft is flown at 9200 m, where the density is 0.454 kg/m^3 and ambient pressure is 30 000 N/m^2. The gage indicates a velocity of 152 m/s. What is the ture speed of the aircraft?

Solution

When the aircraft is flown at 9200 m the stagnation pressure developed is

$$p = \left[30000 + \frac{1}{2} \times .454 \times (152)^2\right] \text{N/m}^2$$
$$= 35\ 245 \text{ N/m}^2$$

Let p_a be the atmospheric pressure at the ground level, p be the pressure applied across the gage during calibration, and V be the equivalent velocity, then using incompressible Bernoulli equation, we obtain

$$p_a + \frac{1}{2}\rho V^2 = p_a + p$$

or $$\frac{1}{2}\rho V^2 = p$$

Thus true speed of the aircraft corresponding to the stagnant pressure developed when it is flown at 9200 m is given by

$$\frac{1}{2} \times 1.224 \times V^2 = 35245$$

$$\therefore \quad V = 240 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. The study of fluid motion involving the consideration of the forces and energies causing the flow of fluid is known as dynamics of fluid flow.
2. The various forces on fluid mass may be classified as (i) body or volume forces, (ii) surface forces, and (iii) line forces.
3. The dynamics of fluid flow is governed by Newton's second law of motion which states that the resultant force on any fluid element must equal the product of the mass and the acceleration of the element, i.e.,
$$\Sigma F = Ma$$
where ΣF represents the resultant external force acting on the fluid element of mass M and a is the total acceleration.
4. The various forces that may influence the motion of a fluid are : gravity force F_g, pressure force F_p viscous force F_v, surface tension force F_s, and compressibility force F_e.
(i) If a certain mass of fluid in motion is influenced by all the above mentioned forces, then according to Newton's second law of motion the equation of motion may be written as
$$Ma = F_g + F_p + F_v + F_t + F_s + F_e$$
(ii) In most cases of the fluids in motion the surface tension and the compressibility forces are not significant, and hence these forces may be neglected, in which case the equation of motion becomes
$$Ma = F_g + F_p + F_v + F_t$$
This is known as Reynolds' equation of motion.
(iii) For viscous flows the turbulent forces are not significant and hence these forces may be neglected, in which case the equation of motion becomes
$$Ma = F_g + F_p + F_v$$
This is known as Navier–Stokes, equation.
(iv) If in the case of fluid flow the viscous forces are not significant, these forces may be neglected, in which case the equation of motion becomes
$$Ma = F_g + F_p$$
This is known as Euler's equation of motion.
5. By integrating Euler's equation of motion, energy equation called, Bernoulli's equation is obtained as
$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = C$$
which is applicable for steady irrotational flow of compressible fluids. For incompressible fluids since mass density ρ is independent of pressure Bernoulli's equation becomes
$$\frac{p}{\rho} + \frac{V^2}{2} + gz = C$$
or
$$\frac{p}{\rho g} + \frac{V^2}{2g} + z = C'$$
or
$$\frac{p}{w} + \frac{V^2}{2g} + z = C'$$
where $\frac{p}{\rho}$ = pressure energy per unit mass of fluid;
$\frac{p}{\rho g}$ or $\frac{p}{w}$ = pressure energy per unit weight of fluid; it is known as 'pressure head' or 'static head'.
$\frac{V^2}{2}$ = kinetic energy per unit mass of fluid;
$\frac{V^2}{2g}$ = kinetic energy per unit weight of fluid; it is known as 'velocity head' or 'kinetic head';
gz = potential energy per unit mass of fluid.
z = potential energy per unit weight of fluid ; it is known as 'potential head' or 'datum head' ; and
C, C' = constants
The sum of the pressure head the velocity head and the potential head is known as the total head or the total energy per unit weight of the fluid.
The Bernoulli's equation thus states that in a steady, irrotational flow of an incompressible fluid the total energy at any point is constant. Accordingly if Bernoulli's equation is apploed between any two points in a steady irrotational flow of an incompressible fluid then we get
$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2$$
The above equation has been derived for an ideal fluid for which there is no loss of energy.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, for the flow of real fluids there is some loss of energy, and hence Bernoulli's equation for real fluids becomes

$$\frac{p_1}{w}+\frac{V_1^2}{2g}+z_1 = \frac{p_2}{w}+\frac{V_2^2}{2g}+z_2+h_L$$

where h_L is the loss of energy (or head) between the two points.

The sum of the pressure head and the potential head i.e., $\left(\frac{p}{w}+z\right)$ is also termed as piezometric head.

6. The kinetic energy correction factor α is given by

$$\alpha = \frac{1}{AV^3}\int V^3 dA$$

7. Bernoulli's equation for the flow of a compressible fluid with isothermal change is obtained as

$$K\log_e p_1+\frac{V_1^2}{2}+gz_1 = K\log_e p_2+\frac{V_2^2}{2}+gz_2+gh_L$$

or $$K'\log_e p_1+\frac{V_1^2}{2g}+z_1 = K'\log_e p_2+\frac{V_2^2}{2g}+z_2+h_L$$

8. Bernoulli's equation for the flow of a compressible fluid with adiabatic change is obtained

$$\frac{k}{k-1}\frac{p_1}{\rho_1}+\frac{V_1^2}{2}+gz_1 = \frac{k}{k-1}\frac{p_2}{\rho_2}+\frac{V_2^2}{2}+gz_2+gh_L$$

or $$\frac{k}{k-1}\frac{p_1}{w_1}+\frac{V_1^2}{2}+z_1 = \frac{k}{k-1}\frac{p_2}{w_2}+\frac{V_2^2}{2g}+z_2+h_L$$

9. Discharge Q through a venturi meter is given by

$$Q = K\frac{a_1a_2\sqrt{2gh}}{\sqrt{a_1^2-a_2^2}}$$

where

a_1 = aera of the inlet of the venturi meter ;
a_2 = aera at the throat of the venturi meter ;
K = coefficient of discharge of the venturi meter ;
h = difference between the pressure heads at the inlet and the throat of the venturimeter, and is known as venturi head ; and
g = acceleration due to gravity.

When the venturimeter is provided in an inclined pipe the venturi head h is equal to the difference between the piezometric heads at the inlet and the throat of the venturimeter.

10. Discharge Q through an orifice meter is given by

$$Q = \frac{C_d a_0 a_1\sqrt{2gh}}{\sqrt{a_1^2-\alpha_0^2}}$$

where

a_0 = area of the orifice;
a_1 = area of the pipe;
C_d = coefficient of discharge of the orifice meter;
h = difference between pressurs heads (or piezometric heads) at sections upstream and downstream of the orifice plate; and
g = acceleration due to gravity.

11. The other devices used for measuring discharge through pipes are nozzle meter, rotameter and elbow meter.

12. Pitot tube is used for measuring the velocity of flow of a fluid at any point in a pipe or a channel. The velocity of flow V is given by

$$V = C\sqrt{2gh}$$

where

C = coefficient of the pitot tube;
g = acceleration due to gravity; and
h = dynamic pressure head.

In the case of channels the dynamic pressure head h is equal to the rise of liquid in the pitot tube above the free surface of liquid.

In the case of pipes the dynamic pressure head h is determined by a differential manometer connected between the pitot tube and the pressure tap meant for measuring the static pressure.

13. A jet of liquid issuing from a nozzle in at mosphere is known as 'free liquid jet'. The equation of the profile or trajectory of the free liquid jet is given as

$$(z-z_0) = x\tan\theta+\frac{gx^2}{2V_0^2}\sec^2\theta$$

where

z = height of any point on the jet above the datum;

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

z_0 = height of the nozzle tip above the datum;
x = distance of any point on the jet from the nozzle tip in the horizontal direction;
θ = angle of inclination of the jet with the horizontal ;
V_0 = velocity of the jet issuing from the nozzle; and
g = acceleration due to gravity.

The maximum height of the free liquid jet above the nozzle tip is given as

$$(z - Z_0)_{max} = \frac{(V_0)_z^2}{2g} = \frac{V_0^2 \sin^2\theta}{2g}$$

where
$(V_0)_z$ = component of velocity V_0 along the z or vertical direction which is equal to $V_0 \sin\theta$.

The horizontal distance from the nozzle tip of the point at which the free liquid jet attains the maximum height is given as

$$L = \frac{V_0^2 \sin 2\theta}{2g}$$

The horizontal distance from the nozzle tip through which the free liquid jet travels is given as

$$2L = \frac{V_0^2 \sin 2\theta}{g}$$

14. A rotating mass of fluid is known as 'vortex' and the motion of a rotating mass of fluid is known as 'vortex motion'. The vortex motion is of two types viz., free vortex motion and forced vortex motion.

A 'free vortex motion' is that in which the fluid mass rotates without any external force being exerted on the fluid mass. The velocity of flow v in a free vortex motion varies inversely with the radial distance r from the centre of vortex motion, i.e.,

$$v\,r = \text{constant} = C$$

A 'forced vortex motion' is that in which the fluid mass is made to rotate by means of some external source of power, which exerts a constant torque on the fluid mass, thereby causing the whole mass of fluid to rotate at constant angular velocity ω. In this case the velocity v at a distance r from the axis of rotation may be expressed as

$$v = r\omega$$

15. In vortex motion the variation of pressure is represented by the following defferential equations :

$$\frac{\partial p}{\partial r} = -\rho\frac{V^2}{r}; \text{ and } \frac{\partial p}{\partial z} = -\rho g$$

After substituting for v from the respective equations and integrating the differential equations between any two points 1 and 2, the following equations are obtained:

(i) For free vortex motion

$$\frac{p_1}{\rho_1} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2$$

which is same as Bernoulli's equation for the flow of an ideal fluid; and

(ii) For forced vortex motion

$$\frac{p_1}{\rho_1} - \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} - \frac{V_2^2}{2} + gz_2$$

PROBLEMS

7.1 What are the different energies of a fluid? Explain each of them.

7.2 State and derive Bernoulli's theorem, mentioning clearly the assumptions underlying it.

7.3 Derive Bernoulli's equation from general energy equation.

7.4 Derive Bernoulli's equation from Euler's equation of motion.

7.5 What is kinetic energy correction factor?

7.6 Starting from first principles obtain the following expression for discharge of a liquid through a venturi meter.

$$Q = K C h^{1/2}$$

where Q is discharge in m^3/s; K is discharge coefficient of the venturi meter; C is constant of venturi meter; and h is venturi head in metres of fluid flowing through the venturi meter.

7.7 A vertical venturi meter of (d/D) ratio equal to 0.6 is fitted in a 0.1 m diameter pipe. The throat is 0.2 m above the inlet. The meter has a coefficient of discharge of 0.92. Determine (i) pressure difference as recorded by two gages fitted at the inlet and throat, (*ii*) difference on a vertical differential mercury manometer (specific gravity of mercury = 13.6) when a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

liquid of specific gravity 0.8 flows through the meter at the rate of 50 litres per second.

[*Ans.* (i) 130.206 kN/m^2 or 1.327 kg(f)/cm^2; (ii)1.024 m]

7.8 Oil of specific gravity 0.90 flows in a pipe 300 mm diameter at the rate of 120 litres per second and the pressure at a point A is 24.525 kPa [0.25 kg(f)/cm^2] (gage). If the point A is 5.2 m above the datum line, calculate the total energy at point A in terms of metres of oil.

[*Ans.* 8.125 m]

7.9 In a vertical pipe conveying water, pressure gages are inserted at A and B, where the diameters are 150 mm and 75 mm respectively. The point B is 2.4 m below A, and when the rate of flow down the pipe is 0.021 m^3/s the pressure at B is 117.72 kN/m^2 [0.12 kg (f)/cm^2] greater than that at A. Assuming that the losses in the pipe between A and B can be expressed as ($kV_A^2/2g$), where V_A is the velocity at A, find the value of k. If the gages at A and B are replaced by tubes filled with water and connected to a U-tube containing mercury of specific gravity 13.6, calculate the value of the reading on the manometer.

[*Ans.* 1.67; 95.2 mm]

7.10 A 0.3 m by 0.15 m venturi meter is mounted on a vertical pipe with flow upwards. 63 litres per second of oil of specific gravity 0.80 and dynamic viscosity 1 poise flows through the pipeline. The throat section is 0.1 m above the inlet section. What is the pressure difference between the inlet and the throat?

[*Ans.* Reynolds Number = $\frac{\rho V d}{\mu}$ =

$\frac{0.80 \times 1000 \times 3.565 \times 0.15}{0.1}$ = 4 278 for which K = 0.96. The required pressure difference = 5.964 kN/m^2 or 5.964 kPa or 0.060 8 kg(f)/cm^2]

7.11 A venturimeter is used for measuring the flow of petrol in a pipeline inclined at 35° to horizontal. The specific gravity of the petrol is 0.81 and throat area ratio is 4. If the difference in mercury levels in the gage is 50 mm, calculate the flow in litres per hour if the pipe diameter is 0.3 m. Take discharge coefficient of the venturi meter as 0.975.

[*Ans.* 2.52×10^5 litres/hour]

7.12 A venturi meter measures the flow of water in a 75 mm diameter pipe. The difference of head between the entrance and the throat of the venturi meter is measured by U-tube containing mercury, the space above the mercury on each side being filled with water. What should be the diameter of the throat of the meter in order that the difference of the levels of the mercury shall be 0.25 m when the quantity of water flowing in the pipe is 630 litres per minute? Assume the discharge coefficient as 0.97.

[*Ans.* 40.9 mm]

7.13 Define kinetic energy correction factor and derive an expression for the same. Velocity distribution forlaminar flow of a real fluid in a pipe is given by the equation $V = V_{max}[1-(r^2/R^2)]$, where V_{max} is velocity at the centre of the pipe, R is pipe radius, and V is the velocity at radius r from the centre of the pipe. Show that kinetic energy correction factor for this flow is 2.

7.14 A venturi meter having inlet diameter 100 mm and throat diameter 25 mm is fitted in a vertical pipe, throat 0.3 m below inlet, for measuring the flow of petrol of specific gravity 0.78. Pressure gages are fitted at inlet and throat. Assuming the loss of head between inlet and throat as 30 times the velocity head at inlet, find: (a) C_d for the meter; (b) the discharge in litres per minute when the inlet gage reads 274.68 kN/m^2 [0.28 kg(f)/cm^2] more than the throat gage.

[*Ans.* (a) 0.946; (b) 244 litres/min]

7.15 A 150 mm × 75 mm venturi meter with a coefficient of discharge 0.98 is to be replaced by an orifice meter having a coefficient of discharge 0.6. If both the meters are to give the same differential mercury manometer reading for a discharge of 100 litres per second and the inlet diameter is to remain 150 mm, what should be the diameter of the orifice.

[*Ans.* 93.5 mm]

7.16 A pitot tube was used to measure the quantity of water flowing in a pipe of 0.3 m diameter. The water was raised to a height of 0.25 m above the centre line of pipe in a vertical limb of the tube. If the mean velocity is 0.78 times the velocity at the centre and coefficient of pitot tube is 0.98, find the quantity of water in litres per second. Static pressure head at the centre of the pipe is 0.2 m. [*Ans.* 53.52 l/s]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

7.17 A pipe carrying oil of specific gravity 0.877 changes in size from 0.15 m at section *A* to 0.45 m at section *B* . Section *A* is 3.6 m lower than *B* and the pressures are 90.252 kN/m^2 [0.92 kg(f)/cm^2] and 59.841 kN/m^2 [0.61 kg(f)/cm^2] respectively. If the discharge is 0.145 m^3/s, determine the head loss and the direction of flow.

[*Ans.* 3.32 m of oil; Flow is from *A* to *B*]

7.18 Water is supplied to a turbine through a 0.3 m diameter pipe at the rate of 0.53 m^3/s. After passing through the turbine water is discharged through a divergent pipe (called draft tube) which has a diameter of 0.6 m at point *B* which is 1m below a point *A* in the 0.3 m pipe. If the pressures at the points *A* and *B* are respectively 147.15 kN/m^2 [1.5 kg(f)/cm^2] and –34.335 kN/m^2 [– 0.35 kg(f)/cm^2] , determine the power delivered to the turbine by the water.

[*Ans.* 115.35 kW (or 156.8 h.p)]

7.19 Find the velocity of flow of carbon tetrachloride specific gravity 1.59 through a pipe when a differential gage attached to a pitot static tube shows a reading of 0.1 m. Take the coefficient of the pitot tube as 0.98. [*Ans.* 3.77 m/s]

7.20 Two horizontal parallel circular discs are placed 50 mm apart and each plate has diameter of 0.25 m. The lower plate has central hole of 50 mm diameter and a 50 mm diameter vertical pipe is connected centrally to it. Water flows through the pipe at the rate of 500 litres per minute. If the pressure at the outer edge of the disc is 10 m of water, determine (i) pressure inside the vertical pipe ; (ii) velocity and pressure at a radius of 80 mm from the centre ; (iii) the resultant pressure on the upper disc ; and (iv) the resultant pressure on the bottom disc.

[*Ans.* (i) 89.117 kN/m^2; (ii) 0.332 m/s, 98.068kN/m^2; (iii) 22.209 N in the downward direction; (iv) 2.502 N in the upward direction]

7.21 In a free cylindrical vortex of water, the tangential velocity at a radius of 0.1 m from the axis of rotation is found to be 10 m/s and the intensity of pressure is 196.2 kN/m^2 [2 kg (f) / cm^2]. Find the intensity of pressure at a radius of 0.2 m from the axis.

[*Ans.* 233.7 kN/m^2, or 233.7 kPa, or 2.382 kg(f)/cm^2]

7.22 The wind velocity in a cyclone may be assumed to be free vortex flow. If at 50 km from centre of cyclone, velocity is 16 km/h, what pressure gradient should be obtained at this point. What reduction in barometric pressure should occur over radial distance of 10 km from this point towards centre of storm. Mass density of air = 1.2 kg/m^3. [*Ans.* 1 in 2110; 0.566 m]

7.23 A Pitot-static tube having a coefficient of 0.98 is used to measure the velocity of water in a pipe. The stagnation pressure recorded is 3 m and the static pressure 2 m. What velocity does this indicate. [*Ans.* 4.34 m/s]

Apprise Education, Reprise Innovations

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Impulse Momentum Equation and its Applications

Chapter 8

8.1 INTRODUCTION

The impulse momentum principle is another very useful principle, in addition to the continuity and the energy principles, the application of which leads to the solution of several fluid flow problems. On the basis of the impulse momentum principle impulse momentum equations are developed. These equations are often used in conjunction with the energy and the continuity equations in order to obtain the solutions of the problems of fluid flow which cannot be solved simply by applying the continuity and the energy equations. In the following paragraphs, the impulse momentum equations applicable to the problems of fluid flow have been derived and some of their applications are discussed.

8.2 IMPULSE-MOMENTUM EQUATIONS

The impulse-momentum equations are derived from the impulse-momentum principle (or simply momentum principle) which states that the impulse exerted on any body is equal to the resulting change in momentum of the body. In other words, this principle is a modified form of Newton's second law of motion. Newton's second law of motion states that the resultant external force acting on any body in any direction is equal to the rate of change of momentum of the body in that direction. Thus for any arbitrarily chosen direction x, it may be expressed as,

$$F_x = \frac{d(M_x)}{dt} \quad \ldots (8.1)$$

In which F_x represents the resultant external force in the x-direction and M_x represents the momentum in the x-direction. Equation 8.1 may also be written as

$$F_x(dt) = d(M_x) \quad \ldots (8.2)$$

In Eq. 8.2 the term $F_x(dt)$ is impulse and the term $d(M_x)$ is the resulting change of momentum. Equating 8.2 is thus known as *impulse momentum equation.* The impulse-momentum relationship in the form as indicated by Eq. 8.2 is, however, applicable to finite or discrete bodies, for which the action of any force may take place and be completed in a finite period of time. On the other hand, steady flow of fluid involves a motion which is continuous and it is not completed in a finite period of time. Therefore the momentum equation has to be expressed in a form particularly suited to the solution of fluid flow problems as explained below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

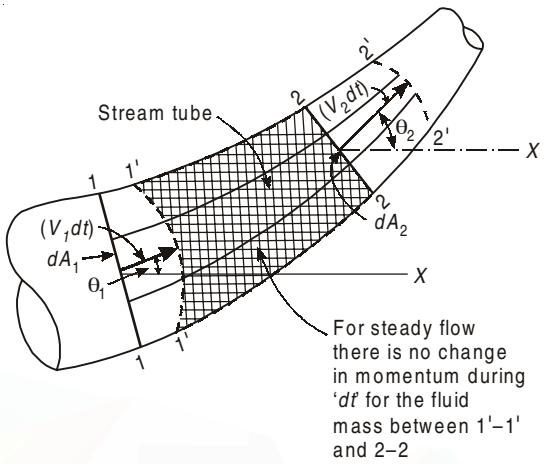


Figure 8.1 *Change of momentum of fluid mass in flow passage*

Consider as a free-body the fluid mass included between sections 1–1 and 2–2 within a certain flow passage as shown in Fig. 8.1. The fluid mass of the free body 1–1 and 2–2 at time t moves to a new position 1′–1′ and 2′–2′ at time $(t + dt)$. Section 1′–1′ and 2′–2′ are curved because the velocities of flow at these two sections are non-uniform.

For steady flow the following continuity equation holds:

$$\begin{bmatrix}\text{Fluid mass with in}\\ \text{sections 1–1 and 1'–1'}\end{bmatrix} = \begin{bmatrix}\text{Fluid mass with in}\\ \text{sections 2 – 2 and 2'–2'}\end{bmatrix}$$

Further for an arbitrary direction x the change in momentum of this mass of fluid during a time interval dt is considered, which may be represented as follows:

$$\begin{bmatrix}\text{Change in momentum}\\ \text{of fluid mass}\\ \text{during } dt\end{bmatrix}_x = \begin{bmatrix}\text{Momentum at}\\ (t+dt)\text{ of fluid}\\ 1'\text{–}1'\text{ and }2'\text{–}2'\end{bmatrix}_x - \begin{bmatrix}\text{Momentum at}\\ t\text{ of fluid}\\ 1-1\text{ and }2\text{–}2\end{bmatrix}_x$$

But

$$\begin{bmatrix}\text{Momentum at}\\ (t+dt)\text{ of fluid}\\ 1'\text{–}1'\text{ and }2'\text{–}2'\end{bmatrix}_x = \begin{bmatrix}\text{Momentum at}\\ (t+dt)\text{ of fluid}\\ 1'\text{–}1'\text{ and }2\text{–}2\end{bmatrix}_x + \begin{bmatrix}\text{Momentum at}\\ (t+dt)\text{ of fluid}\\ 2-2\text{ and }2'-2'\end{bmatrix}_x$$

and

$$\begin{bmatrix}\text{Momentum at}\\ t\text{ of fluid}\\ 1\text{–}1\text{ and }2-2\end{bmatrix}_x = \begin{bmatrix}\text{Momentum at}\\ t\text{ of fluid}\\ 1\text{–}1\text{ and }1'-1'\end{bmatrix}_x + \begin{bmatrix}\text{Momentum at}\\ t\text{ of fluid}\\ 1'-1'\text{ and }2\text{–}2\end{bmatrix}_x$$

Moreover, when the flow is steady, the state of the flowing fluid in the flow passage within sections 1′–1′ and 2–2 remains unchanged at all times.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Therefore

$$\begin{bmatrix}\text{Momentum at}\\ (t+dt)\text{ of fluid}\\ 1'-1'\text{ and }2-2\end{bmatrix}_x = \begin{bmatrix}\text{Momentum at}\\ t\text{ of fluid}\\ 1'-1'\text{ and }2-2\end{bmatrix}_x$$

From which it follows that

$$\begin{bmatrix}\text{Change in momentum}\\ \text{of fluid mass}\\ \text{during } dt\end{bmatrix}_x = \begin{bmatrix}\text{Momentum at}\\ (t+dt)\text{ of fluid}\\ 2-2\text{ and }2'-2'\end{bmatrix}_x - \begin{bmatrix}\text{Momentum at}\\ t\text{ of fluid}\\ 1-1\text{ and }1'-1'\end{bmatrix}_x$$

The above relationship when expressed in terms of mathematical symbols, it becomes

$$\sum d(mv_x) = \int_{A_2} \rho_2 v_2 dt\, dA_2 (v_2)_x - \int_{A_1} \rho_1 v_1\, dt\, dA_1 (v_1)_x$$

where $(\rho_2 v_2\, dt\, dA_2)$ and $(\rho_1 v_1\, dt\, dA_1)$ represent the mass of flow of fluid during the time interval dt in a stream tube across sections 2–2 and 1–1 respectively as shown in Fig. 8.1.

Further according to Newton's second law of motion (Eq. 8.1), the relationship between the resultant external force and the time rate of change of momentum of the fluid flow in the passage may be written in the following form:

$$\Sigma F_x = \frac{\sum d(mv_x)}{dt}$$

$$= \int_{A_2} \rho_2 v_2\, dA_2 (v_2 \cos\theta_2) - \int_{A_1} \rho_1 v_1\, dA_1 (v_1 \cos\theta_1) \quad \text{... (8.3)}$$

in which $v_2 \cos\theta_2 = (v_2)_x$ and $v_1 \cos\theta_1 = (v_1)_x$ (see Fig. 8.1). Equation 8.3 may be integrated if the velocity distributions of fluid flow at both sections are known. Since in most of the problems of fluid flow we have to deal with only the mean velocity of flow at each section, it is preferable to express the impulse-momentum equation in terms of the mean velocities. Thus if V_1 and V_2 are the mean velocities at sections 1–1 and 2–2 respectively, then the impulse-momentum Eq. 8.3 may be written as

$$\Sigma F_x = \rho_2 A_2 \cos\theta_2 V_2^2 - \rho_1 A_1 \cos\theta_1 V_1^2 \quad \text{... (8.4)}$$

For a steady flow of incompressible fluid, the impulse-momentum equation for fluid flow may be simplified to the form noted below. The continuity equation for such a flow may be expressed as $Q = A_1 V_1 = A_2 V_2$ and $\rho_1 = \rho_2 = \rho$. Thus introducing these expressions in Eq. 8.4 it becomes

or
$$\left.\begin{aligned}\sum F_x &= \rho Q(V_2 \cos\theta_2 - V_1 \cos\theta_1\\ \sum F_x &= \rho Q[(V_2)_x - (V_1)_x]\end{aligned}\right\} \quad \ldots (8.5)$$

in which suffix x is introduced to represent the components of the velocities in the x-direction. The term ΣF_x should include all the external forces acting on the free-body of the fluid under consideration.

If D'Alembert's principle is applied to the flow system, the system is brought into relative static equilibrium with the inclusion of inertia forces. The resulting impulse-momentum equation takes the following form :

$$\Sigma Fx - \rho Q\,(V_2)_x + \rho Q(V_1)_x = 0 \quad \text{... (8.6)}$$

The inertia force (ρQV) in fluid flow is usually called the momentum flux.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

It may however be noted that the impulse-momentum equation given above has been derived for one direction only, but the same method may be extended to derive the corresponding equations for the other directions of reference as well. Accordingly the impulse-momentum equations for y and z directions may be written as

$$\Sigma F_y = \rho Q [(V_2)_y - (V_1)_y] \quad \text{... (8.7)}$$

$$\Sigma F_z = \rho Q [(V_2)_z - (V_1)_z] \quad \text{... (8.8)}$$

Further the general impulse-momentum equation for steady flow of fluid may be written in a vector form as

$$\Sigma F \rightarrow \rho Q V_2 + \rightarrow \rho Q V_1 = 0 \quad \text{... (8.9)}$$

The impulse-momentum equations are often called simply momentum equations. From these equations it may be noted that if the resultant external force that acts on the fluid mass is zero, the momentum of the fluid mass remains constant. This principle is known as the *law of conservation of momentum.*

8.3 MOMENTUM CORRECTION FACTOR

The above derived impulse-momentum equations in terms of the mean velocities of flow are based on the assumption that the velocity of flow at each section is uniform, that is the velocity is same at different points on the same section. However, in actual practice the velocity is not uniform over a cross-section of the flow passage, on account of which the momentum flux computed on the basis of the mean velocity of flow at any section is not equal to the actual momentum flux flowing through the section. The actual momentum flux flowing through any section may be obtained by integrating the momentum flux flowing through different elementary areas of the cross-section. Thus if v is the velocity of flowing fluid at any point through an elementary area dA of the cross-section, then the mass of fluid flowing per unit time is ($\rho v dA$) and the corresponding momentum flux flowing through this elementary area is ($\rho v^2 dA$). The total momentum flux flowing through the entire cross-section A is equal to

$$\int_A \rho v^2 dA = \frac{w}{g} \int_A v^2 dA$$

which may be evaluated from the known velocity distribution at the cross-section. It is however more convenient to express the momentum flux flowing through any cross-section in terms of the mean velocity of flow. But the actual momentum flux is always greater than that computed by using the mean velocity of flow. Hence in order to account for this difference in the values of the momentum flux due to the non-uniform velocity distribution at any cross-section a factor called *momentum correction factor* represented by β (Greek 'beta') is introduced, so that the momentum flux computed by using the mean velocity V may be expressed as ($\beta \rho A V^2$) and it is then equal to the actual total momentum flux flowing through the entire cross-section. Thus equating the two, the value of the momentum correction factor may be obtained as

$$\beta \rho A V^2 = \rho \int_A v^2 dA$$

Therefore

$$\beta = \frac{1}{AV^2} \int_A v^2 dA \quad \text{... (8.10)}$$

Mathematically the square of the average is less than the average of the squares, that is $V^2 < \frac{1}{A} \int_A v^2 dA$, the numerical value of β will always be greater than 1. The actual value of β depends on the velocity

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

distribution at the flow section. If the velocity is uniform over the entire cross-section, β will be equal to 1. For turbulent flow in pipes the value of β lies between 1.02 and 1.05. However, for laminar flow in pipes the value of β is 1.33.

In view of the above discussion, if the velocity distribution is non-uniform, then the momentum correction factor will be required to be introduced in the impulse-momentum equations expressed in terms of mean velocity at each section. Thus Eqs. 8.5, 8.7 and 8.8 are modified as follows:

$$\left.\begin{aligned} \Sigma F_x &= \rho Q\,[\beta_2 (V_2)_x - \beta_1 (V_1)_x] \\ \Sigma F_y &= \rho Q\,[\beta_2 (V_2)_y - \beta_1 (V_1)_y] \\ \Sigma F_z &= \rho Q\,[\beta_2 (V_2)_z - \beta_1 (V_1)_z] \end{aligned}\right\} \quad \text{... (8.11)}$$

in which β_1 and β_2 are the momentum correction factors at sections 1–1 and 2–2 respectively. However, in most of the problems of turbulent flow, the value of β is nearly equal to 1 and therefore it may be assumed as one, without any appreciable error being introduced. Accordingly the impulse-momentum equations for the three reference directions may be represented by Eq. 8.5, 8.7 and 8.8.

8.4 APPLICATIONS OF THE IMPULSE-MOMENTUM EQUATION

The impulse-momentum equation, together with the energy equation and the continuity equation provides the basic mathematical relationships for solving various engineering problems in fluid mechanics. Since the impulse-momentum equation relates the resultant external forces on a chosen free-body of fluid or control volume in a flow passage, to the change of momentum flux at the two end sections, it is especially valuable in solving those problems in fluid mechanics in which detailed information about the flow process within the control volume may be either not available or rather difficult to evaluate. Thus in order to apply the impulse-momentum equation, a control volume is first chosen which includes the portion of the flow passage which is to be studied. The boundaries of the control volume are usually extended upto such an extent that its end sections lie in the region of uniform flow. All the external forces acting on this control volume are then considered and the momentum equations in the corresponding directions of reference are applied to evaluate the unknown quantities.

In general the impulse-momentum equation is used to determine the resultant forces exerted on the boundaries of a flow passage by a stream of flowing fluid as the flow changes its direction or the magnitude of velocity or both. The problems of this type include the pipe bend, jet propulsion, propellers and stationary and moving plates or vanes. The application of impulse-momentum equation to the problems of pipe bends, jet propulsion and propellers is illustrated in the following paragraphs. However the problems of stationary and moving vanes are discussed in the chapter of 'Impact of Free Jets'. Furthermore, the impulse-momentum equation may also be applied to solve the problems of non-uniform flow in which an abrupt change of flow section occurs. The problems of this type include sudden enlargement in pipes, hydraulic jump in open channels etc., which have been discussed in some of the next chapters.

8.5 FORCE ON A PIPE BEND

As stated earlier the impulse-momentum equation is applied to determine the resultant force (or thrust) exerted by a flowing fluid on a pipe bend. Figure 8.2 shows a reducing pipe bend through which a fluid of density ρ flows steadily from sections 1 to 2. It is desired to find the force exerted by the flowing fluid on the pipe bend. For this the portion of the bend lying between sections 1 and 2 may be chosen as a control volume and all the external forces acting on this may be considered as explained below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(1) At sections 1 and 2 the fluid in the control volume will be subjected to pressure forces p_1A_1 and p_2A_2 by the fluid adjacent to these sections as shown in Fig. 8.2, where p_1, p_2 and A_1, A_2 are the mean pressures and cross-sectional areas at sections 1 and 2 respectively.

(2) The boundary surface of the bend will exert forces on the fluid in the control volume. These forces will be distributed non-uniformly over the curved surface of the bend. But for the ease of computation it is assumed that these distributed forces are equivalent to a single concentrated force R, which has R_x and R_y as its components along x and y directions respectively as shown in Fig. 8.2. It may however be stated that according to Newton's third law of motion, the force exerted by the flowing fluid on the bend, which is required to be determined, will be equal and opposite to R.

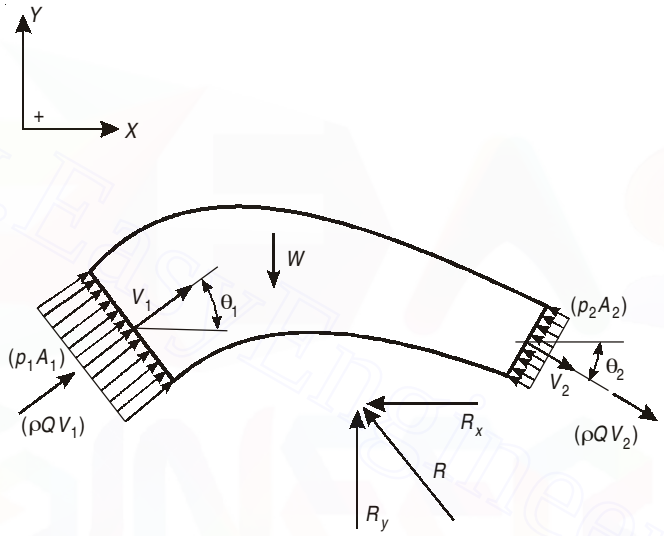


Figure 8.2 *Change of momentum of flow in a reducing pipe bend and forces acting on the bend*

(3) The self-weight W of the fluid in the control volume will be acting in the vertical downward direction.

Thus by applying the impulse-momentum equation in both x and y directions the following expressions are obtained.

For x direction:

$$p_1A_1 \cos\theta_1 - p_2A_2 \cos\theta_2 - R_x = \rho Q (V_2 \cos\theta_2 - V_1 \cos\theta_1) \quad ...(8.12)$$

For y direction:

$$p_1A_1 \sin\theta_1 + p_2A_2 \sin\theta_2 + R_y - W = \rho Q (-V_2 \sin\theta_2 - V_1 \sin\theta_1) \quad ...(8.13)$$

From the above Eqs 8.12 and 8.13 R_x and R_y can be determined from which the magnitude and direction of the force R exerted by the bend on the fluid can be computed. The force (or thrust) exerted by the fluid on the bend is equal and opposite to R.

Often the continuity equation, the energy equation and the equation of state are required to be used to determine the unknown flow characteristics to be used in the above equations. For a horizontal bend in Eq. 8.13 the term W, representing the weight of the fluid in the bend will be eliminated. The method discussed above has been illustrated in Illustrative Ex. 8.2.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The quantity on the right hand side of Eqs. 8.12 and 8.13 is often termed as *dynamic thrust* exerted by the flowing fluid on the bend and *vice-versa*, in order to distinguish it from the static pressure forces appearing on the left hand side of these equations.

8.6 JET PROPULSION–REACTION OF JET

When a jet of fluid issues from an opening and strikes an obstruction placed in its path, it exerts a force on the obstruction. This force exerted by the jet is known as the action of the jet. Recalling Newton's third law of motion, since every action is accompanied by an equal and opposite reaction, the jet while coming out of opening exerts a force on the opening in the form of back kick. This force exerted by the jet on the source from which it is issued is known as the reaction of the jet and is therefore equal in magnitude but opposite in direction to the action of the jet. Further if the source issuing the jet is free to move, it will start moving in the direction opposite to that of jet. Thus the reaction of jet can be utilized for the propulsion of various bodies. The principle of jet propulsion, which is applied in the propulsion of surface ships, aircrafts, rockets etc., may be explained by some of the examples described below.

(a) Jet Propulsion of Orifice Tank. Consider a jet of fluid of area a being issued under a constant head H from an orifice provided in the side of a tank, which is large enough so that the velocity within the tank may be neglected [see Fig. 8.3 (a)]. Let the velocity of the jet be V assumed to be given by $V = C_v \sqrt{2gH}$; and Q be the discharge of fluid coming out of orifice. Applying the impulse-momentum equation, the force exerted on the fluid to change its velocity from O to V is

$$F = \frac{wQ}{g}(V - O) = \frac{waV^2}{g}$$

$$= 2waC_v^2H \qquad \ldots (8.14)$$

This issuing jet will exert a force on the tank which will be the reaction of the jet and its value will be equal to that of F given by Eq. 8.14 but in a direction opposite to V. A physical explanation for the existence of the reaction is that at the vena-contracta the pressure of the fluid is reduced to zero gage

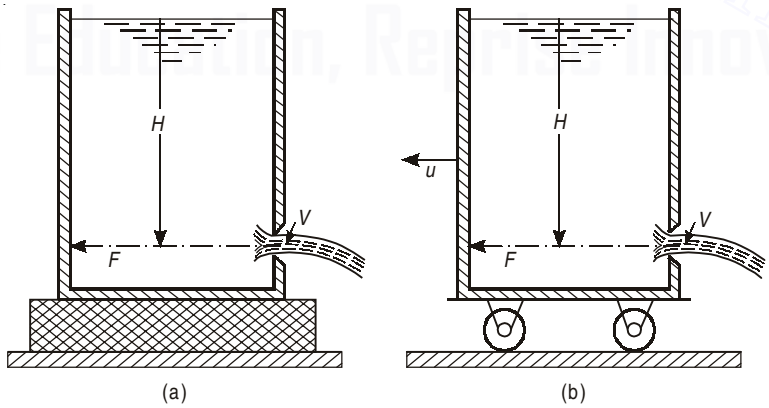


Figure 8.3 *Jet propulsion of an orifice tank*

pressure and there is also a reduction of pressure on the tank walls immediately adjacent to the orifice where the velocity of fluid becomes appreciable. However, on the opposite side of the tank at the same

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

depth the pressure over a corresponding area is wH and the difference of pressure between the two sides of the tank gives rise to the reaction force. Further it is seen from Eq. 8.14 that in the ideal case with no friction since $C_v = 1$, the reaction of the jet is twice the hydrostatic force exerted upon an area of the same size as the jet and at the same depth below the surface.

Now if the tank considered above is mounted on a frictionless trolley as shown in Fig. 8.3 (b), and the orifice is initially kept plugged, then as soon as the orifice is opened the jet will be issued and due to the reaction of the jet the tank will start moving with some velocity say u in the direction opposite to the direction of the issuing jet. Thus the jet issuing from the orifice exerts a propelling force on the tank.

As the tank starts moving with a velocity u, the actual velocity of the issue of the jet will be V_r which is the velocity of the issuing jet relative to the moving tank. Thus V_r will be equal to the vectorial difference of the absolute velocity V of the jet and the velocity of propulsion u of the tank, i.e., $V_r = [V-(-u)] = (V+u)$. The negative sign for u has been considered because its direction is opposite to that of V. Thus applying impulse-momentum equation:

Propelling force, F = Reaction of jet

$$= \frac{W}{g}[(V+u)-u]$$

where W is the weight of fluid actually coming out per second. Since $W = wa\,(V+u)$, as the effective velocity of the efflux of jet is $V_r = (V+u)$, the expression for F becomes

$$F = \frac{wa(V+u)}{g}V \qquad \ldots (8.15)$$

Work done by the jet on the moving tank = $(F \times u)$

Actual kinetic energy of the issuing jet

$$= \frac{WV_r^2}{2g} = \frac{W(V+u)^2}{2g}$$

Hence in this case the efficiency of propulsion

$$\eta = \frac{F \times u}{\frac{WV_r^2}{2g}} = \frac{2Vu}{(V+u)^2} \qquad \ldots (8.16)$$

Again for a given value of V the efficiency will be maximum if $(d\eta/du) = 0$. Thus

$$\frac{d\eta}{du} = 2\left[\frac{(V+u)^2V - 2(V+u)Vu}{(V+u)^4}\right] = 0$$

or $u = V$, (since $V \neq 0$ and also $V \neq -u$)

Substituting the value of u in Eq. 8.16 the value of maximum efficiency is obtained as

$$\eta_{max} = 0.5 \text{ or } 50\%$$

(b) Jet Propulsion of Ships. The principle of jet propulsion was earlier used for the propulsion of small ships. The ship carries centrifugal pumps which lift water from the surrounding sea and discharge it in the form of a jet by forcing through the orifice provided at the back of the ship, as shown

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in Fig. 8.4. The reaction produced by the jet entering the sea propels the ship in the direction opposite to that of the jet.

The pump intakes may have two alternative arrangements. In one case the intakes may face in the same direction as that of the issuing jet or the intakes may be on the sides of the ship. In the second arrangement the pump intakes may face in the direction of the motion of the ship. The main difference in the two arrangements is that if the pump intakes face in the direction of the jet then the water has to be sucked by the pumps against the motion of the ship, accordingly more work will be required to be done by the pumps. On the other hand if the pump intakes face in the direction of motion of the ship then water will enter the pipe intakes due to the movement of the ship itself and hence less work will be required to be done by the pumps.

Let V be the absolute velocity of the issuing jet and u be the velocity of the moving ship. Thus the velocity of the jet relative to the motion of the ship will be $V_r = (V + u)$. Since the effective velocity of the issue of the jet is V_r, the kinetic energy available with the water

$$= \frac{WV_r^2}{2g}$$

where W represents the weight of water issuing from the jet per second. If a represents the area of the issuing jet, then $W = wQ = (waV_r)$.

Applying the impulse-momentum equation in the direction of the jet,

Propelling force $$F = \frac{W}{g}[(V+u)-u] = \frac{W}{g}V \quad \text{... (8.17)}$$

The above expression for the propelling force may be readily derived by bringing the ship to a stationary state before the impulse-momentum equation is applied. For this a velocity equal in magnitude to that of the ship but in opposite direction, i.e., $-u$, is applied to the whole system. Thereby bringing the ship to rest, but making the effective velocity of the jet as $(V + u)$ and also developing a velocity equal to u in the same direction as that of the jet for the water in the surrounding sea. The application of the impulse-momentum equation will then provide the expression for the propelling force as given by Eq. 8.17.

The work done per second on the ship by the reaction of the jet is equal to

$$(F \times u) = \frac{WVu}{g} = \frac{W(V_r - u)u}{g} = \frac{waV_r(V_r - u)u}{g} \quad \text{... (8.18)}$$

which is the output of the system.

Now if it is assumed that the pump intakes face in the same direction as that of the issuing jet, then the energy required to be supplied will be equal to the kinetic energy of the jet. Thus energy supplied per second

$$\frac{WV_r^2}{2g} = \frac{waV_r^3}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

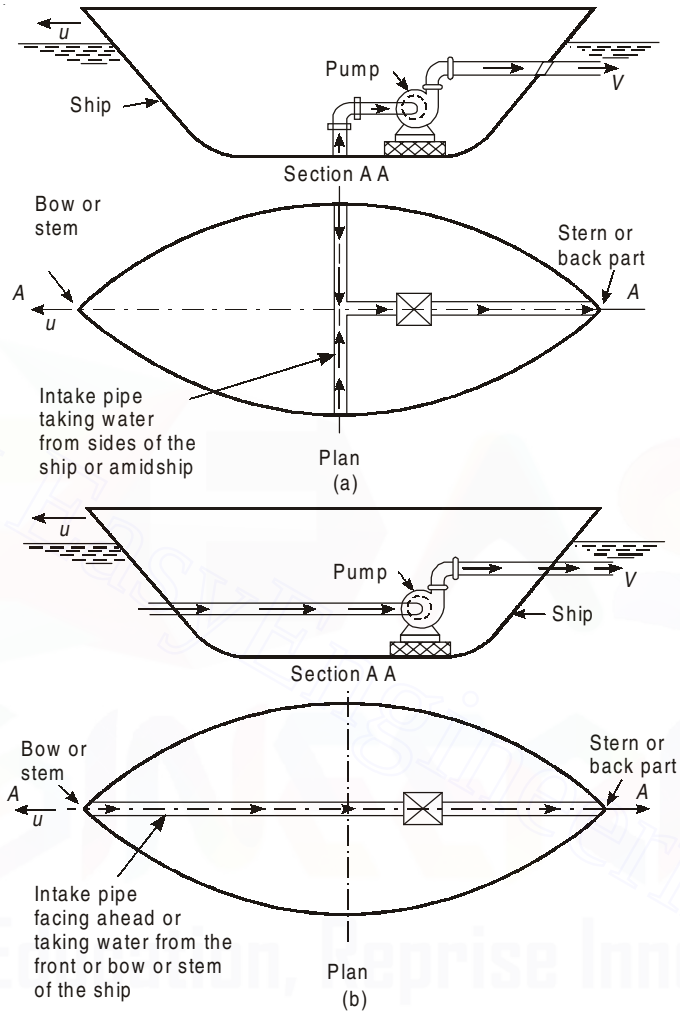


Figure 8.4 *Jet propulsion of ships*

∴ Efficiency of the propulsion

$$\eta = \frac{\frac{waV_r(V_r - u)u}{g}}{\frac{waV_r^3}{2g}} = \frac{2(V_r - u)u}{V_r^2}$$

$$= \frac{2Vu}{(V+u)^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For a given jet velocity V, the condition for maximum efficiency of propulsion is given by $\left(\frac{d\eta}{du}\right) = 0$

Thus $$\frac{d\eta}{du} = \frac{2V[(V+u)^2 - 2(V+u)u]}{(V+u)^4} = 0$$

$$u = V, \text{ (since } V \neq 0 \text{ and also } V \neq -u)$$

Hence for maximum efficiency of propulsion $u = V$ and by substitution

$$\eta_{max} = \frac{2u^2}{(2u)^2} = 0.5 \text{ or } 50\%$$

In the above derivation the loss of head due to friction etc., in the intake and ejecting pipes has been neglected. But if this loss of head is to be considered and is equal to H_L, then the corresponding loss of energy per sec = (WH_L). In which case the work done by the pump or the total energy supplied per second.

$$= \left(\frac{WV_r^2}{2g} + WH_L\right)$$

and then the efficiency of jet propulsion

$$\eta = \frac{\frac{WVu}{g}}{\left(\frac{WV_r^2}{2g} + WHL\right)} = \frac{2Vu}{[V_r^2 + 2gH_L]}$$

In the above case it was assumed that the pump intake faces in the direction of the jet or is on one side of the ship. But if the pump intake faces in the direction of the motion of the ship, then since the water is possessing an initial kinetic energy equal to $\left(\frac{Wu^2}{2g}\right)$, corresponding to the velocity of the moving ship, the energy required to be supplied is reduced by this amount. Hence in this case the energy supplied per second,

$$= \left[\left(\frac{W}{2g}V_r^2\right) - \left(\frac{W}{2g}u^2\right)\right] = \frac{W}{2g}(V_r^2 - u^2)$$

However in this case also the work done by this jet on the ship will be same as represented by Eq. 8.18. Accordingly the efficiency of propulsion will be

$$\eta = \frac{\frac{W}{g}(V_r - u)u}{\frac{W}{g}(V_r^2 - u^2)} = \frac{2u}{(V_r + u)} = \frac{2u}{(V + 2u)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In this case, however it is not possible to derive a practical condition for maximum efficiency. But for $u = V$, which is the condition for the maximum efficiency in the previous case, corresponding value of the efficiency for this case will be

$$\eta = \frac{2u}{u+2u} = \frac{2}{3} = 0.667 \text{ or } 66.7\%$$

Since in actual practice the velocity of the ship u will normally be less than the velocity of the jet V, and therefore the limiting value of u is equal to V. Accordingly the above obtained value of the efficiency may be considered as the maximum possible efficiency for this case.

Again in this case also if the head loss due to friction etc. in the intake and ejecting pipes is equal to H_L, then total energy supplied per second.

$$= \left[\frac{W}{2g}(V_r^2 - u^2) + WH_L\right]$$

Accordingly the efficiency of jet propulsion becomes

$$\eta = \frac{\frac{WVu}{g}}{\left[\frac{W}{2g}(V_r^2 - u^2) + WH_L\right]} = \frac{2Vu}{\left[(V_r^2 - u^2) + 2gH_L\right]}$$

It may however be stated that the jet propulsion in ships is now not commonly adopted because the overall efficiency for such units is much lower than that of screw propeller units.

8.7 MOMENTUM THEORY OF PROPELLERS

A propeller is a revolving mechanism which uses the torque of a shaft to produce axial thrust. The conversion of torque into axial thrust is done by a propeller by changing the momentum of the fluid in which it is submerged. When a propeller submerged in an undisturbed fluid rotates, it exerts a force on the fluid and pushes the fluid backwards. The reaction to this force on the fluid provides a forward force on the propeller itself and this force is the so-called 'propeller thrust' which is used for propulsion. Although the complete design of a propeller cannot be done according to the momentum theory, yet the application of this theory leads to some useful results as indicated by simple analysis of the problem below.

Figure 8.5 shows a propeller moving to the left with velocity V through still fluid. By applying a velocity equal in magnitude to that of the propeller but in opposite direction (i.e., $-V$) on the entire system, the propeller may be considered to be a stationary 'actuating rotor' occupying a fixed position while the fluid flows past it from left to right in the form of a slip stream (that is, the fluid on which it directly acts). The fluid enters the slip stream with velocity V at section 1 upstream from the propeller where the flow is undisturbed, and the fluid velocity increases as it approaches and leaves the propeller. At section 2 some distance behind the propeller, the fluid leaves the slip stream with velocity V_j. For a propeller to operate in a body of still fluid, the pressure at some distance ahead of and behind the propeller (i.e., at sections 1 and 2) and the pressure over the slip stream boundary are the same being

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

equal to the pressure of the undisturbed fluid. As shown in the lower portion of Fig. 8.5, the pressure decreases from the value p_0, rises at the propeller, and then drops to p_0 again. It is assumed that all

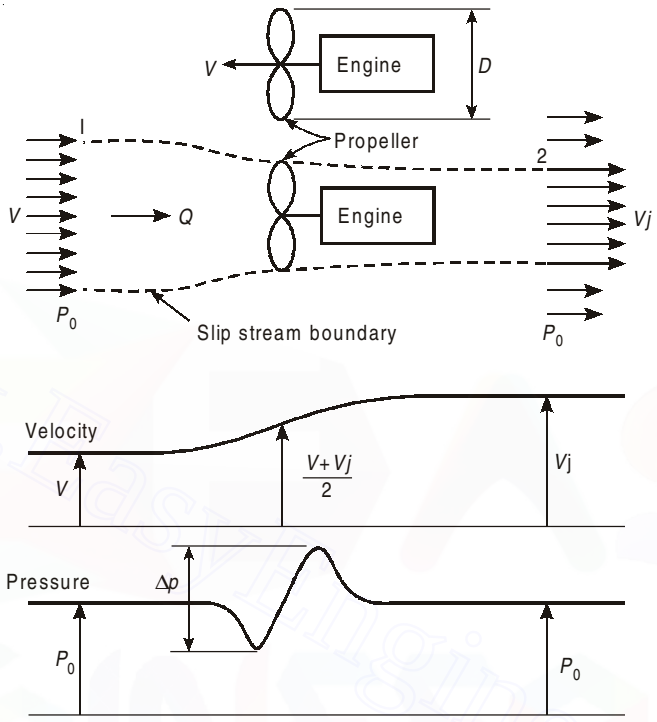


Figure 8.5 *Propeller in a fluid stream*

fluid elements passing through the propeller have their pressure increased by exactly the same amount Δp. The rotational effect of the propeller is neglected. Therefore, the thrust on the propeller is equal to

$$T_p = \frac{\pi D^2}{4}(\Delta p) \quad \ldots (8.19)$$

By applying the momentum equation to the free-body of fluid between sections 1 and 2 and the slip stream boundary, the only force acting on it in the flow direction is propeller thrust T_p, since the outer boundary of the body is everywhere at the same pressure.

Therefore

$$T_p = \rho Q(V_j - V) \quad \ldots (8.20)$$

where Q is the rate of flow through the slip stream.

Applying the energy equation between sections 1 and 2

$$p_0 + \rho \frac{V^2}{2} + \Delta p = p_0 + \rho \frac{V_j^2}{2} \quad \ldots (8.21)$$

in which Δp is the work which the propeller performs on the fluid in the slip stream. Simultaneous solution of the above equations yields

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = \frac{\pi D^2}{4}\frac{V + V_j}{2} \quad \ldots (8.22)$$

Equation 8.22 shows that the velocity of flow at the propeller is the average of the approaching velocity V and the exit velocity V_j. This result known as Froude's theorem after William Froude (1810–79) is one of the principal assumptions in propeller design.

If the undisturbed fluid be considered stationary, the propeller advances through it at velocity V. The rate at which the useful work is done by the propeller is then equal to the product of the propeller thrust T_p and the velocity V. That is

$$\text{Power output} = T_p V = \rho Q(V_j - V)V \quad \ldots (8.23)$$

In addition to the useful work, some power is lost in increasing the kinetic energy of flow in the slip stream, which is given as

$$\text{Power lost} = \rho Q \frac{(V_j - V)^2}{2} \quad \ldots (8.24)$$

since $(V_j - V)$ is the velocity of the downstream fluid relative to earth. The power supplied to the propeller by the engine is the sum of power output and power lost. Thus

$$\text{Power input} = \left[\rho Q(V_j - V)V + \rho Q \frac{(V_j - V)^2}{2}\right] \quad \ldots (8.25)$$

The theoretical propulsive efficiency η_{th}, sometimes known as the Froude efficiency is given by the ratio of the Eqs. 8.23 and 8.25:

$$\eta_{th} = \frac{\text{Power output}}{\text{Power input}}$$

$$= \frac{\rho Q(V_j - V)V}{\left[\rho Q(V_j - V)V + \rho Q \frac{(V_j - V)^2}{2}\right]}$$

$$= \frac{2}{1 + (V_j / V)} \quad \ldots (8.26)$$

It may however be noted that this efficiency does not account for friction or the effects of the rotational motion imparted by the propeller to the fluid, and hence it is considered to be theoretical efficiency. Further the propulsive efficiency, which is a function of the velocity ratio (V_j / V), increases as the ratio (V_j / V) decreases. As may be seen the efficiency will have a limiting value of 100 per cent if (V_j / V) is equal to one. This condition is however impossible, because such a propeller will produce no thrust due to zero velocity change. In practice, an aircraft propeller may have a maximum efficiency of about 0.85 to 0.9 times the value given by Eq. 8.26. However, owing to compressibility effects, the efficiency of an aircraft propeller drops rapidly with speeds above 640 km per hour. Ship propeller efficiencies are usually less, owing to restrictions in diameter.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

8.8 ANGULAR MOMENTUM PRINCIPLE—MOMENT OF MOMENTUM EQUATION

The angular momentum principle states that the torque exerted on any body is equal to the rate of change of angular momentum. The torque is defined as the moment of the force and the angular momentum is defined as the moment of momentum; the moments being taken about the axis of rotation.

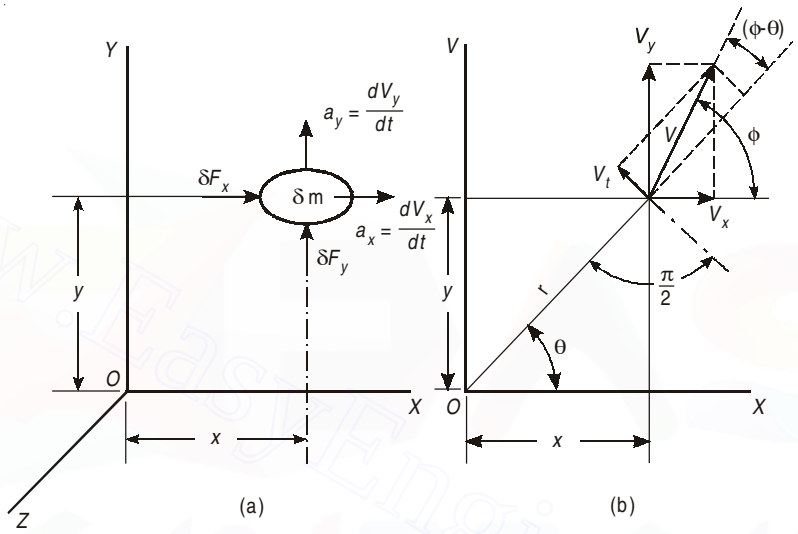


Figure 8.6 *Fluid mass subjected to torque–definition sketch*

Consider a fluid mass δm which is rotating about the z-axis as shown in Fig. 8.6 (a). Let V_x and V_y be its velocity components in x any y directions respectively.

If $a_x = \frac{dV_x}{dt}$ and $a_y = \frac{dV_y}{dt}$ represent the acceleration components of the fluid mass, one obtains

$$\delta F_x = \frac{dV_x}{dt}\delta m,\ \delta F_y = \frac{dV_y}{dt}\delta m$$

where δF_x and δF_y are the components of external forces causing the acceleration.

The moment of the external forces about z-axis (counter-clockwise being considered positive) or the torque δT_z, is then obtained as

$$\delta T_z = (x\,\delta F_y - y\,\delta F_x)$$

$$= \left(x\frac{dV_y}{dt} - y\frac{dV_x}{dt}\right)\delta m$$

By the rules of differentiation

$$\frac{d}{dt}(xV_y - yV_x) = \frac{dx}{dt}V_y - \frac{dy}{dt}V_x + x\frac{dV_y}{dt} - y\frac{dV_x}{dt}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= V_x V_y - V_y V_x + x \frac{dV_y}{dt} - y \frac{dV_x}{dt}$$

Therefore, since $(V_x V_y - V_y V_x) = 0$ and δm is constant,

$$\delta T_z = \frac{d}{dt}\left(xV_y - yV_x\right) \delta m$$

$$= \frac{d}{dt} [(xV_y - yV_x)\delta m] \quad \ldots (8.27)$$

The quantities $(\delta m V_y)x$ and $(\delta m V_x)y$ represent the "moments of momentum" or "angular momentum" about the z-axis. Therefore the right hand side of the above expression represents the rate of change of angular momentum about z-axis, and this is equal to the torque.

In the above derivation, since z-axis is arbitrarily chosen, a torque equation for the x or y axis may also be similarly obtained.

Hence it may be stated that the resultant external torque about any axis is equal to the rate of change of angular momentum about that axis. This is the *law of moment of momentum* (or law of angular momentum).

It is usually convenient to express $(xV_y - yV_x)$ in terms of V_t and r, where V_t is the tangential velocity and r is the radial distance as defined in Fig. 8.6 (b).

From Fig. 8.6 (b) since,

$$x = r \cos\theta; \quad y = r \sin\theta$$

$$V_x = V \cos\phi; V_y = V \sin\phi$$

Hence

$$(xV_y - yV_x) = r \cos\theta (V \sin\phi) - r \sin\theta (V \cos\phi)$$

$$= r V \sin(\phi - \theta)$$

$$= r V_t$$

Thus by substituting in Eq. 8.27

$$\delta T_z = \frac{d(rV_t \delta m)}{dt} \quad \ldots (8.28)$$

Applying Eq. 8.27 or 8.28 to each of the several small fluid masses of a system and summing all the resulting equations, the resultant external torque T_z for a steady flow system is obtained as

$$\Sigma(\delta \Delta T_z) = \frac{\Sigma d(rV_t \delta m)}{dt}$$

or

$$T_z = \rho Q(r_2 V_{t_2} - r_1 V_{t_1}) \quad \ldots (8.29)$$

in which r_2 and V_{t_2} are the radial distance and tangential velocity at section 2 and r_1 and V_{t_1} are the same quantities at section 1 of the control volume.

By rewriting Eq. 8.29 in the form

$$T_z - \rho Q r_2 V_{t_2} + \rho Q r_1 V_{t_2} = 0 \quad \ldots (8.30)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

it can be shown that the moment of the momentum flux across an area about any axis equals the moment of all the external forces applied at the centre of the area about the same axis. Further it may be seen from Eq. 8.30 that if the external forces that act on the fluid mass exert no net moment about a fixed axis (i.e., $T_z = 0$), the moment of momentum of the fluid mass with respect to that axis remains constant. This principle is known as the *law of conservation of moment of momentum* or *the law of conservation of angular momentum.*

The concept of angular momentum is applied in analysing the flow problems, such as flow through turbomachinery, where torques are more significant in the analysis than forces. In Chapter 20 the work done by the flowing fluid on a wheel of a radial flow hydraulic turbine which has radially fixed curved vanes, has been evaluated by applying the principle of angular momentum.

IILUSTRATIVE EXAMPLES

Example 8.1. *Velocity distribution for laminar flow of real fluid in a pipe is given as* $v = V_{max}\,[1-(r^2/R^2)]$, *where* V_{max} *is velocity at the centre of the pipe, R is pipe radius, and v is velocity at radius r from the centre of the pipe. Determine the momentum correction factor.*

Solution

From Eq. 8.10, Momentum correction factor is given as

$$\beta = \frac{1}{AV^2}\int_A v^2 dA$$

Mean velocity

$$V = \frac{Q}{A} = \frac{\int v dA}{A}$$

$$= \frac{\int_0^R V_{\max}\left(\frac{R^2 - r^2}{R^2}\right)(2\pi r dr)}{\pi R^2}$$

$$= \frac{2V_{\max}}{R^4}\int_0^R (R^2 r - r^3)dr$$

$$= \frac{V_{\max}}{2}$$

Thus

$$\beta = \frac{1}{(\pi R^2)\left(\frac{V_{\max}}{2}\right)^2}\int_0^R V_{\max}{}^2\left(\frac{R^2 - r^2}{R^2}\right)^2 (2\pi r dr)$$

$$= \frac{8}{R^6}\left(\frac{1}{2}R^6 - \frac{1}{2}R^6 + \frac{1}{6}R^6\right) = \frac{4}{3} = 1.33.$$

Example 8.2. *A bend in pipeline conveying water gradually reduces from 0.6 m to 0.3 m diameter and deflects the flow through angle of 60°. At the larger end the gage pressure is 171.675 kN/m*^{2} [*1.75 kg(f)/cm*^{2}].

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Determine the magnitude and direction of the force exerted on the bend, (a) when there is no flow, (b) when the flow is 876 litres/s.

Solution

(a) When there is no flow the pressure at both the sections of the bend is same, i.e.,

$$p_1 = p_2 = 171.675 \text{ kN/m}^2 \quad \text{(SI units)}$$

or $$p_1 = p_2 = 1.75 \text{ kg/cm}^2 \quad \text{(Metric units)}$$

Let R be the force exerted on the bend and R_x and R_y be its components as shown in Fig. Ex. 8.2.

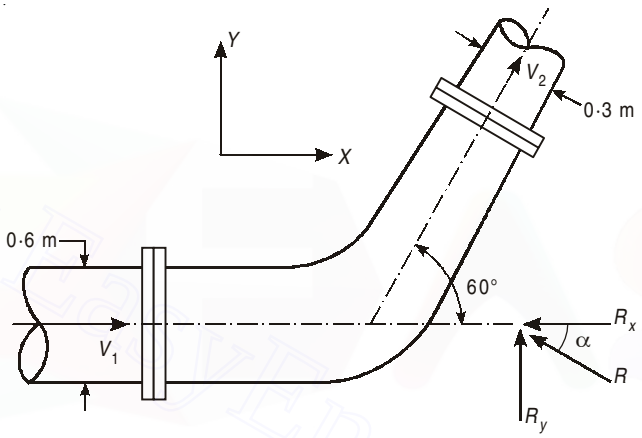


Figure Ex. 8.2

SI Units

By applying Eq. 8.12, we get

$$171.675 \times \frac{\pi}{4}(0.6)^2 - 171.675 \times \frac{\pi}{4}(0.3)^2 \cos 60° - R_x = 0$$

or $$R_x = 42.472 \text{ kN}$$

Similarly by applying Eq. 8.13, we get

$$R_y - 171.675 \times \frac{\pi}{4}(0.3)^2 \sin 60° = 0$$

or $$R_y = 10.509 \text{ kN}$$

∴ Resultant force R on the bend

$$= \sqrt{R_x^2 + R_y^2}$$

$$= \sqrt{(42.472)^2 + (10.509)^2} = 43.753 \text{ kN}$$

which acts to the right at an angle α with x-axis given by

$$\alpha = \tan^{-1}\left(\frac{10.509}{42.472}\right) = 13° 54'$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Metric Units

By applying Eq. 8.12, we get

$$1.75 \times 10^4 \times \frac{\pi}{4}(0.6)^2 - 1.75 \times 10^4 \times \frac{\pi}{4}(0.3)^2 \cos 60° - R_x = 0$$

or $R_x = 4329.507$ kg(f)

Similarly by applying Eq. 8.13, we get

$$R_y - 1.75 \times 10^4 \times \frac{\pi}{4}(0.3)^2 \sin 60° = 0$$

or $R_y = 1071.275$ kg(f)

$\therefore$ Resultant force R on bend

$$= \sqrt{R_x^2 + R_y^2}$$

$$= \sqrt{(4329.507)^2 + (1071.275)^2} = 4460 \text{ kg(f)}$$

which acts to the right at an angle α with x-axis given by

$$\alpha = \tan^{-1}\left(\frac{1071.275}{4329.507}\right) = 13° \ 54'$$

(b) For continuity of flow

$$Q = A_1V_1 = A_2V_2$$

or $876 \times 10^{-3} = \frac{\pi}{4}(0.6)^2 \ V_1 = \frac{\pi}{4}(0.3)^2 \ V_2$

$\therefore$ $V_1 = 3.1$ m/s; and $V_2 = 12.4$ m/s

SI Units

Neglecting frictional losses, and by applying Bernoulli's equation, we get

$$\frac{p1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g}$$

or $\frac{171.675 \times 10^3}{9810} + \frac{(3.1)^2}{2 \times 9.81} = \frac{p_2}{w} + \frac{(12.4)^2}{2 \times 9.81}$

or $\frac{p_2}{w} = (17.50 + 0.49 - 7.84) = 10.15$ m

$\therefore$ $p_2 = (10.15 \times 9\ 810)$

$= 99\ 572 \text{ N/m}^2 = 99.572 \text{ kN/m}^2$

By applying Eq. 8.12, we get

$$171\ 675 \times \frac{\pi}{4}(0.6)^2 - 99572 \times \frac{\pi}{4}(0.3)^2 \cos 60° - R_x$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= (1000 \times 876 \times 10^{-3})\ (12.4 \cos 60° - 3.1)$$

or $R_x = 42\,305 \text{ N} = 42.305 \text{ kN}$

Similarly by applying Eq. 8.13, we get

$$R_y - 99\,572 \times \frac{\pi}{4}(0.3)^2 \sin 60°$$

$$= (1000 \times 876 \times 10\text{-}3)\ (12.4 \sin 60°)$$

or $R_y = 15\,502 \text{ N} = 15.502 \text{ kN}$

$\therefore$ Resultant force F on the bend

$$= \sqrt{R_x^2 + R_y^2}$$

$$= \sqrt{(42.305)^2 + (15.502)}^{\,2} = 45.056 \text{ kN}$$

which acts to the right at an angle α with x-axis given by

$$\alpha = \tan^{-1}\left(\frac{15.502}{42.305}\right) = 20° 8'$$

Metric Units

Neglecting frictional losses, and by applying Bernoulli's equation, we get

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_1^2}{2g}$$

or $$\frac{1.75 \times 10^4}{1000} + \frac{(3.1)^2}{2 \times 9.81} = \frac{p_2}{w} + \frac{(12.4)^2}{2 \times 9.81}$$

or $$\frac{p_2}{w} = (17.5 + 0.49 - 7.84) = 10.15 \text{ m}$$

$\therefore$ $$p_2 = (10.15 \times 1000)$$

$$= 10150 \text{ kg(f)/m}^2 = 1.015 \times 10^4 \text{ kg(f)/m}^2$$

By applying Eq. 8.12, we get

$$1.75 \times 10^4 \times \frac{\pi}{4}(0.6)^2 - 1.015 \times 10^4 \times \frac{\pi}{4}(0.3)^2 \cos 60° - R_x$$

$$= \frac{1000 \times 876 \times 10^{-3}}{9.81}(12.4 \cos 60° - 3.1)$$

or $R_x = 4312.46 \text{ kg(f)}$

Similarly by applying Eq. 8.13, we get

$$R_y - 1.015 \times 10^4 \times \frac{\pi}{4}(0.3)^2 \sin 60°$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{1000 \times 876 \times 10^{-3}}{9.81}(12.4 \sin 60°)$$

or $$R_y = 1580.27 \text{ kg(f)}$$

$\therefore$ Resultant force F on the bend

$$= \sqrt{R_x^2 + R_y^2}$$

$$= \sqrt{(4312.46)^2 + (1580.27)^2} = 4\,593 \text{ kg(f)}$$

which acts to the right at an angle α with x-axis given by

$$\alpha = \tan^{-1}\left(\frac{1580.27}{4312.46}\right) = 20° 8'$$

Exapmle 8.3. *Water flows through a 0.9 m diameter pipe at the end of which there is a reducer connecting to a 0.6 m diameter pipe. If the gage pressure at the entrance to the reducer is 412.02 kN/m² [4.2kg(f)/cm²] and the velocity is 2 m/s, determine the resultant thrust on the reducer, assuming that the frictional loss of head in the reducer is 1.5 m.*

Solution

For continuity of flow

$$\frac{\pi}{4}(0.9)^2 \times 2 = \frac{\pi}{4}(0.6)^2 \times V_2$$

$$V_2 = 4.5 \text{ m/s}$$

SI Units

Applying Bernoulli's equation, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g} + h_f$$

or $$\frac{412.02 \times 10^3}{9810} + \frac{(2)^2}{2 \times 9.81} = \frac{P_2}{w} + \frac{(4.5)^2}{2 \times 9.81} + 1.5$$

or $$\frac{p_2}{w} = (42.0 + 0.204 - 1.032 - 1.5) = 39.672 \text{ m}$$

$\therefore$ $$p_2 = (39.672 \times 9810)$$

$$= 389\,182 \text{ N/m}^2 = 389.182 \text{ kN/m}^2$$

Let F_x be the force exerted by the reducer on the fluid, acting opposite to the direction of flow, then applying Eq. 8.12, we get

$$412.02 \times 10^3 \frac{\pi}{4}(0.9)^2 - 389.182 \times 10^3 \times \frac{\pi}{4}(0.6)^2 - F_x$$

$$= 1000 \times \frac{\pi}{4}(0.9)^2 \times 2(4.5 - 2)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $F_x = 148\,896 \text{ N} = 148.896 \text{ kN}$

∴ The resultant thrust exerted by the fluid on the reducer =148.896 kN, which is acting in the direction of flow.

Metric Units

Applying Bernoulli's equation, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_f$$

or $$\frac{4.2\times10^4}{1000} + \frac{(2)^2}{2\times9.81} = \frac{P_2}{w} + \frac{(4.5)^2}{2\times9.81} + 1.5$$

or $$\frac{p_2}{w} = (42.0 + 0.204 - 1.032 - 1.5) = 39.672 \text{ m}$$

∴ $$p_2 = (39.672 \times 1000)$$

$$p_2 = 39.672 \times 10^3 \text{ kg(f)/m}^2$$

Let F_x be the force exerted by the reducer on the fluid, acting opposite to the direction of flow, then applying Eq. 8.12, we get

$$4.2\times10^4 \times \frac{\pi}{4}(0.9)^2 - 39.672\times10^3 \times \frac{\pi}{4}(0.6)^2 - F_x$$

$$= \frac{1000}{9.81} \times \frac{\pi}{4}(0.9)^2 \times 2(4.5-2)$$

or $F_x = 15\,178 \text{ kg(f)}$

∴ The resultant thrust exerted by the fluid on the reducer = 15 178 kg(f), which is acting in the direction of flow.

Example 8.4. *A tank 1.5 m high stands on a trolley and is full of water. It has an orifice of diameter 0.1 m at 0.3 m from the bottom of the tank. If the orifice is suddenly opened, what will be the propelling force on the trolley? Coefficient of discharge of the orifice is 0.60.*

Solution

Discharge from the orifice

$$= C_d\, a\, \sqrt{2gH}$$

$$= 0.60 \times \frac{\pi}{4}(0.1)^2\, (2 \times 9.81 \times 1.2)^{1/2}$$

$$= 0.023 \text{ m}^3/\text{s}$$

Velocity of the jet issuing from the orifice

$$= \frac{Q}{a} = \frac{0.023\times4}{\pi\times(0.1)^2} = 2.93 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Propelling force $F = \dfrac{wQV}{g}$

$= \dfrac{9810 \times 0.023 \times 2.93}{9.81} = 67.39$ N

Example 8.5. *The resistance to motion of a vessel is 24.525 kN at a velocity of 4.5 m/s. The jet efficiency is to be 80% and the mechanical efficiency of the pumps is 75% hydraulic losses in the ducts are 5% of the relative kinetic energy at exit. Determine (a) the velocity of the jet; (b) the orifice area at exit; (c) the power required to drive the pumps for the given speed of the vessel, assuming that the water is drawn in through the intakes facing in the direction of the motion of the ship.*

Solution

(*a*) $u = 4.5$ m/s and the efficiency = 80%

But efficiency

$$\eta = \frac{2u}{V_r + u}$$

or $0.8 = \dfrac{2 \times 4.5}{V_r + 4.5}$

$V_r = 6.75$ m/s

and $V = 2.25$ m/s

(b) Work done per second

= resistance × velocity of vessel

= (24.525 × 4.5) = 110.363 kN m/s

$= \dfrac{W}{g}(V_r - u)\,u$

$= \dfrac{W}{g}(6.75 - 4.5)\,4.5$

$\therefore \quad W = \dfrac{110.363 \times 9.81}{4.5 \times 2.25}$

= 106.929 kN/s = 106929 N/s

$W = waV_r$

$\therefore \quad a = \dfrac{W}{wVr} = \dfrac{106929}{9810 \times 6.75} = 1.615$ m^2

Area of orifice at exit = 1.615 m^2

(c) Power supplied to jet

$= \dfrac{W}{2g}(V_r^2 - u^2)$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Loss of energy in ducts

$$= 0.05 \times \frac{V_r^2}{2g} \times W$$

∴ Total power required from pumps

$$= \frac{W}{2g}\left[(V_r^2 - u^2) + 0.05 \quad V_r^2\right]$$

$$= \frac{106929}{2 \times 9.81}\left[(6.75^2 - 4.5^2) + 0.05 \times 6.75^2\right] W$$

$$= 150\ 369\ W = 150.369 \text{ kW}$$

The mechanical efficiency of the pumps is 75%

∴ Power required to drive pumps

$$= \frac{150.369}{0.75} = 200.5 \text{ kW}$$

Example 8.6. *A ship whose resistance is 24.525 kN is to be driven at 5 m/s by means of a jet of water directed under water. The velocity of the jet is to be 7.5 m/s relative to the ship. The efficiency of the pump operating the jet is estimated to be 80%, the frictional resistance of the pipes being equal to 3 m of water. Calculate (a) the power required to drive the pump; (b) the overall efficiency of the system in the following cases: (i) the water enters the ship through an inlet facing ahead; (ii) the water enters through an inlet in the side of the ship.*

Solution

$$u = 5 \text{ m/s}; V_r = 7.5 \text{ m/s}; R = 24.525 \text{ kN}$$

$$\eta_p = 0.8; \text{ and } H_L = 3 \text{ m}$$

The reaction of the jet should be just equal to the resistance to the motion of the ship

$$F = R = 24.525 \text{ kN}$$

But $$F = \frac{W}{g} V = \frac{W}{g}(V_r - u)$$

or $$24.525 = \frac{W}{9.81}(7.5 - 5)$$

∴ $$W = 96.236 \text{ kN} = 96\ 236 \text{ N}$$

(a) (i) When the water enters the ship through an inlet facing ahead, the output of the pump

$$= \left[\left(\frac{WVr^2}{2g} - \frac{Wu^2}{2g}\right) + WH_L\right]$$

$$= W\left[\frac{(V_r^2 - u^2)}{2g} + H_L\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

∴ Output of the pump

$$= 96\ 236 \left[\frac{(7.5^2 - 5^2)}{2 \times 9.81} + 3.0\right]$$

$$= 441\ 989 \text{ W} = 441.989 \text{ kW}$$

$$\therefore \text{ Input of the pump} = \frac{\text{Output}}{h_p}$$

$$= \frac{441.989}{0.8} = 552.486 \text{ kW}$$

∴ Power required to drive the pump

$$= 552.486 \text{ kW}$$

(ii) When the water enters through an inlet in the side of the ship, the output of the pump

$$h = \left[\frac{WV_r^2}{2g} + WH_L\right]$$

$$= W\left[\frac{V_r^2}{2g} + H_L\right]$$

∴ Output of the pump

$$= 96\ 236 \left[\frac{7.5^2}{2 \times 9.81} + 3.0\right]$$

$$= 564\ 614 \text{ W} = 564.614 \text{ kW}$$

∴ Input of the pump

$$= \frac{\text{Output}}{h}$$

$$= \frac{564.614}{0.8} = 705.768 \text{ kW}$$

(b) (i) The overall efficiency of the system for this case

$$\eta = \frac{F \times u}{\text{Imput of the pump}}$$

$$= \frac{24.525 \times 10^3 \times 5}{552.486 \times 10^3}$$

$$= 0.222 \text{ or } 22.2\%$$

(ii) The overall efficiency of the system for this case

$$\eta = \frac{F \times u}{\text{Imput of the pump}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{24.525 \times 10^3 \times 5}{705.768 \times 10^3}$$

$$= 0.174 \text{ or } 17.4\%$$

Example 8.7. *The diameter of a pipe bend is 0.3 m at inlet and 0.15 m at outlet and the flow is turned through 120° in a vertical plane. The axis at inlet is horizontal and the centre of the outlet section is 1.5 m below the centre of the inlet section. The total volume of fluid contained in the bend is 0.085 m³. Neglecting friction, calculate the magnitude and direction of the force exerted on the bend by the water flowing through it at 225 l/s when the inlet pressure is 137.34 kN/m².*

Solution

For continuity of flow

$$Q = A_1V_1 = A_2V_2$$

or $$225 \times 10^{-3} = \frac{\pi}{4}(0.3)^2 V_1 = \frac{\pi}{4}(0.15)^2 V_2$$

$\therefore$ $$V_1 = 3.18 \text{ m/s; and } V_2 = 12.73 \text{ m/s}$$

Neglecting friction losses, by applying Bernoulli's equation, we get

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + Z_2$$

or $$\frac{137.34 \times 10^3}{9810} + \frac{(3.18)^2}{2 \times 9.81} + 1.5 = \frac{p_2}{w} + \frac{(12.73)^2}{2 \times 9.81} + 0$$

or $$\frac{p_2}{w} = (14.0 + 0.515 + 1.5 - 8.26) = 7.755 \text{ m}$$

$\therefore$ $$p_2 = (7.755 \times 9810)$$

$$= 76\,077 \text{ N/m}^2 = 76.077 \text{ kN/m}^2$$

By applying Eq. 8.12, we get

$$137.34 \times 103 \times \frac{\pi}{4}(0.3)^2 - 76.077 \times 10^3 \times \frac{\pi}{4}(0.15)^2 \cos 120° - F_x$$

$$= 1000 \times 225 \times 10^{-3} (12.73 \cos 120° - 3.18)$$

or $$F_x = \frac{\pi}{4}(0.15)^2 [549.36 \times 10^3 + 38.039 \times 10^3] + 225(6.37 + 3.18)$$

or $$F_x = 12529 \text{ N} = 12.529 \text{ kN}$$

Similarly by applying Eq. 8.13, we get

$$F_y + (0.085 \times 9810) - 76.077 \times 103 \times \frac{\pi}{4}(0.15)^2 \sin 60°$$

$$= 1000 \times 225 \times 10^{-3} (12.73 \sin 60°)$$

or $$F_y = (2480.51 - 833.85 + 1164.28)$$

$$= 2810.94 \text{ N} \approx 2.811 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $F = \sqrt{F_x^2 + F_y^2}$

$= \sqrt{(12.529)^2 + (2.811)^2} = 12.84$ kN

$\tan \alpha = \dfrac{2.811}{12.529} = 0.2244$

$\therefore \quad \alpha = 12°39'$

$\therefore$ Force of 12.84 kN acts on the bend at an angle of 12°39′ upwards from inlet axis.

Example 8.8. *A boat travelling at 12 m/s in fresh water has a 0.6 m diameter propeller which takes 4.25 m^3 of water per second between its blades. Assuming that the effects of the propeller hub and the boat hull on flow conditions are negligible, calculate the thrust on the boat, the theoretical efficiency of the propulsion, and the power input to the propeller.*

Solution

From Eq. 8.22, we have

$$Q = \frac{\pi D^2}{4} \frac{V + V_j}{2}$$

or $4.25 = \dfrac{\pi}{4}(0.6)^2 \dfrac{12 + V_j}{2}$

$\therefore \quad V_j = 18.06$ m/s

From Eq. 8.20, we have

$$T_p = \rho Q(V_j - V)$$

or $T_p = 1000 \times 4.25\ (18.06 - 12)$

$\therefore \quad T_p = 25\ 755$ N $= 25.755$ kN

Again from Eq. 8.26, we have

$$\eta_{th} = \frac{2}{1 + (V_j / V)}$$

$$= \frac{2}{1 + (18.06/12)} = 0.798 \text{ or } 79.8\%$$

From Eq. 8.25

$$\text{Power input} = \left[\rho Q(V_j - V)V + \rho Q \frac{(V_j - V)^2}{2}\right]$$

$$= \rho Q (V_j - V)\left[V + \frac{V_j - V}{2}\right]$$

$$= 1000 \times 4.25(18.06 - 12)\left[12 + \frac{18.06 - 12}{2}\right]$$

$= 387\ 098$ W $= 387.098$ kW

Example 8.9 *A water sprinkler has 10 mm diameter nozzles at either end of a rotating arm, each of which is discharging water in opposite direction at right angle to the rotating arm, at a velocity of 8 m/s. If the axis of*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

rotation is at a distance of 0.15 m from one end and 0.2 m from the other, determine the torque required to hold the arm stationary. If friction is neglected, determine the constant angular speed of the arm.

Solution

The rate of change of moment of momentum is the torque required to hold the arm stationary.

Initial moment of momentum is zero. Final moment of momentum

$$= \rho Q(V_2 r_2 + V_1 r_1)$$

$$\therefore \quad \text{Torque } T = \rho Q(V_2 r_2 + V_1 r_1)$$

$$= 1000 \times \left[\frac{\pi}{4} \times (0.01)^2 \times 8\right][8 \times 0.2 + 8 \times 0.15]$$

$$= 1.759 \text{ N-m}$$

If the angular velocity of the sprinkler is ω, then the absolute velocities of flow through the nozzle are

$$V_1 = 8 - 0.15\,\omega$$

and $$V_2 = 8 - 0.2\,\omega$$

Since the moment of momentum of flow entering is zero and there is no friction, the moment of momentum leaving the sprinkler must also be zero.

Thus $$\rho Q[(8 - 0.15\,\omega) \times 0.15 + (8 - 0.2\,\omega) \times 0.2] = 0$$

or $$\omega = \frac{2.8}{0.0625} = 44.8 \text{ rad/s}$$

Example 8.10. *Two different liquids which are miscible are mixed in a device shown in Fig. Ex. 8.10. The device consists of a pipe of diameter 10 cm with a bend at one end. A smaller pipe of diameter 5 cm and negligible wall thickness is introduced into it as shown in the figure and a liquid of specific gravity 0.8 is pumped through it at a constant rate so that it issues out at section 1 with a uniform velocity of 6 m/s. The other liquid, whose specific gravity is 0.9, is pumped through the larger pipe and it has a uniform velocity of 3 m/s at section 1. The pressure is the same in both the fluid streams at section 1. At section 2 the mixed stream has the same density, velocity and pressure at every location. The net resistive force acting along the pipe wall between the sections 1 and 2 is estimated to be 5 kg(f).*

Determine the pressure drop and rate of energy dissipation between the sections 1 and 2.

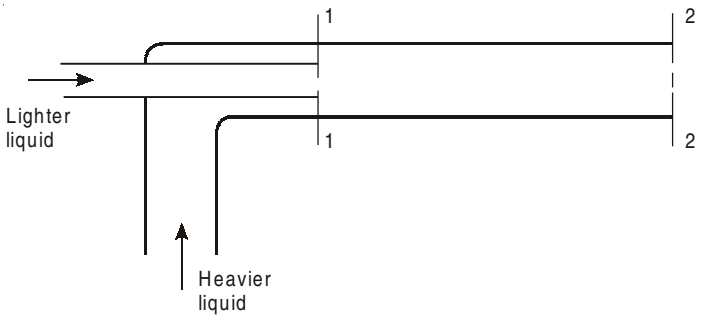


Figure Ex. 8.10

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

At section 1, discharge through smaller pipe is

$$Q_1' = \frac{\pi}{4} \times (0.05)^2 \times 6 = 0.01178 \text{ m}^3/\text{s}$$

and discharge through the larger pipe is

$$Q_2' = \frac{\pi}{4}\left[(0.10)^2 - (0.05)^2 \right] \times 3 = 0.01767 \text{ m}^3/\text{s}$$

$\therefore$ Total discharge through the larger pipe between sections 1 and 2 is

$$Q = (Q_1' + Q_2')$$

$$= (0.01178 + 0.01767) = 0.02945 \text{ m}^3/\text{s}$$

$\therefore$ Velocity of flow at section 2 is

$$V_2 = \frac{0.02945}{(\pi/4) \times (0.10)^2} = 3.75 \text{ m/s}$$

Specific gravity of the mixed liquid is

$$\rho' = \frac{(0.8 \times 0.01178) + (0.9 \times 0.01767)}{(0.01178 + 0.01767)} = 0.86$$

If p_1 and p_2 are the pressures at sections 1 and 2, then applying momentum equation between sections 1 and 2, we get

$$(p_1 - p_2) \times \frac{\pi}{4}(0.10)^2 - 5 = \frac{0.86 \times 1000}{9.81} \times 0.02945 \times 3.75$$

$$-\left[\frac{0.8 \times 1000}{9.81} \times 0.01178 \times 6 + \frac{0.9 \times 1000}{9.81} \times 0.01767 \times 3 \right]$$

or $$(p_1 - p_2) \times \frac{\pi}{4}(0.10)^2 - 5 = 9.6816 - (5.7639 + 4.8633)$$

$\therefore$ $$(p_1 - p_2) = \frac{(5 - 0.9456)}{(\pi/4) \times (0.10)^2}$$

$$= 516.22 \text{ kg/m}^2 = 0.0516 \text{ kg/cm}^2$$

Rate of energy dissipation between sections 1 and 2

= Rate of inflow of energy at section 1 – Rate of outflow of energy at section 2

Rate of inflow of energy at section 1

$$= (0.8 \times 1000 \times 0.01178)\left[\frac{p_1}{0.86 \times 1000} + \frac{(6)^2}{2 \times 9.81} \right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$+(0.9\times1000\times0.01767)\left[\frac{p_1}{0.86\times1000}+\frac{(3)^2}{2\times9.81}\right]$$

$$= (0.02945\,p_1 + 24.5867)\ \text{kg-m/s}$$

Rate of outflow of energy at section 2

$$= (0.86\times1000\times0.02945)\left[\frac{p_2}{0.86\times1000}+\frac{(3.75)^2}{2\times9.81}\right]$$

$$= (0.02945\,p_2 + 18.1530)\ \text{kg-m/s}$$

∴ Rate of energy dissipation between sections 1 and 2

$$= \left[(0.02945\,p_1 + 24.5867) - (0.02945\,p_2 + 18.1530)\right]$$

$$= 0.02945(p_1 - p_2) + (24.5867 - 18.1530)$$

$$= (0.02945\times516.22) + 6.4337$$

$$= 21.6364\ \text{kg-m/s}$$

SUMMARY OF MAIN POINTS

1. The impulse-momentum principle (or simply momentum principle) states that the impulse exerted on any body is equal to the resulting change in momentum of the body. Mathematically it may be expressed as

   $F(dt) = d(M)$

   in which $F(dt)$ is impulse and $d(M)$ is the resulting change of momentum.

2. The impulse momentum equation for steady flow of fluid between sections 1 and 2 may be written as

   $$\Sigma F = \rho Q (V_2 - V_1)$$

   where

   ΣF = resultant external force acting on the fluid;
   ρ = mass density of the fluid;
   Q = discharge of fluid;
   V_1 = velocity of flow at section 1 and
   V_2 = velocity of flow at section 2.

3. The impulse-momentum equations for x, y and z directions may be written as

   $$\Sigma F_x = \rho Q [(V_2)_x - (V_1)_x]$$
   $$\Sigma F_y = \delta Q [(V_2)_y - (V_1)_y]$$
   $$\Sigma F_x = \delta Q [(V_2)_z - (V_1)_z]$$

   in which the components of the resultant external force and those of the velocities in the respective directions are considered.

3. The momentum correction factor β is given as

   $$\frac{1}{AV^2}\int v^2 dA$$

4. The force exerted by the fluid on a pipe bend is given as

   For x direction

   $R_x = [(p_1A_1)_x - (p_2A_2)_x] - \delta Q [(V_2)_x - (V_1)_x]$

   For y direction

   $R_y = \delta Q [(V_2)_y - (V_1)_y] + [(p_2 A_2)_y - (p_1A_1)_y] + W$

   However, if the bend is located in horizontal plane then the self weight of the fluid $W = 0$.

   The force R_x acts along the x axis in the direction of flow and the force R_y acts along the y axis in the downward direction. The resultant force R exerted by the fluid on the bend is given as

   $$R = \sqrt{R_x^2 + R_y^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which acts at an angle θ with the x axis given as

$$\tan\theta = \frac{R_y}{R_x}$$

5. The efficiency of jet propulsion of a ship is given as:

(i) When the pump intake faces in the same direction as that of the issuing jet

$$\eta = \frac{2Vu}{(V+u)^2 + 2gH_L}$$

(ii) When the pump intake faces in the direction of the motion of the ship

$$\eta = \frac{2Vu}{V(V+2u) + 2gh_L}$$

where

V = velocity of the issuing jet;
u = velocity of the moving ship;
H_L = head loss due to friction etc. in the intake and the ejecting pipes; and
g = acceleration due to gravity.

6. The theoretical propulsive efficiency η_{th} of a propeller also called Froude efficiency is given as

$$\eta_{th} = \frac{2}{1+(V_j/V)}$$

where

V_j = velocity of the fluid leaving the slip stream or exit velocity; and
V = velocity of propeller moving through still fluid or velocity of fluid entering the ship stream or approach velocity.

7. The angular momentum principle states that the torque exerted on a rotating fluid mass is equal to the rate of change of angular momentum. The torque is defined as the moment of the force and the angular momentum is defined as the moment of momentum; the moments being taken about the axis of rotation. Mathematically it is given as

$$T = \delta Q (r_2 V_2 - r_1 V_1)$$

where

T = torque exerted on the fluid;
V_2 = velocity at a radial distance r_2; and
V_1 = velocity at a radial distance r_1

PROBLEMS

8.1 The velocity distribution in a pipe is given by

$$v = V_{max}\left(1-\frac{r}{R}\right)^K$$

where R is the radius of the pipe, r is any radius at which the velocity is v and K is a constant index. Find the momentum correction factor.

$$\left[Ans.\ \frac{(K+1)^2(K+2)^2}{2(2K+1)(2K+2)}\right]$$

8.2 Two large plates are spaced 50 mm apart. If the velocity profile between the plates is represented by $v = V_{max}\ (1–1600\ r^2)$ where r is measured in m from the centre line between the plates, determine the momentum correction factor. [*Ans.* 1.2]

8.3 Water at a pressure of 294.3 kN/m² [3kg(f)/cm²] flows through a horizontal pipe of 100 mm diameter with a velocity of 2 m/s.

(a) If the diameter of the pipe gradually reduces to 50 mm what is the axial force on the pipe assuming no loss of energy.

(b) If a bend is connected to the pipe which turns through 30° and tapers uniformly from 100 mm to 50 mm. Find the force exerted on the bend other than those due to gravity.

[*Ans.* (a) 1.698 kN { 173.08 kg (f) } ; (b) 1.804 kN {183.94 kg(f) }]

8.4 A pipe of 1 m diameter carrying 2.5 m³/s of water, is deflected through a 90° bend. The ends of the bend are anchored by the rods at right angles to the bend (one tie rod at each end). Find the tension in each rod. Also determine the resultant dynamic thrust on the bend and the direction of this thrust.

[*Ans.* 7.958kN { 811.2 kg(f)};11.254 kN {1147.2 kg(f) } at 45° with horizonal]

8.5 A 100 mm diameter orifice at the end of a 150 mm diameter pipe yields a jet of oil 85 mm in diameter. What force will be exerted upon the orifice plate when the pressure intensity

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the approaching flow is 13.734 kN/m² [0.14 kg(f)/cm²].

[*Ans.* 117.72 N{12 kg(f)}]

8.6 If a 0.552 kW (0.75 h.p.) motor is required by a ventilating fan to produce a 0.6 m stream of air having a velocity of 12 m/s, what is the efficiency of the fan? Take r = 1.207 kg/m³ (0.123 msl/m³).

[*Ans.* 53.4%]

8.7 A small ship is fitted with jets of total area 0.65 m². The velocity through the jets relative to ship is 9 m/s and ship's speed is 18.5 km/hour. The engine efficiency is 85%; the pump efficiency is 65% and pipe losses are equal to 10% of the kinetic energy of the jets. Determine the propelling force and the overall efficiency. Water is drawn in amid ship (i.e., at the middle of the ship from sides) and sea water weighs 10.055 kN/m³ [1025 kg(f)/m³].

[*Ans.* 23.145 kN{2359.4 kg(f)}; 24.6%]

8.8 A jet propelled boat has two jets each of 100 mm diameter. The boat travels at 30 km/hour. If the resistance of the motion of the boat is 88.29 u^2 N [9u^2 kg(f)] where u is the velocity of the boat in m/s, calculate:

(a) the velocity of the jet relative to boat;

(b) the power required to drive the pump if the pump efficiency is 80%; and

(c) the overall efficiency.

Assume that the pump intake is facing the direction of motion of the boat.

[*Ans.* (a) 24.36 m/s; (b) 125.30 kW (170.33 h.p.); (c) 50.96%]

8.9 To propel a light aircraft at an absolute velocity of 250 km per hour against a head wind of 48 km per hour a thrust of 10.3 kN [1050 kg(f)] is required. Assume a theoretical efficiency of 90% and a constant air density of 1.207 kg/m³ (0.123 msl/m³), determine the diameter of ideal propeller required and the power needed to drive it.

[*Ans.* 2.54 m; 947.45 kW (1288 h.p.)]

8.10 A pipeline 0.6 m diameter conveying oil (sp. gr. 0.85) at the flow rate of 1800 litres per second has a 90° bend in the horizontal plane. The pressure at the entrance to the bend is 147.15 kN/m² [1.5 kg(f)/cm²] and loss of head in the bend is 2 m of oil. Find the magnitude and direction of the force exerted by the oil on the bend.

[*Ans.* 69.368 kN{7071kg(f)} to the right at 42°15' with horizontal]

8.11 A nozzle at the end of a 80 mm hose produces a jet 40 mm in diameter. Determine the longitudinal stress in the joint at the base of the nozzle when it is discharging 1200 litres of water per minute.

[*Ans.* 358.37 N{36.53 kg(f)} tensile]

8.12 A motor boat is driven at 4 m/s by means of a jet of water issuing from an opening 100 mm square directly behind the boat and having discharge equal to 0.5 m³/s. Find the driving force. The coefficient of contraction of the jet is 0.62.

[*Ans.* 38.322 kN {3906.5 kg (f)}]

8.13 Oil of sp. gr. 0.8 flows through a horizontal pipe of diameter 100 mm which is provided with a nozzle 25 mm diameter. If the pressure at the base of the nozzle is 784.8 kN/m² [8 kg(f)/cm²), find the force exerted on the nozzle.

[*Ans.* 5.439 kN{554.39 kg(f)}]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Flow Through Orifices and Mouthpieces

Chapter 9

9.1 DEFINATION

An *orifice* is an opening having a closed perimeter, made in the walls or the bottom of a tank or a vessel containing fluid, through which the fluid may be discharged.

A *mouthpiece* is a short tube of length not more than two to three times its diameter, which is fitted to a circular opening or orifice of the same diameter, provided in a tank or a vessel containing fluid, such that it is an extension of the orifice and through which also the fluid may be discharged.

Both the orifices and the mouthpieces are usually used for measuring the rate of flow of fluid.

9.2 CLASSIFICATIONS OF ORIFICES AND MOUTHPIECES

The orifices may be classified on the basis of their size, shape, shape of the upstream edges and the discharge conditions. According to the size, the orifices may be classified as *small and large orifices*. According to the shape, the orifices may be classified as *circular, rectangular, square and triangular*. Out of these, however, circular and rectangular orifices are most commonly used. The upstream edge of the orifice may be made either sharp or it may be rounded. Thus according to the shape of the upstream edge the orifices may be classified as *sharp-edged orifices* and *bell-mouthed orifices* or orifices with the round corners. As shown in Fig. 9.1, a sharp-edged orifice has the bevelled side facing the downstream so that there is a minimum contact with the fluid flowing through the orifice and consequently the minimum frictional effects. A sharp-edged orifice is considered as a standard orifice and it is mostly used for the purposes of discharge measurement. The orifices may be made to discharge fluid from the tank freely in the atmosphere or it may be made to discharge fluid into the same fluid contained on the downstream side of the orifice such that the whole or the part of the outlet side of the orifice is submerged in the same fluid. Thus according to these discharge conditions, the orifices may be classified as *orifices discharging free* and *drowned* or *submerged orifices*. The drowned or submerged orifices may be further classified as *fully submerged orifices* (or totally submerged or totally drowned orifices) and *partially submerged orifices*.

The mouthpieces may be classified on the basis of their shape, position and the discharge conditions. According to the shape the mouthpieces may be classified as *cylindrical, convergent, divergent* and *convergent-divergent.* According to the position the mouthpieces may be classified as *external* and *internal mouthpieces*. An external mouthpiece is the one which is fitted to tank or reservoir such that it is projecting outside the tank or reservoir and it may be of any of the shapes noted above. An internal mouthpiece is also called *re-entrant or Borda's mouthpiece* which is fitted to a tank or a reservoir such

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

that it is projecting inside the tank or reservoir and it is generally of cylindrical shape only. According to the discharge conditions the mouthpieces may be classified as *running full* and *running free*

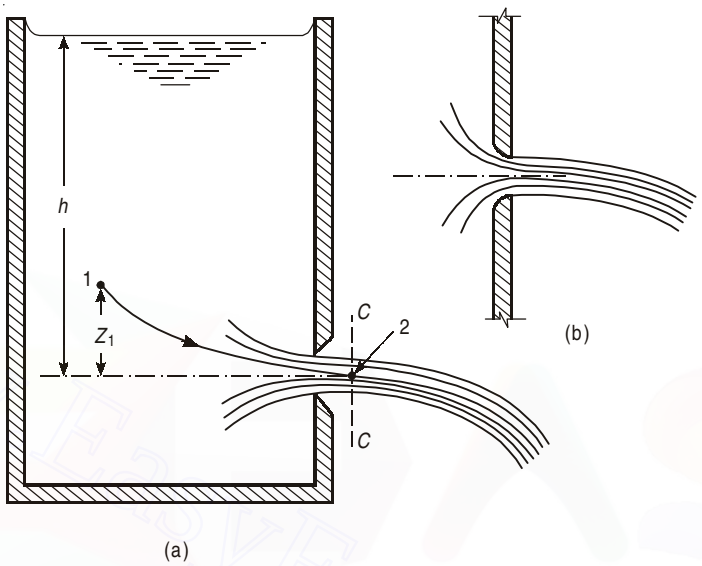


Figure 9.1 (*a*) *Sharp-edged orifice discharging free* (*b*) *Bell-mouthed orifice*

mouthpieces. Generally these conditions of discharge may be developed only in the case of internal mouthpieces.

9.3 SHARP-EDGED ORIFICE DISCHARGING FREE

Figure 9.1 shows a sharp-edged small orifice in one side of a reservoir containing liquid. The liquid will emerge from the orifice as a free jet, that is, a jet discharged in the atmosphere and will therefore be under the influence of gravity only. It may be observed that the liquid approaching the orifice gradually converges towards it, to form a jet whose cross-sectional area is less than that of the orifice. This is so because the liquid particles lying close to the inner wall, have a motion, along the wall towards the orifice, which cannot be abruptly changed in direction at the orifice edge. Since an abrupt change of direction of motion is impossible, the streamlines continue to converge beyond the orifice upto a certain distance until they become parallel at the section *C–C*. Strict parallelism of the streamlines should be approached asymptotically in an ideal fluid; but in practice however, frictional effects, produce the section *C–C* with parallel flow at only a short distance (about half the orifice diameter) from the orifice. The section *C–C* of the jet, at which the streamlines are straight and parallel to each other and perpendicular to the plane of the orifice, and the jet has the minimum cross-sectional area, is known as *vena-contracta* (Latin word meaning contracted vein or jet). Beyond the section *C–C* the jet may, however, diverge again and it undergoes a downward deflection due to gravity.

Since at vena-contracta the streamlines are parallel and straight, the pressure in the jet at this section is uniform and it is equal to the pressure of the fluid surrounding the jet, which in the case of a free jet, is thus equal to the atmospheric pressure. Moreover, at vena-contracta since the jet has the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

least cross-sectional area, the velocity of flow of liquid at this section will be maximum by the principle of continuity.

Let it be assumed that the flow through the orifice is steady under a constant head h measured above the centre of the orifice. Then considering two points 1 and 2 along a particular streamline the point 1 being inside the reservoir and the point 2 being at the centre of the jet at vena-contracta, as shown in Fig. 9.1, and applying the Bernoulli's equation between the points 1 and 2, we have, neglecting the loss of energy between these points,

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + 0$$

where p_1 and V_1 are respectively the pressure intensity and velocity of flow at point 1; z_1 is the height of point 1 above the datum which is taken as a horizontal plane passing through the centre of the orifice; p_2 and V_2 are respectively the pressure intensity and velocity of flow at point 2, for which the datum head is zero, and w is the specific weight of the liquid. Since V_1 is the velocity of the liquid approaching the orifice, it is often termed as *velocity of approach*.

Since the hydrostatic conditions prevail, the pressure p_1 may be expressed as

$$p_1 = p_a + w(h - z_1)$$

where p_a is the intensity of atmospheric pressure.

Also $$p_2 = p_a$$

Thus by substituting for p_1 and p_2 in the above expression, we obtain

$$\frac{V_2^2}{2g} = \frac{V_1^2}{2g} + h \qquad ...(9.1)$$

Now if the orifice is small in comparison with the head h, the velocity of jet V_2 may be considered to be constant across the vena-contracta. Further if the velocity of approach V_1 is assumed to be constant across the cross-section of the reservoir at point 1, then by the equation of continuity the discharge Q may be expressed as

$$Q = a_1 V_1 = a_c V_2$$

where a_1 is the cross-sectional area of the reservoir and a_c is the cross-sectional area of the jet at vena-contracta. By substituting for V_1 in Eq. 9.1 the expression for the velocity of flow of jet V_2 may be obtained as

$$V_2 = \sqrt{\frac{2gh}{1 - \left(\frac{a_c}{a_1}\right)^2}} \qquad ...(9.2)$$

However, if the reservoir is assumed to be sufficiently large and the point 1 is considered to be sufficiently far from the orifice then the velocity of approach V_1 will be very small in comparison with V_2 and hence it may be neglected in Eq. 9.1, in which case

$$\frac{V_2^2}{2g} = h$$

or $$V_2 = \sqrt{2gh} \qquad ...(9.3)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 9.3 is known as *Torricelli's formula* in honour of Evangelista Torricelli (1608–47) who in 1643 demonstrated, experimentally that the velocity with which a jet of liquid emerges from a small orifice is proportional to the square root of the head above the orifice. It may however be noted that Eq. 9.3 gives a velocity of jet at vena-contracta which is always more than the average velocity of the jet at a section in the plane of the orifice itself where neither the pressure nor the velocity is constant.

In the above analysis the loss of energy that takes place as the liquid flows through the orifice has been neglected. But in actual practice as the real fluid flows through an orifice there is always some loss of energy due to friction and surface tension. As such Eq. 9.3 represents only the ideal (or theoretical) velocity of the jet and the actual velocity of the jet at vena-contracta is slightly less than the ideal velocity given by Eq. 9.3. The actual velocity of jet at vena-contracta may therefore be determined by multiplying the ideal velocity by a factor called coefficient of velocity.

The *coefficient of veloeity* C_v is defined as the ratio between the actual velocity of jet at vena-contracta and the ideal (or the theoretical) velocity of the jet. Thus if V is the actual velocity of the jet at vena-contracta then

$$C_v = \frac{V}{V_{th}} = \frac{V}{\sqrt{2gh}}$$

or
$$V = C_v \sqrt{2gh} \qquad ...(9.4)$$

The difference between the theoretical and the actual velocities of the jet at vena-contracta is mainly due to friction at the orifice. For sharp-edged orifices since the liquid passing through these touches only a line, the friction is considerably less and hence the difference between the theoretical and the actual velocities is very small. Therefore, for sharp-edged orifices discharging water and other liquids of similar viscosity the value of C_v is slightly less than unity. Experimentally it has been observed that the value of C_v varies from 0.95 to 0.99 for different orifices, depending on the shape and size of the orifice and on the head of liquid under which the flow takes place. In general an average value of 0.97 or 0.98 may be assumed for C_v, for sharp-edged orifices discharging water and other liquids of similar viscosity. For orifices which are not sharp-edged the value of C_v may be markedly lower.

As indicated earlier a jet of liquid issuing from an orifice has its cross-sectional area at vena-contracta less than the area of the orifice. In other words the jet of liquid issuing from an orifice undergoes a contraction. The actual area of the jet at vena-contracta may therefore be determined by multiplying the area of the orifice by a factor called coefficient of contraction.

The *coefficient of contraction* C_c is defined as the ratio between the area of the jet at vena-contracta and the area of the orifice. Thus if a_c is area of the jet at vena-contracta and a is the area of the orifice, then

$$C_v = \frac{a_c}{a}$$

or
$$a_c = C_c\, a \qquad ...(9.5)$$

The theoretical value of C_c for a sharp-edged orifice is equal to $\left(\frac{\pi}{\pi+2}\right)$ which equals 0.611, but in practice its value varies from 0.61 to 0.69 depending on the size and shape of the orifice and the head of liquid under which the flow takes place. However, for general purposes for small sharp-edged orifices an average value of 0.64 or 0.65 may be assumed for C_c.

In the case of a bell-mouthed orifice as shown in Fig. 9.1 (b) the contraction after the orifice is almost

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

completely eliminated. The coefficient of contraction is therefore equal to unity for a bell-mouthed orifice. But on account of friction the coefficient of velocity for a bell-mouthed orifice is considerably reduced. Moreover, the velocity at the edge of the jet is reduced due to friction and so the velocity over the cross-section of jet emerging from a bell-mouthed orifice is non-uniform.

Theoretically the cross-sectional area of the jet of liquid issuing from an orifice will be equal to the area of the orifice, which may be considered as an ideal (or theoretical) cross-sectional area of the jet. The product of the ideal cross-sectional area of the jet and the ideal velocity of jet given by Eq. 9.3 will therefore give ideal (or theoretical) discharge. However, on account of the effect of friction due to which the actual velocity of the jet is reduced and due to the contraction of the jet, the actual discharge of liquid through an orifice is always less than the ideal (or theoretical) discharge. The actual discharge of liquid through an orifice may therefore be determined by multiplying the ideal discharge by a factor called coefficient of discharge.

The *coefficient of discharge* C_d is defined as the ratio between the actual discharge issued from an orifice and its theoretical (or ideal) discharge.

Thus if Q_{th} is the theoretical discharge and Q is the actual discharge emerging from an orifice then

$$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} = \frac{Q}{Q_{th}} \quad ...(9.6)$$

But actual discharge Q = (actual area of the jet at vena-contracta × actual velocity of jet there)

i.e., $$Q = (a_c \times V) = (C_c a) \times (C_v \sqrt{2gh})$$

Similarly, theoretical discharge Q_{th} = (theoretical area of jet × theoretical velocity of jet)

i.e., $$Q_{th} = (a \times \sqrt{2gh})$$

By substituting the values in Eq. 9.6

$$C_d = \frac{Q}{Q_{th}} = \frac{(C_c a) \times \left(C_v \sqrt{2gh}\right)}{a \times \sqrt{2gh}}$$

Therefore

$$C_d = C_c \times C_v \quad ...(9.7)$$

Equation 9.7 indicates that the coefficient of discharge C_d of an orifice is equal to the product of the coefficient of contraction C_c and the coefficient of velocity C_v and it may therefore be found by first determining C_c and C_v of the orifice and by multiplying these together. The coefficient of discharge varies from 0.61 to 0.65 depending on the size and the shape of the orifice, and the head of liquid under which the flow takes place. For general purposes for small sharp-edged orifices discharging water or liquids of similar viscosity, an average value of 0.62 or 0.63 may be adopted for C_d.

In addition to the three coefficients for an orifice defined above, one more coefficient may be introduced which is called the coefficient of resistance.

The *coefficient of resistance* C_r is defined as the ratio of the loss of kinetic energy as the liquid flows through an orifice and the actual kinetic energy possessed by the flowing fluid.

The loss of kinetic energy as the liquid flows through an orifice is equal to the difference between the theoretical kinetic energy and the actual kinetic energy.

Theoretical kinetic energy per unit weight of liquid

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{V_{th}^2}{2g} = \frac{(\sqrt{2gh})^2}{2g} = h$$

Actual kinetic energy per unit weight of liquid

$$= \frac{V^2}{2g} = \frac{C_v^2\left(\sqrt{2gh}\right)^2}{2g} = hC_v^2$$

$\therefore$ Loss of kinetic energy per unit weight of liquid

$$= (h - C_v^2 h) = h\,(1 - C_v^2)$$

$$\therefore \quad C_r = \frac{h(1-C_v^2)}{hC_v^2} = \left(\frac{1}{C_v^2} - 1\right) \qquad ...(9.8)$$

The coefficient of resistance thus accounts for the loss of energy that occurs as any liquid flows through an orifice and its value may be determined by knowing the value of the coefficient of velocity C_v of the orifice.

Often the values of the various coefficients for an orifice, defined above are required to be determined in the laboratory in order to calibrate the orifice for being used as a measuring device. The various methods commonly adopted for determining these coefficients in the laboratory are discussed in the next section.

9.4 EXPERIMENTAL DETERMINATION OF THE COEFFICIENTS FOR AN ORIFICE

There are several methods by which the value of each of these coefficients of an orifice may be determined, some of which are commonly adopted, are described below.

(a) Determination of the Coefficient of Velocity C_v. The different methods which are adopted for the determination of C_v are as follows:

(i) Jet Distance Measurement Method. Consider a tank provided with a small orifice on one of the sides and an arrangement for the supply of liquid to the tank as shown in Fig. 9.2. By adjusting the inflow of the liquid in the tank equal to the outflow (or efflux) through the orifice a constant head h of the liquid above the centre of the orifice is maintained under which a jet of liquid emerges through the orifice. Let V be the velocity of the jet at vena-contracta. Now consider a liquid particle in the jet of the liquid, which is at vena-contracta at any instant of time and in time t as it is carried along the jet it occupies a new position represented by a point in the jet as shown in Fig. 9.2. If x and y are respectively the horizontal and the vertical distances of the point from the vena-contracta, then it may be considered that as the fluid particle moves from its original position at vena-contracta to its new position at the point in time t, it is displaced horizontally through a distance x from the vena-contracta and at the same time it drops down under the action of gravity through a vertical distance y from the vena-contracta. If the air resistance is negligible the horizontal component V of jet velocity remains unchanged (as there is no acceleration acting on the jet in the horizontal direction), and hence the horizontal distance travelled by a particle in time t after it has left the vena-contracta is

$$x = Vt \qquad ...(i)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

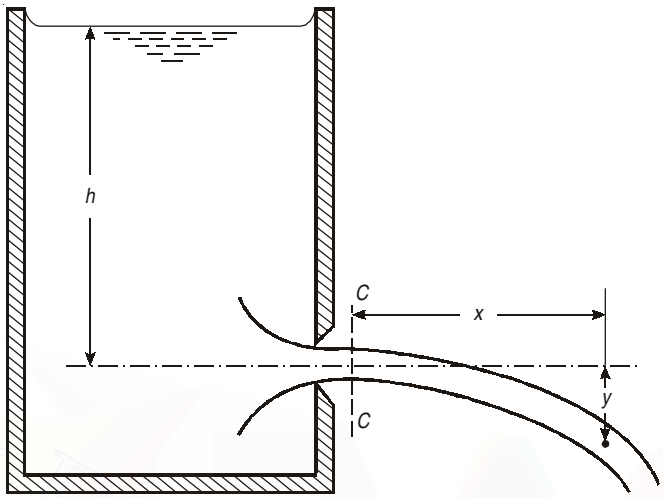


Figure 9.2 *Determination of C_v by jet distance measurement method*

In the vertical direction since there is a uniform downward acceleration g, and the vertical component of velocity is initially zero for horizontal discharge, the vertical distance travelled by the particle in the same time t is

$$y = \frac{1}{2} gt^2 \qquad ...(ii)$$

By eliminating t between the Eqs. (i) and (ii), we have

$$x^2 = \frac{2V^2}{g} y \text{ ; or } V = \sqrt{\frac{gx^2}{2y}} \qquad ...(9.9)$$

which shows that the path of a liquid particle moving along a free-jet of liquid is parabolic, with the vertex of the parabola at the vena-contracta.

Equation 9.9 represents the actual velocity of flow of jet at vena-contracta. The theoretical velocity of jet emerging from an orifice under a constant head h is given by Eq. 9.3 as

$$V_{th} = \sqrt{2gh}$$

Therefore coefficient of velocity

$$C_v = \frac{V}{V_{th}} = \frac{\sqrt{\frac{gx^2}{2y}}}{\sqrt{2gh}} = \sqrt{\frac{x^2}{4hy}} \qquad ...(9.10)$$

If the coordinates, x and y, of a point in the jet and the head h of the liquid are measured, then Eq. 9.10 may be used to obtain the value of the coefficient of velocity C_v of the orifice. For the measurement of the coordinates of any point in the jet generally a scale and sliding apparatus is fitted to the orifice

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

tank. It consists of a simple triangular frame at the base of which a graduated scale is provided which facilitates the measurement of the x coordinate of any point in the jet. A vertical graduated scale which can slide on the horizontal scale is also provided, on which the vertical depth y of the point in the jet can be determined. In order to facilitate the exact location of the point in the jet a hooked pointer is provided which is attached to the vertical scale.

(ii) Velocity Measurement Method. In this method the actual velocity of jet at vena-contracta is measured by inserting a previously calibrated pitot tube as shown in the Fig. 9.3. The theoretical velocity may however be computed by measuring the head h over the centre of the orifice. Knowing the actual and the theoretical velocities of flow of liquid through the orifices, its coefficient of velocity may be computed by Eq. 9.4. However, this method is suitable only if the jet diameter is not too small.

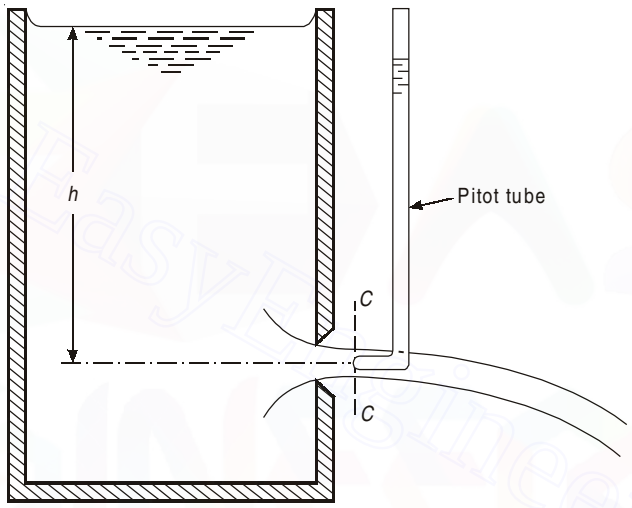


Figure 9.3 *Determination of C_v by the velocity measurement method*

The actual velocity of the jet at vena-contracta may also be determined by measuring the cross-sectional area of the jet at vena-contracta and the actual discharge of liquid flowing through the orifice under a constant head h. Thus if Q is the discharge actually flowing through the orifice and a_c is the cross-sectional area of the jet at vena-contracta then the actual velocity of the jet at vena-contracta is

$$V = \frac{Q}{a_c}$$

Again $$V_{th} = \sqrt{2gh}$$

$\therefore$ Coefficient of velocity

$$C_v = \frac{V}{V_{th}} = \frac{Q}{a_c\sqrt{2gh}}$$

(iii) Momentum Method. This method involves the impulse momentum equation for the determination of the actual velocity of jet at vena-contracta. As shown in Fig. 9.4 a tank provided with an orifice in one of its sides and containing liquid is supported in a suspended position on knife edges on either

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

side of the tank. On the vertical side of the tank opposite to that having the orifice a short projecting platform is provided on which the desired weights may be placed. By adjusting the inflow equal to the outflow through the orifice a constant head h of the liquid is maintained above the centre of the orifice.

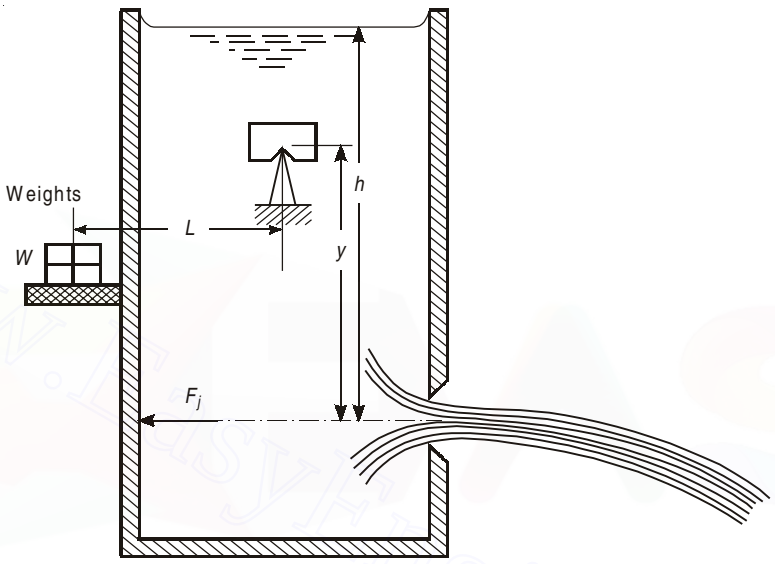


Figure 9.4 *Determination of C_v by the momentum method*

Originally when no liquid is discharged through the orifice the tank will remain in a vertical position. But as the jet of liquid is issued from the orifice some change of momentum takes place for the liquid flowing through the orifice, which results in exerting a force F_j on the tank in the direction opposite to that of the flowing jet. The force exerted on the tank causes the tank to be tilted towards the issuing jet from its vertical position. The vertical position of the tank is however restored by placing the necessary weights on the platform.

The force exerted by the issuing jet on the tank may be computed by the application of the impulse-momentum principle. Thus if W_l is the weight of the liquid discharged through the orifice per second and V is the velocity of the issuing jet of liquid at vena-contracta, then the momentum per second of liquid leaving the tank in the horizontal direction is $\left[\frac{(W_l V)}{g}\right]$. Neglecting the velocity of approach the initial momentum possessed by this liquid is equal to zero. Therefore the rate of change of momentum of liquid leaving the tank is $\left[\frac{(W_l V)}{g}\right]$ which is according to impulse-momentum principle equal to the force F_j exerted on the tank by the issuing jet. That is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F_j = \frac{W_l V}{g}$$

Due to this force exerted on the tank, it will get tilted. Now let W be the weight required to be placed on the platform in order to counter balance the effect of the force of the issuing jet and to bring back the tank to its vertical position. Then by taking the moments of the forces about the knife edge, we have

$$W \times L = F_j \times y = \frac{W_l V}{g} \times y$$

or
$$V = \frac{W}{W_l} \times \frac{L}{y} \times g \qquad ...(9.11)$$

The weight of the liquid emerging from the orifice per second W_l, can be determined by actually collecting the liquid in a tank for a known period of time and weighing the same. Thus all the terms except the velocity V of the jet are known in Eq. 9.11, from which the actual velocity of the jet at vena-contracta can be determined. Again the theoretical velocity of the jet is equal to $\sqrt{2gh}$ which may be computed by measuring the head h above the centre of the orifice. The coefficient of velocity C_v is then given by

$$C_v = \frac{V}{V_{th}} = \frac{WLg}{W_l y\sqrt{2gh}} \qquad ...(9.12)$$

(b) Determination of the Coefficient of Contraction C_c. The coefficient of contraction for an orifice may be determined either by a direct method or by an indirect method. In the direct method of the determination of C_c the radius of the jet at vena-contracta is determined with the help of a micrometer contraction gage. As shown in Fig. 9.5 (a) the gage consists of a circular collar or a ring which is provided with four micrometer screws fitted at the diametrically opposite points at right angles to each other. The pointed ends of the screws are so adjusted that in closed position they all meet at the centre of the ring and the reading on the scale of each screw is zero and when opened these may be made to touch the outer periphery of the jet as shown in Fig. 9.5 (b). However there may be some zero error which may be noted beforehand. In order to determine the radius of the jet of liquid issuing from an orifice under a constant head, the gage with its screws open is held in front of the orifice at vena-contracta such that the jet of liquid passes through the ring. The screws are then moved and so adjusted that their pointed ends are just touching the outer periphery of the jet. The reading of each of the micrometer screws (corrected for the zero error if any) then gives the radius of the jet. The mean of the readings of all the four screws will give a fairly accurate value of the radius of the jet from which the actual area of the jet at vena-contracta may be computed. The area of the orifice being already known the coefficient of contraction C_c of the orifice may then be computed from Eq. 9.5.

The above described method is, however, applicable only for the determination of C_c of a circular orifice. Moreover even for a circular orifice the jet of liquid issuing from the orifice is quite irregular and hence it is quite difficult to measure the correct radius of the jet at vena-contracta. Therefore, generally the indirect method for the determination of the coefficient of contraction for an orifice is adopted, which in addition to being applicable for an orifice of any shape, avoids the cumbersome process of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

measurement of the actual area of the jet at vena-contracta. In this indirect method the value of C_c is obtained by using the relation derived from Eq. 9.7 as

$$C_c = \frac{C_d}{C_v}$$

in which the values of C_d and C_v for an orifice are determined previously.

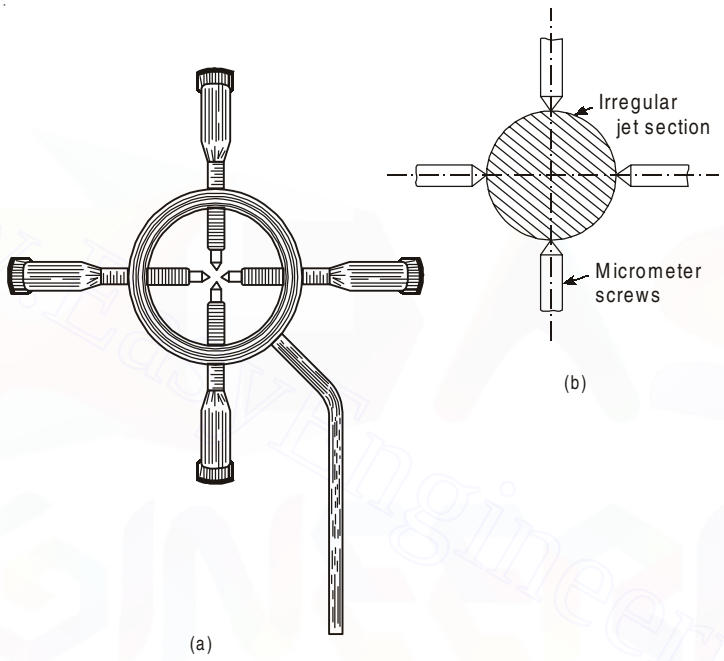


Figure 9.5 (*a*) *Micrometer contraction gage* (*b*) *Screws touching the jet periphery*

(c) Determination of the Coefficient of Discharge C_d. The coefficient of discharge for an orifice may be more easily determined than the other coefficients. To determine C_d for an orifice, it is allowed to discharge under a constant head and for a known period of time the liquid passing through the orifice is collected in a tank and it is measured either by weight or by volume. The actual discharge Q passing through the orifice may then be computed by dividing the volume of the liquid collected by the time during which it is collected. The theoretical discharge of the orifice being equal to $(a \times \sqrt{2gh})$ which may be computed from the known area of the orifice a and the constant head h under which it is discharging. The value of the coefficient of discharge for the orifice may then be determined as

$$C_d = \frac{Q}{a \times \sqrt{2gh}} \qquad ...(9.13)$$

The value of C_d for an orifice may also be determined by first determining its C_c and C_v and then multiplying the two in accordance with Eq. 9.7 to obtain the value of C_d. But generally this method is not preferred because as stated earlier the value of C_c cannot be determined accurately by the direct experimental method, which will result in giving an inaccurate value of C_d for the orifice.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The coefficient of resistance C_r for an orifice may be computed by using Eq. 9.8 if the value of the coefficient of velocity C_v for the orifice is known.

9.5 FLOW THROUGH LARGE VERTICAL ORIFICE

So far we have dealt with an orifice whose vertical dimension has been small in comparison to the head, in which case the velocity in the entire cross-section of the jet could be considered to be constant and equation $Q = C_d\, a\sqrt{2gh}$ was applicable for the computation of the discharge actually passing through the orifice. But if a vertical orifice provided in the side of a tank has its vertical dimension large enough as compared with the head then the velocity of the liquid flowing through such an orifice varies over the entire cross-section of the jet because of a considerable variation in head at different points in the vertical section of the orifice. Therefore for a large vertical orifice, since the velocity of the flowing liquid cannot be considered as constant for the enitre cross-section, the discharge cannot be computed by the same equation as that for a small orifice, but it has to be determined by integrating the discharges through small elements of the area. The expressions for the discharge flowing through large rectangular and circular orifices are derived below.

(a) Large Rectangular Orifice. Consider a large, vertical rectangular orifice of breadth b, depth d, provided in the side of a reservoir, and discharging into the atmosphere. A vena-contracta of breadth b_c and depth d_c is formed as shown in Fig. 9.6. The streamlines at vena-contracta are parallel and practically straight, thus the pressure at any point in the plane of the vena-contracta is atmospheric. Consider an elementary horizontal strip of depth dh out of the area of the vena-contracta at a depth h below the free surface of liquid in the reservoir. The area of the strip is $(b_c dh)$ and the velocity of flow through the strip is $C_v\sqrt{2gh}$, and hence the discharge through the elementary strip equals

$$dQ = C_v\sqrt{2gh} \times b_c dh$$

where C_v is the coefficient of velocity for the orifice.

The total discharge of liquid passing through the orifice may be obtained by integrating the above expression across the plane of the vena-contracta. Thus if H_c represents the depth of the centre of the vena-contracta below the free surface of the liquid then h varies from $(H_c - d_c/2)$ to $(H_c + d_c/2)$ and the total discharge is

$$Q = C_v b_c \sqrt{2g} \int_{(H_c - d_c/2)}^{(H_c + d_c/2)} h^{1/2} dh$$

$$= \frac{2}{3} C_v b_c \sqrt{2g} \left\{ \left(H_c + \frac{d_c}{2} \right)^{3/2} - \left(H_c - \frac{d_c}{2} \right)^{3/2} \right\}$$

However the above derived expression for the discharge through a large rectangular orifice involves the quantities such as b_c, d_c and H_c which are rather difficult to be determined. This difficulty may be overcome by introducing b, d and H respectively in place of b_c, d_c and H_c and a coefficient of contraction C_c, where H is the depth of the centre of the orifice below the free surface in the reservoir. Then

$$Q = \frac{2}{3} C_c C_v b \sqrt{2g} \left\{ \left(H + \frac{d}{2} \right)^{3/2} - \left(H - \frac{d}{2} \right)^{3/2} \right\}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$Q = \frac{2}{3} C_d b \sqrt{2g} \left\{ \left(H + \frac{d}{2} \right)^{3/2} - \left(H - \frac{d}{2} \right)^{3/2} \right\} \quad ...(9.14)$$

where C_d is the coefficient of discharge for the orifice .

Often Eq. 9.14 is expressed in a slightly different form. Thus if the height of the free surface above the top edge of the orifice is H_1 and that above the bottom edge of the orifice is H_2, then since

$$H_1 = \left(H - \frac{d}{2} \right)$$

and $$H_2 = \left(H + \frac{d}{2} \right)$$

$$Q = \frac{2}{3} C_d b \sqrt{2g} \left\{ H_2^{3/2} - H_1^{3/2} \right\} \quad ...(9.15)$$

However, if the same rectangular orifice is treated as a small orifice the discharge through it may be expressed as

$$Q = C_d (b \times d) \sqrt{2gH} \quad ...(9.16)$$

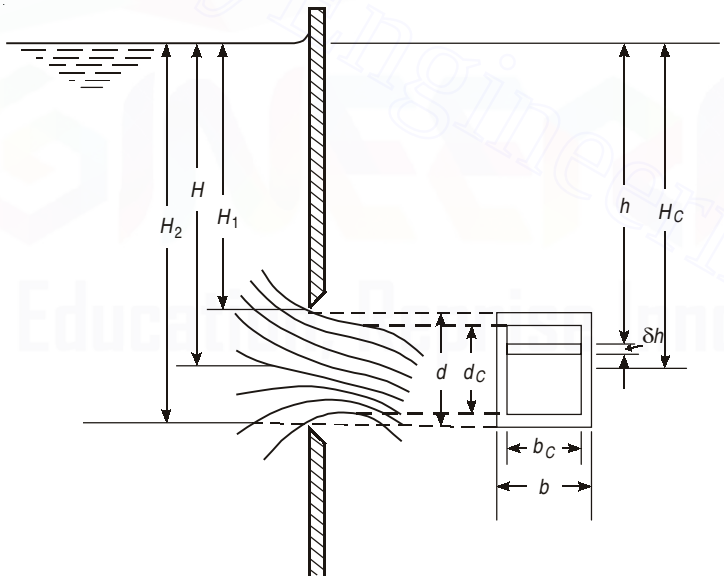


Figure 9.6 *Large rectangular orifice*

By comparing the values of the discharge computed by Eqs 9.14 and 9.16 it can be shown that there is only a slight difference in the two values which may however be neglected provided the head H has certain minimum value. Thus if $H = d$ then from Eq. 9.14

$$Q = 0.99[C_d (b \times d) \sqrt{2gH}\,]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which shows that for the head above the centre of the orifice equal to its depth there is an error of only 1% in the actual discharge computed by Eq. 9.14 and the one computed by treating it as a small orifice and using Eq. 9.16. Moreover for $H > d$ the use of Eq. 9.16 in place of Eq. 9.14 will give an error of even less than 1%. *Thus as long as $H > d$,* i.e., *head above the centre of the orifice is greater than its vertical dimension, it may be treated as a small orifice.*

In deriving the above expression for the discharge through a large rectangular orifice the velocity of approach has been neglected. But if the velocity of approach is not negligible and if V_1 is the velocity of approach then the discharge through the elementary strip equals

$$dQ = C_v \sqrt{2g\left(h + \frac{V_1^2}{2g}\right)} \times b_c dh$$

By integrating this, the expression for the total discharge through the orifice is obtained as

$$Q = \frac{2}{3} C_d b \sqrt{2g} \left\{ \left(H + \frac{d}{2} + \frac{V_1^2}{2g} \right)^{3/2} - \left(H - \frac{d}{2} + \frac{V_1^2}{2g} \right)^{3/2} \right\}$$

or

$$Q = \frac{2}{3} C_d b \sqrt{2g} \left\{ \left(H_2 + \frac{V_1^2}{2g} \right)^{3/2} - \left(H_1 + \frac{V_1^2}{2g} \right)^{3/2} \right\} \quad ...(9.17)$$

(b) Large Circular Orifice. Consider a circular orifice of diameter d provided in the side of a reservoir containing liquid upto a height H above the centre of the orifice which is discharging liquid into the atmosphere. A vena-contracta of diameter d_c will be formed as shown in Fig. 9.7. Again the streamlines at vena-contracta are parallel and practically straight and hence the pressure at any point in the plane

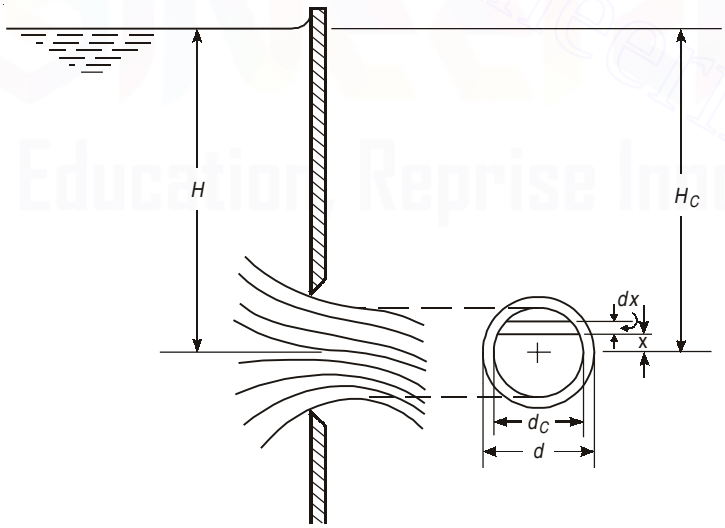


Figure 9.7 *Large circular orifice*

of the vena-contracta is atmospheric. Consider an elementary horizontal strip of depth dx out of the area of the vena-contracta at a distance x from the centre of the vena-contracta as shown in Fig. 9.7. The

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

area of the strip is $\left\{2\sqrt{(d_c/2)^2 x^2}\right\} \times dx$ and the velocity of flow through the strip is $C_v\sqrt{2g(H_c - x)}$ where C_v represents the coefficient of velocity for the orifice and H_c is the depth of the centre of the vena-contracta below the free surface of the liquid. The discharge through the elementary strip is

$$dQ = C_v\sqrt{2g(H_c - x)} \left\{2\sqrt{(d_c/2)^2 - x^2}\right\}dx$$

By making x to vary between the values $(-d_c/2)$ and $(+d_c/2)$, the integration of this expression will give the discharge from the entire orifice. For the integration it will be necessary to expand the term $(H_c - x)^{1/2}$ by the binomial theorem.

$$(H_c - x)^{1/2} = H_c^{1/2} - \frac{H_c^{-1/2}x}{2} - \frac{H_c^{-3/2}x^2}{8} - \frac{H_c^{-5/2}x^3}{16} - .....$$

$$\therefore \quad dQ = 2C_v\sqrt{2g}\left[H_c^{1/2}\left\{\left(\frac{d_c}{2}\right)^2 - x^2\right\}^{1/2} - \frac{\left\{\left(\frac{d_c}{2}\right)^2 - x^2\right\}^{1/2} x}{2H_c^{1/2}} - \frac{\left\{\left(\frac{d_c}{2}\right)^2 - x^2\right\}^{1/2} x^2}{8H_c^{3/2}} - \frac{\left\{\left(\frac{d_c}{2}\right)^2 - x^2\right\}^{1/2} x^3}{16H_c^{5/2}} ..... \right]dx$$

By integrating the above expression we obtain

$$Q = C_v\frac{\pi d_c^2}{4}\sqrt{2gH_c}\left[1 - \frac{d_c^2}{128H_c^2} - \frac{5d_c^4}{16384H_c^4} - .....\right]$$

The above derived expression for the discharge through a circular orifice involves the quantities such as d_c and H_c which are difficult to be determined. This difficulty may be overcome by introducing d and H respectively in place of d_c and H_c and a coefficient of contraction C_c. Then

$$Q = C_c C_v\frac{\pi d^2}{4}\sqrt{2gH}\left[1 - \frac{d^2}{128H^2} - \frac{5d^4}{16384H^4} - .....\right]$$

or

$$Q = C_d\frac{\pi d^2}{4}\sqrt{2gH}\left[1 - \frac{d^2}{128H^2} - \frac{5d^4}{16384H^4} - .....\right] \qquad ...(9.18)$$

where C_d is the coefficient of discharge for the orifice. This is an exact expression for the discharge through a large circular orifice. Since the quantity in the brackets in Eq. 9.18 has a value less than unity, and hence the actual discharge flowing through a large circular orifice is always less than that given by the expression

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = C_d \frac{\pi d^2}{4}\sqrt{2gH} \qquad ...(9.19)$$

which represents the discharge through the same orifice if it is treated as a small orifice. Now if $H = d$ then from Eq. 9.18

$$Q = 0.992 \left[C_d \frac{\pi d^2}{4}\sqrt{2gH} \right] \qquad ...(9.20)$$

which shows that for the head above the centre of the orifice equal to its diameter there is an error of less than even 1% in the discharge computed by the exact expression of Eq. 9.18 and the one computed by treating it as a small orifice and using Eq. 9.19. As such it may be stated that *as long as* $H > d$ i.e., *the head above the centre of the orifice is greater than its diameter, it may be treated as a small orifice.*

Exact information regarding the value of C_d for large orifices is not available but a value of 0.60 may be used for approximate computations for both rectangular as well as circular orifice when the width of the rectangular orifice or the diameter of the circular orifice is about 0.3 m or more.

9.6 FLOW UNDER PRESSURE THROUGH ORIFICES

In the above analysis for the flow through the small and the large orifices it has been assumed that the liquid in the reservoir or the tank containing the orifice is at atmospheric pressure so that the pressures on the liquid surface in the reservoir and on the issuing jet are same, being equal to the atmospheric pressure. However, there may be a case when the liquid contained in the reservoir may be under pressure and it is then discharged through an orifice freely in atmosphere. Obviously in this case the pressures on the liquid surface in the reservoir and on the emerging jet are not equal. But even in this case also the same equations as derived earlier may be applied with the modification that the head on the orifice is computed as the sum of the static head of the liquid on the orifice and the head of the liquid equivalent to the pressure (gage) intensity on the liquid surface in the reservoir.

9.7 FLOW THROUGH SUBMERGED (OR DROWNED) ORIFICE

1. Totally Submerged Orifice. If an orifice has its whole of the outlet side submerged under liquid so that it discharges a jet of liquid into the liquid of the same kind then it is known as totally submerged or totally drowned orifice. Figure 9.8 shows a totally submerged orifice for which let H_1 be the height of the liquid on the upstream side above the centre of the orifice and H_2 be the height of the liquid on the downstream side above the centre of the orifice. A vena-contracta will be formed in this case also and the pressure there corresponds to the head H_2. Now considering two points 1 and 2; point 1 being in the reservoir on the upstream side of the orifice and point 2 being at vena-contracta as shown in Fig. 9.8, and applying Bernoulli's equation between the points 1 and 2, we have, neglecting the loss of energy between these points,

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

But $$\frac{p_1}{w} = (H_1 + z_2 - z_1)$$

and if the velocity V_1 at point 1 is negligible then by substitution the velocity at vena-contracta is

$$V_2 = \sqrt{2g(H_1 - H_2)} \qquad ...(9.21)$$

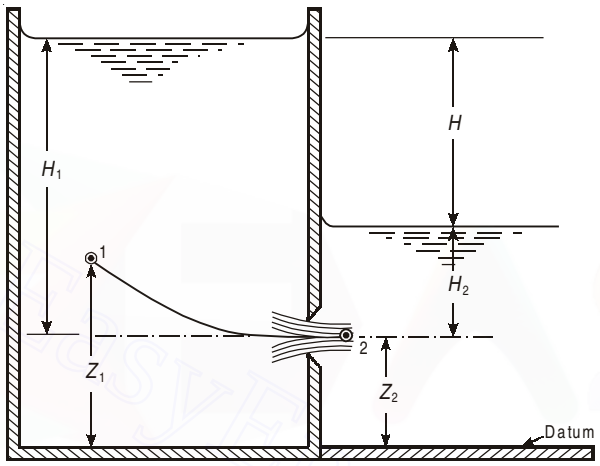


Figure 9.8 *Totally submerged orifice*

Equation 9.21 indicates that the Torricelli's formula is still applicable in this case provided that the head refers to the difference of head $(H_1 - H_2)$ on the two sides of the orifice. Since the loss of energy has been neglected, Eq. 9.21 gives only an ideal (or theoretical) velocity of flow through the orifice. Now if a is the cross-sectional area of the orifice, then the ideal (or theoretical) discharge through the orifice is $\left[a \times \sqrt{2g(H_1 - H_2)}\right]$ and introducing the coefficient of discharge C_d, the actual discharge through a totally submerged orifice is given by

$$Q = C_d\, a \sqrt{2g(H_1 - H_2)} \qquad ...(9.22)$$

In a totally submerged orifice the issuing jet is interfered with the liquid present on the outlet side of the orifice. This results in slightly reducing the coefficient of discharge. As such the coefficient of discharge for a totally submerged orifice is slightly less than that for an orifice discharging free.

2. Partially Submerged Orifice. If the outlet side of an orifice is only partly submerged under liquid then it is known as partially submerged or partially drowned orifice. As such in a partially submerged orifice its upper portion behaves as an orifice discharging free, while the lower portion behaves as a submerged orifice. Obviously, only a large orifice having its vertical dimension sufficiently large can behave as a partially submerged orifice. The discharge through a partially submerged orifice may be determined by computing separately the discharge through the free and the submerged portions and then adding together the two discharges thus computed.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Figure 9.9 shows a partially submerged orifice. Let H_1 be the height of the liquid on the upstream side above the bottom edge of the orifice; H_2 be the height of the liquid on the upstream side above the

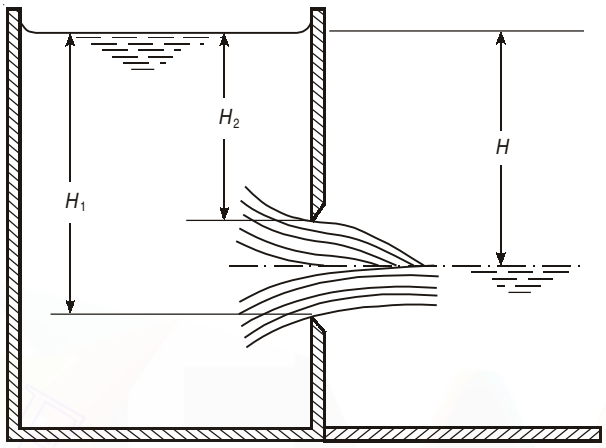


Figure 9.9 *Partially submerged orifice*

top edge of the orifice; and H be the difference between the liquid surfaces on the upstream and the downstream sides of the orifice. Now if Q_1 and Q_2 are respectively the discharges through the free and the submerged portions, then the total discharge Q through the orifice is

$$Q = Q_1 + Q_2 \qquad ...(9.23)$$

Again if the breadth of the orifice is b then from Eq. 9.15

$$Q_1 = \frac{2}{3} C_{d_1} b \sqrt{2g} \left\{H^{3/2} - H_2^{3/2}\right\}$$

where C_{d_1} represents the coefficient of discharge for the free portion of the orifice. Similarly for the submerged portion, area a is equal to $\{b \times (H_1 - H)\}$ and from Eq. 9.22

$$Q_2 = C_{d_2} b (H_1 - H) \sqrt{2gH}$$

where C_{d_2} represents the coefficient of discharge for the submerged portion of the orifice. Then

$$Q = \frac{2}{3} C_{d_1} b \sqrt{2g} \left\{H^{3/2} - H_2^{3/2}\right\} + C_{d_2} b (H_1 - H) \sqrt{2gH} \qquad ...(9.24)$$

9.8 ENERGY OR HEAD LOSSES OF FLOWING LIQUID DUE TO SUDDEN CHANGE IN VELOCITY

When the velocity of the flowing liquid changes, either in magnitude or direction, there is a large-scale turbulence generated due to the formation of the eddies in which a portion of the energy possessed by the flowing liquid is utilized which is ultimately dissipated as heat, and hence it is to be considered as lost. The change in the magnitude of the velocity of flow of liquid is due to the change in the cross-sectional area of the flow passage, and the change in the direction of the velocity of flow of liquid is due to the change in the direction of the flow passage. The change in the velocity of flow of liquid may be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

either gradual or sudden and in both the cases the energy is lost. But as compared with the gradual change of velocity, the loss of energy is much more when the sudden change of velocity takes place. Since the losses of energy have been found to vary as the square of the mean velocity of flow, they are frequently expressed in terms of the velocity head of the flowing liquid. Some of the losses of energy which may be caused due to the change of velocity are as indicated below:

(a) Loss of energy due to sudden enlargement.
(b) Loss of energy due to sudden contraction.
(c) Loss of energy at the entrance to a pipe from the large vessel.
(d) Loss of energy at the exit from a pipe.
(e) Loss of energy due to an obstruction in the flow passage.
(f) Loss of energy due to gradual contraction or enlargement.
(g) Loss of energy in bends.
(h) Loss of energy in various pipe fittings.

All the above noted losses of energy are termed as 'minor' losses, because the magnitude of these losses is generally quite small as compared with the loss of energy due to friction in long pipes which is distinguished as 'major' loss. Although the turbulence produced due to the change of velocity may persist for a considerable distance downstream and on account of which the flow after the sudden change of velocity is rendered extremely complicated, yet in general all the above noted minor losses are confined to a very short length of the passage of the flowing liquid. Therefore, it is possible to derive analytically the expressions representing the approximate losses of energy in some of the above cases as indicated below. However, the actual losses of energy in all the cases may be determined only experimentally.

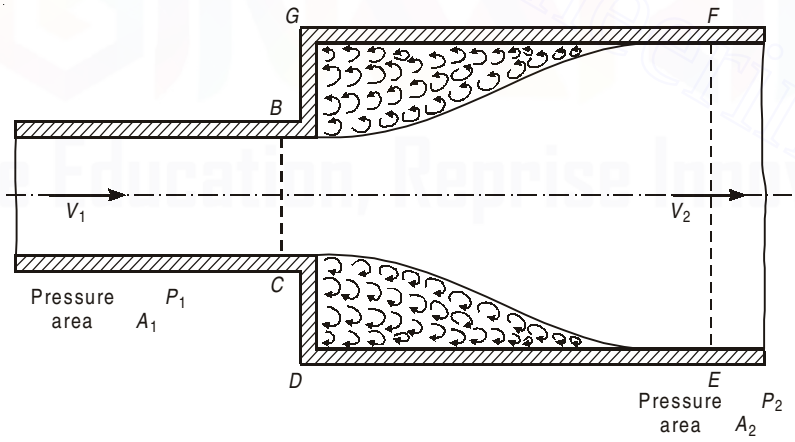


Figure 9.10 *Flow through a sudden expansion in a pipe*

(*a*) *Loss of energy due to sudden enlargement.* The loss of energy that occurs when a pipe of certain diameter suddenly expands to a large diameter, can be determined by applying the impulse momentum equation in addition to the Bernoulli's equation and the continuity equation. As shown in Fig. 9.10, consider a pipe of cross-sectional area A_1 and carrying a liquid of specific weight w, connected to

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

another pipe of larger cross-sectional area A_2, at a certain section. Since there is sudden change in the cross-sectional area of the flow passage, the liquid emerging from the smaller pipe is not able to follow the abrupt change of the boundary. Consequently at this section the flow separates from the boundary, forming regions of separation in which turbulent eddies are formed which result in the loss of energy which is ultimately dissipated as heat. The region of separation however extends upto some distance on the downstream where the flowing liquid again follows the boundary of the pipe of larger cross-sectional area.

Consider a control volume *BCDEFG*, as shown in Fig. 9.10, which includes the non-uniform region of separation and its end sections *BC* and *EF* are located in the narrower and the wider pipes respectively, where the velocities may be assumed to be practically uniform. Let V_1 and V_2 be the velocities of flow of liquid in the narrower and the wider pipes respectively. From continuity, since $Q = A_1V_1 = A_2V_2$, the velocity V_2 is smaller than V_1 and hence a change of momentum takes place as the liquid flows from the narrower pipe to the wider pipe. This change of momentum per second will be equal to the net force acting in the direction of flow on the liquid in the control volume *BCDEFG*. Now if p_1 and p_2 are the respective pressure intensities at the end sections *BC* and *FE* of the control volume and p' is the mean pressure of the eddying fluid over the annular face *GD* joining the two pipes, then the force acting on the liquid in the control volume in the direction of flow is

$$[p_1A_1 + p'(A_2 - A_1) - p_2A_2]$$

It is experimentally found that the pressure p' is equal to pressure p_1. The net force is thus equal to $(p_1 - p_2)A_2$. The momentum of liquid passing through section *BC* is equal to $\left(\frac{wQV_1}{g}\right)$ and that passing through section *FE* is equal to $\left(\frac{wQV_2}{g}\right)$ and hence the change of momentum per second is equal to $\left[\frac{wQ(V_2 - V_1)}{g}\right]$. Then from the impulse momentum equation

$$(p_1 - p_2)A_2 = \frac{wQ}{g}(V_2 - V_1)$$

or
$$\frac{p_1 - p_2}{w} = \frac{1}{g}\frac{Q}{A_2}(V_2 - V_1) = \frac{1}{g}V_2(V_2 - V_1) \quad ...(i)$$

Now if h_L is the loss of head between the sections *BC* and *EF* due to the sudden enlargement, then applying the Bernoulli's equation between the sections *BC* and *FE*, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_L$$

For the sake of convenience the pipe has been assumed to be horizontal, so that $z_1 = z_2$ and hence

$$h_L = \left(\frac{p_1}{w} - \frac{p_2}{w}\right) + \frac{V_1^2}{2g} - \frac{V_2^2}{2g} \quad ...(ii)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substituting for $\frac{(p_1 - p_2)}{w}$ from Eq. (i) in Eq. (ii) we get

$$h_L = \frac{1}{g} V_2 (V_2 - V_1) + \frac{V_1^2}{2g} - \frac{V_2^2}{2g}$$

or
$$h_L = \frac{(V_1 - V_2)^2}{2g} \quad ...(9.25)$$

Equation 9.25 gives the loss of head due to sudden enlargement. Since by continuity, $A_1 V_1 = A_2 V_2$, Eq. 9.25 may also be alternatively expressed as

$$h_L = \frac{V_1^2}{2g}\left(1 - \frac{A_1}{A_2}\right)^2 = \frac{V_2^2}{2g}\left(\frac{A_2}{A_1} - 1\right)^2 \quad ...(9.26)$$

The expression for the loss of head due to sudden enlargement was first obtained by J.C. Borda (1753–99) and L. Carnot (1738–1823) and is therefore sometimes known as the *Borda-Carnot equation* for head loss.

(*b*) *Loss of energy due to sudden contraction*. Consider a pipe carrying some liquid of specific weight w, whose cross-sectional area at a certain section reduces abruptly from A_1 to A_2 as shown in Fig. 9.11. Geometrically a sudden contraction is reverse of the sudden enlargement but it is not possible to apply the impulse momentum equation to a control volume between sections 1–1 and 2–2 in the case of sudden contraction. This is because just upstream of the junction between the two pipes, in the wider pipe the streamlines are curved and the liquid is accelerated, due to which the pressure at the annular face varies in an unknown manner which cannot be determined easily. Moreover, in the region just upstream of the junction there being converging or accelerating flow no major loss of energy occurs. However, immediately downstream of the junction as the liquid flows from the wider pipe to the narrower pipe a vena-contracta is formed, after which the stream of liquid widens again to fill completely the narrower pipe. In between the vena-contracta and the wall of the pipe a lot of eddies are formed which cause a considerable dissipation of energy. It is due to the formation of these eddies and the consequent dissipation of energy that the most of the loss of the energy is caused in the case of sudden contraction. Between the vena-contracta and section 2–2 at a certain distance away from it where the velocity has again become almost uniform, the flow pattern is similar to that after a sudden enlargement. As such Eq. 9.25 may be applied between the sections at vena-contracta and 2–2 to obtain an approximate value of the loss of head due to sudden contraction. Thus if A_c is the cross-sectional area of the stream at vena-contracta, and V_c and V_2 are the velocities of the flow of liquid at vena-contracta and in the narrower pipe at section 2–2 respectively, then applying Eq. 9.25 the loss of head due to sudden contraction is obtained as

$$h_L = \frac{(V_c - V_2)^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

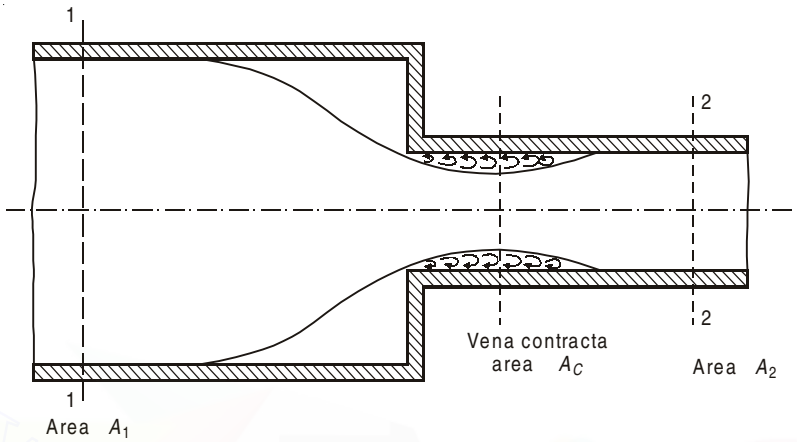


Figure 9.11 *Flow through a sudden contraction in a pipe*

Again by continuity, $A_c V_c = A_2 V_2$ and hence by substitution

$$h_L = \frac{V_2^2}{2g}\left(\frac{A_2}{A_c} - 1\right)^2 \qquad ...(9.27)$$

Further since the ratio of the area of the jet at a vena-contracta and the area of the opening is defined as a coefficient of contraction C_c. Thus in this case

$$C_c = \left(\frac{A_c}{A_2}\right)$$

and hence Eq. 9.27 becomes

$$h_L = \frac{V_2^2}{2g}\left(\frac{1}{C_c} - 1\right)^2 \qquad ...(9.28)$$

Often the loss of head due to sudden contraction is expressed as

$$h_L = k\frac{V_2^2}{2g} \qquad ...(9.29)$$

The value of C_c or k is however not constant but depends on the ratio $\left(\frac{A_2}{A_1}\right)$. Table 9.1 gives some of the values of the coefficient k for the different values of $\left(\frac{D_2}{D_1}\right)$, the ratio of the diameters of the narrower and the wider pipes, which may be adopted for the computation of the loss of head due to sudden contraction.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

TABLE 9.1

$\frac{D_2}{D_1}$	0	0.2	0.4	0.6	0.8	1.0
k	0.5	0.45	0.38	0.28	0.14	0

If C_c is assumed to be equal to 0.62 then by substituting in Eq. 9.28

$$h_L = \frac{V_2^2}{2g}\left(\frac{1}{0.62}-1\right)^2 = 0.375\ \frac{V_2^2}{2g}$$

i.e., $k = 0.375.$

However, in general the loss of head due to sudden contraction is taken to be equal to $0.5\ \frac{V_2^2}{2g}$ i.e., the value of k is adopted as 0.5.

(*c*) *Loss of energy at the entrance to a pipe from the large vessel.* When a liquid enters a pipe from a large vessel (or tank or reservoir) some loss of energy occurs at the entrance to the pipe which is sometimes known as inlet loss of energy. The flow pattern at the entrance to the pipe is similar to that in the case of a sudden contraction and hence the loss of energy at the entrance to the pipe is assumed to be the same as in the case of sudden contraction. The loss of energy actually depends on the form of the entrance. Thus for a rounded or bell-mouthed entrance the loss of energy is relatively much less as compared with a sharp-cornered entrance. In general, for a sharp-cornered entrance the loss of head at the entrance is taken to be equal to $0.5\ \frac{V^2}{2g}$, where V is the mean velocity of flow of liquid in the pipe.

(*d*) *Loss of energy at the exit from a pipe.* The outlet end of a pipe carrying a liquid may be either left free so that the liquid is discharged freely in the atmosphere or it may be connected to a large reservoir so that the pipe outlet becomes submerged and the liquid is discharged into a large body of static liquid. The liquid leaving the pipe at its outlet end still-possesses a kinetic energy corresponding to the velocity of flow of liquid in the pipe, which is ultimately dissipated either in the form of a free jet or it is lost in turbulence in the reservoir, depending on the condition of the outlet. Therefore, the loss of head at the exit from a pipe is equal to $\frac{V^2}{2g}$, where V is the mean velocity of flow in the pipe.

When the outlet end of a pipe is connected to a large reservoir, the flow pattern is similar to that of the sudden enlargement and hence the loss of head may also be determined by using Eq. 9.26 with the condition that in this case $A_2 \to \infty$. The loss of head at the exit of the pipe is then equal to $\frac{V^2}{2g}$, where V is the velocity of flow of liquid in the pipe.

(*e*) *Loss of energy due to an obstruction in the flow passage.* The loss of energy due to an obstruction in a pipe takes place on account of the reduction in the cross-sectional area of the pipe by the presence of the obstruction which is followed by an abrupt enlargement of the stream beyond the obstruction. As such the loss of head due to an obstruction may be computed by applying Eq. 9.25.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider a pipe of cross-sectional area A, and let an obstruction of maximum cross-sectional area a be placed in the pipe, as shown in Fig. 9.12. Thus at section 1–1 the area of the flow passage is reduced to $(A-a)$. As the liquid flows through the pipe a vena-contracta is formed just beyond section 1–1, after which the stream of liquid widens again. Between the vena-contracta and section 2–2 at a certain distance away from the obstruction on the downstream where the velocity has again become almost uniform, the flow pattern is similar to that after a sudden enlargement. Thus if V_c and V are respectively the velocities at vena-contracta and section 2–2, then applying Eq. 9.25 the loss of head due to the obstruction is obtained as

$$h_L = \frac{(V_c - V)^2}{2g}$$

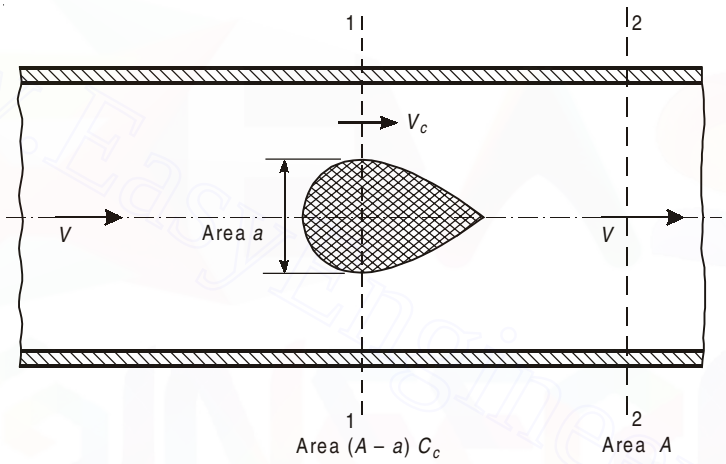


Figure 9.12 *Flow through a pipe with an obstruction*

Further, if C_c is the coefficient of contraction then the area of the stream at vena-contracta is $C_c(A-a)$ and by continuity equation

$$C_c(A-a)\,V_c = AV\,;\ \text{or}\ V_c = \frac{A}{C_c(A-a)}\,V$$

By substituting V_c, we have

$$h_L = \left[\frac{A}{C_c(A-a)} - 1\right]^2 \frac{V^2}{2g} \qquad \ldots(9.30)$$

Equation 9.30 thus represents the loss of head due to an obstruction in the flow passage. The value of C_c is however not constant but it depends on the type of obstruction. Therefore it has to be suitably assumed. For general purposes it may be assumed to vary between 0.60 and 0.66.

(*f*) *Loss of energy due to gradual contraction or enlargement.* The loss of energy can be considerably, reduced if in place of a sudden contraction or a sudden enlargement a gradual contraction or gradual enlargement is provided. This is so because in a gradual contraction or a gradual enlargement the velocity of the fluid is gradually increased or reduced, and thus, as far as possible, the formation of the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

eddies responsible for the dissipation of energy is eliminated. Generally the loss of head in a gradual contraction or a gradual enlargement may be expressed as

$$h_L = k\left(\frac{V_1 - V_2}{2g}\right)^2 \quad ...(9.31)$$

where k is a coefficient and V_1 and V_2 are the mean velocities at the inlet and the outlet. In general, the value of k depends on the angle of convergence or divergence and on the ratio of the upstream and the downstream cross-sectional areas. However, for a gradual contraction the value of k is usually very small even for larger values of the angle of convergence. Therefore, for a gradual contraction without sharp corners the loss of energy caused is so small that it may usually be neglected. On the other hand, for a gradual enlargement the value of k considerably depends on the angle of divergence. The value of k increases as the angle of divergence increases for a given ratio of the cross-sectional areas at the inlet and the outlet. Moreover in the case of a gradual enlargement except for very small angles of divergence, the flow of fluid is always subjet to separation from the boundaries, and the consequent formation of the eddies result in the loss of energy. Therefore in the case of gradual enlargement the loss of energy cannot be completely eliminated.

(*g*) *Loss of energy in bends.* The bends are provided in a flow passage to change the direction of flow. The change in the direction of flow also results in causing loss of energy. The loss of energy in the bends is due to the separation of flow from the boundary and the consequent formation of the eddies resulting in the dissipation of energy in turbulence. In general the loss of head in bends provided in pipes may be expressed as

$$h_L = k\frac{V^2}{2g} \quad ...(9.32)$$

where k is a coefficient and V is the mean velocity of flow of fluid. The value of k however depends on the total angle of the bend and on the relative radius of curvature R/d, where R is the radius of curvature of the pipe axis and d is the diameter of the pipe.

(*h*) *Loss of energy in various pipe fittings*. All pipe fittings such as valves, couplings etc., cause a loss of energy. The loss of head in the various pipe fittings may also be represented as

$$h_L = k\frac{V^2}{2g} \quad ...(9.33)$$

where k is a coefficient and V is the mean velocity of flow in the pipe. The value of the coefficient k actually depends on the type of the pipe fitting.

9.9 FLOW THROUGH AN EXTERNAL CYLINDRICAL MOUTHPIECE

A mouthpiece is a short tube fitted to a circular orifice provided in a tank or a reservoir. As indicated below by fitting a mouthpiece the discharge through an orifice may be increased. The increase in the discharge would, however, result only if the mouthpiece is running full so that the jet of liquid emerging from the mouthpiece is of the same diameter as that of the mouthpiece. A mouthpiece will be running full if its length is equal to about two to three times its diameter and also the head on the mouthpiece does not exceed certain value as indicated below. However, if the length of the mouthpiece is less and nearly equal to its diameter, then even at low heads it will be running free in which case the jet of liquid

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

will emerge from the mouthpiece without touching its sides and it will behave as an orifice only. Since the external cylindrical mouthpieces generally run full, only this condition of discharge has been considered for these mouthpieces.

Consider a tank having a cylindrical mouthpiece of cross-sectional area a, attached externally to one of its sides, as shown in Fig. 9.13, and filled with a liquid of specific weight w upto a constant height H above the centre of the mouthpiece. The liquid will emerge through the mouthpiece in the form of a free jet. Since the entrance section of the mouthpiece is exactly same as a sharp-edged orifice, the jet of liquid entering the mouthpiece contracts to form a vena-contracta at a section cc only a short distance from the entrance-section. Beyond the section cc the jet again expands to fill the mouthpiece completely, so that the jet of liquid that emerges from the mouthpiece has its cross-sectional area equal to that of the mouthpiece.

Now if a_c is the cross-sectional area of the jet at vena-contracta and C_c is the coefficient of contraction for the orifice (at the entrance to the mouthpiece) then

$$a_c = C_c a$$

Assuming a value of the coefficient of contraction C_c to be 0.62 then

$$a_c = 0.62a$$

Further, if V_c is the velocity of the jet at vena-contracta and V is the velocity of the jet at the outlet of the mouthpiece then by continuity

$$a_c V_c = aV$$

or

$$V_c = \frac{a}{a_c} V = \frac{V}{0.62}$$

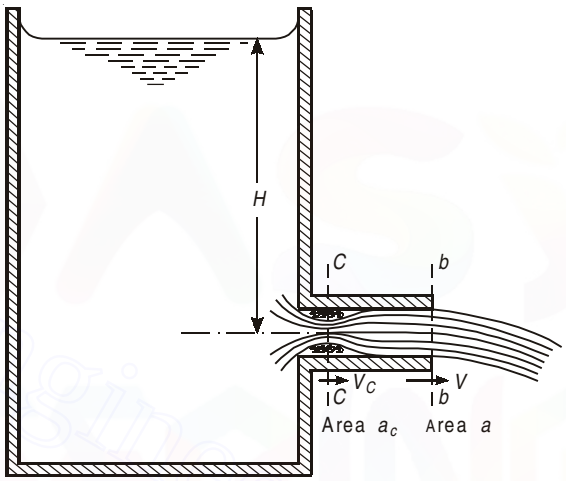


Figure 9.13 *Flow through an external cylindrical mouthpiece*

If H_a represents the atmospheric pressure head then applying Bernoulli's equation between the free surface of the liquid in the tank and a section bb just at the outlet of the mouthpiece, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

where h_L represents the loss of head between the sections cc and bb. Between the sections cc and bb the flow pattern is similar to that after a sudden enlargement and hence the loss of head h_L between these sections is that due to sudden enlargement which may be computed by Eq. 9.25. Thus

$$h_L = \frac{(V_c - V)^2}{2g} = \frac{\left(\frac{V}{0.62} - V\right)^2}{2g} = 0.375 \frac{V^2}{2g}$$

By substituting the value of h_L, we have

$$H_a + H = H_a + \frac{V^2}{2g} + 0.375 \frac{V^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$H = 1.375 \frac{V^2}{2g}$$

$$\therefore \quad V = \sqrt{\frac{2gH}{1.375}} = 0.855\sqrt{2gH}$$

Again the theoretical velocity of flow of liquid through the mouthpiece is

$$V_{th} = \sqrt{2gH}$$

Therefore the coefficient of velocity for the mouthpiece is

$$C_v = \frac{V}{V_{th}} = \frac{0.855\sqrt{2gH}}{\sqrt{2gH}} = 0.855$$

Further the coefficient of contraction C_c for the mouthpiece is equal to 1, because the cross-sectional area of the jet emerging from the mouthpiece is equal to the cross-sectional area of the mouthpiece. Therefore the coefficient of discharge for the mouthpiece is

$$C_d = C_c \times C_v = 1 \times 0.855 = 0.855$$

However, in actual practice the frictional resistance reduces the value of the coefficient of velocity for the mouthpiece and the actual value of C_v for an external cylindrical mouthpiece is approximately 0.82. Accordingly the actual value of C_d for a mouthpiece is approximately 0.82. The discharge through the mouthpiece is

$$Q = C_d\, a\sqrt{2gH} = 0.82\, a\sqrt{2gH}$$

Since the coefficient of discharge for an external cylindrical mouthpiece is more than that for a standard orifice, the discharge through such a mouthpiece is greater than that through a standard orifice of the same diameter under the same head.

In order to determine the pressure at the vena-contracta apply Bernoulli's equation between the free surface of the liquid in the tank and the section *cc*. Thus if H_c represents the absolute pressure head at vena-contracta, then

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

Since the flow of liquid from the tank upto section *cc* is a converging type of flow, the loss of energy may be neglected. Then

$$H_c = H_a + H - \frac{V_c^2}{2g}$$

But $V_c = \frac{V}{0.62}$ and $H = 1.375 \frac{V^2}{2g}$, so by substitution, we get

$$H_c = H_a + 1.375 \frac{V^2}{2g} - \frac{1}{2g}\left(\frac{V}{0.62}\right)^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$H_c = H_a - 1.225 \frac{V^2}{2g} \quad ...(9.34)$$

or $$H_c = H_a - \frac{1.225}{1.375} H$$

or $$H_c = H_a - 0.89\, H \quad ...(9.35)$$

Equation 9.35 is however only theoretical because as stated earlier in actual practice the value of C_v is 0.82 which is slightly less than the theoretical value of 0.855. Thus considering the actual value of C_v

$$V = 0.82\sqrt{2gH}$$

or $$\frac{V^2}{2g} = (0.82)^2 H$$

By substituting in Eq. 9.34

$$H_c = H_a - 1.225\,(0.82)^2 H$$

or $$H_c = H_a - 0.82\, H \quad ...(9.36)$$

From Eqs 9.34 and 9.36, it is observed that the pressure head at vena-contracta is less than atmospheric pressure by $1.225 \frac{V^2}{2g}$ or $0.82\,H$. In other words, the pressure at vena-contracta is vacuum or suction pressure. In fact, it is this reduction in the pressure at the vena-contracta in the case of a mouthpiece that results in increasing the effective head causing the flow, thereby increasing the discharge through the mouthpiece.

The minimum possible value upto which the pressure at vena-contracta may be reduced is the absolute zero pressure. Then from Eq. 9.36, the maximum or the limiting value of the head corresponding to which this minimum pressure would occur may be obtained as

$$H_c = 0 = H_a - 0.82\, H$$

or $$H = \frac{H_a}{0.82}$$

If the discharging liquid is water, then $H_a = 10.3$ m of water

and $$H = \frac{10.3}{0.82} = 12.56 \text{ m}$$

Thus, if the head H is equal to 12.56 m of water then the pressure at vena-contracta is reduced to absolute zero. However in actual practice the pressure at vena-contracta cannot be reduced below the vapour pressure of the flowing liquid. As such in actual practice when the head H becomes equal to only about 12.2 m of water the pressure at vena-contracta becomes equal to the vapour pressure of the flowing water, which then starts vapourizing and due to the bubbles of the vapour so released, the flow becomes unsteady. A further increase in the head H causes the velocity of the central core of the emerging jet of liquid to become so great that it sweeps aside the annulus of eddying liquid that surrounds it. The jet then comes out clear of the wall of the mouthpiece, so that it behaves as if the mouthpiece has been removed altogether and the flow is taking place only through the orifice.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consequently the coefficient of discharge is also reduced to about 0.6 which is same as that for an orifice. Under such condition of discharge, the external cylindrical mouthpiece is said to be running free.

9.10 FLOW THROUGH A CONVERGENT DIVERGENT MOUTHPIECE

In the case of a cylindrical mouthpiece, on account of the formation of the vena-contracta and the subsequent enlargement of the jet there occurs a loss of energy, which results in reducing the coefficient of discharge of the mouthpiece. However, if the mouth-piece is made to conform to the shape of the jet upto vena-contracta then the loss of energy may be eliminated. Such a mouthpiece is known as *convergent mouthpiece.* Theoretically, the coefficient of discharge for a convergent mouthpiece is equal to unity, but in actual practice on account of frictional resistance some energy is lost due to which the coefficient of discharge for this mouthpiece is about 0.975, Fig. 9.14 (a).

The loss of energy owing to the sudden enlargement of a jet in a mouthpiece can however be eliminated by making the mouthpiece gradually diverging. Such a mouthpiece is known as *divergent mouthpiece*. The value of the coefficient of discharge for this mouth-piece however depends on the angle of divergence and the length of the mouthpiece, Fig. 9.14 (b)

Often a convergent-divergent mouthpiece is also used which is made convergent upto vena-contracta and then diverges as shown in Fig. 9.14 (c). In such a mouthpiece as the divergence increases the velocity at the section *cc* increases, which in turn causes an increase in the vacuum or the suction pressure at vena-contracta. Since the pressure at vena-contracta cannot be reduced below a certain value (which is theoretically equal to the absolute zero but practically equal to the vapour pressure of the flowing liquid), there is a limit to the amount of divergence for a steady flow to be maintained. Thus the value of the maximum divergence which may be provided for a convergent-divergent mouthpiece may be determined as indicated below.

Applying Bernoulli's equation between the free liquid surface in the reservoir and the sections *cc* and *bb* as shown in Fig. 9.14 (c) we have neglecting the losses,

$$H_a + H = H_c + \frac{V_c^2}{2g} = H_a + \frac{V^2}{2g}$$

where H_a is the atmospheric pressure head in terms of the flowing liquid, H is the height of the free liquid surface above the centre of the mouthpiece, H_c is the absolute pressure head at section *cc*, and V_c and V are the velocities of flow at the sections *cc* and *bb* respectively.

From the above equation

$$\frac{V^2}{2g} = H$$

and

$$\frac{V_c^2}{2g} = H + H_a - H_c$$

Now if a_c and a are the cross-sectional areas at the vena-contracta and the outlet end of the mouthpiece, then by continuity

$$a_c V_c = aV$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{a}{a_c} = \frac{V_c}{V} = \sqrt{\frac{H + H_a - H_c}{H}}$$

or $$\frac{a}{a_c} = \sqrt{1 + \frac{H_a - H_c}{H}} \qquad ...(9.37)$$

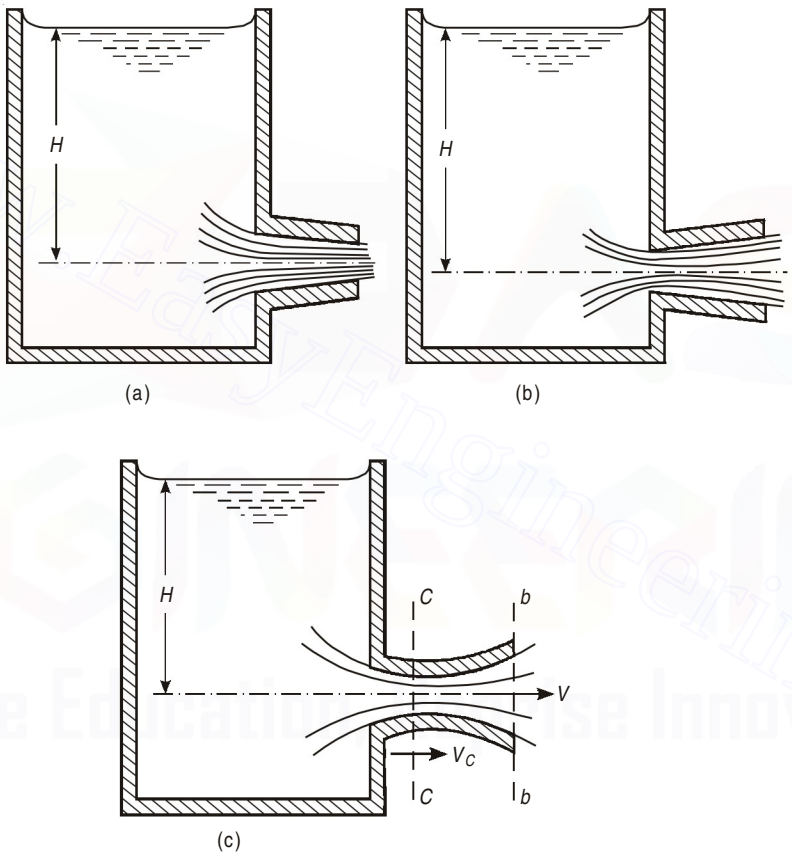


Figure 9.14 (*a*) *Convergent mouthpiece*, (*b*) *Divergent mouthpiece*, (*c*) *Convergent-Divergent mouthpiece*

If the flowing liquid is water then the limiting value of the suction pressure at vena-contracta $(H_a - H_c) = 7.8$ m, hence the maximum value of the ratio

$$\frac{a}{a_c} = \sqrt{1 + \frac{7.8}{H}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

9.11 FLOW THROUGH INTERNAL OR RE-ENTRANT OR BORDA'S MOUTHPIECE

An internal mouthpiece is a short cylindrical tube attached to a circular orifice in the side of a reservoir or tank such that it projects inwardly as shown in Fig. 9.15. Since in the case of internal mouthpiece both the conditions of discharge viz., running free and running full may be developed, the same have been discussed below.

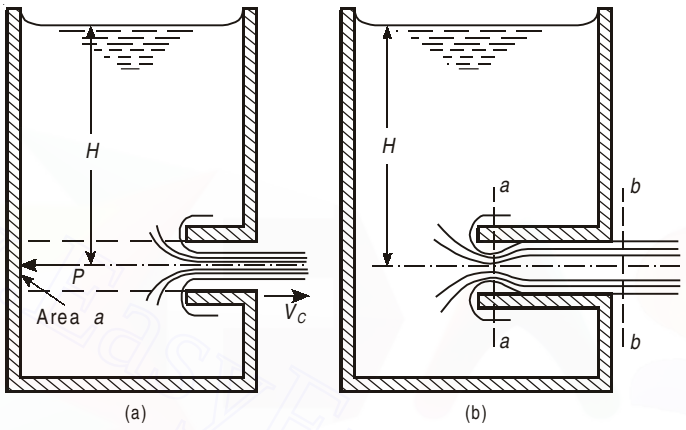


Figure 9.15 *Borda's mouthpiece (a) running free; (b) running full*

1. Borda's Mouthpiece Running Free. Consider a Borda's mouthpiece of cross-sectional area a, discharging liquid under a constant head H above the centre of the mouthpiece. As the jet of liquid enters the mouthpiece it contracts and vena-contracta is formed. Assuming that the length of the mouthpiece is small the jet cannot expand to fill the mouthpiece before it emerges from the mouthpiece, and hence it runs free as shown in Fig.9.15 (a). Let a_c be the cross-sectional area of the emerging jet and V_c be its velocity. The value of the coefficient of contraction C_c for such a mouthpiece can be analytically determined by the application of the impulse-momentum equation and the energy equation.

If we consider a circle of area a on the left hand wall of the tank, exactly opposite the orifice, so that if the mouthpiece were prolonged it would meet the wall in this circle, then the total pressure (or static thrust) on the area a is $P = waH$. But on the right hand wall there can be no counter balancing thrust because of the opening in the wall. Therefore it is the unbalanced reaction (waH) which may be considered to be acting at the entrance section of the mouthpiece and causing the liquid to flow through the mouthpiece. The force thus acting is equal to the rate of change of momentum of the flowing liquid. The mass of liquid flowing through the mouthpiece per second is $\left(\frac{wa_cV_c}{g}\right)$ and the change in the velocity of flow of liquid is from 0 in the tank to V_c as the liquid flows through the mouthpiece. Hence the rate of change of momentum imposed by the force (waH) is $\left(\frac{wa_cV_c}{g}\right)V_c$. Thus applying the impulse-momentum equation, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$waH = \left(\frac{wa_c V_c}{g}\right) V_c$$

$$\frac{a_c}{a} = \frac{gH}{V_c^2} \qquad ...(i)$$

Since the jet of liquid is not expanding and also the jet emerges clear of the walls of the mouthpiece, the loss of energy is altogether eliminated. As such applying the energy equation between the free surface of the liquid and the section *aa* of the jet just outside the mouthpiece, we have

$$H_a + H = H_a + \frac{V_c^2}{2g}$$

or $$\frac{V_c^2}{2g} = H \qquad ...(ii)$$

By combining Eqs (i) and (ii), we get

$$C_c = \frac{a_c}{a} = \frac{1}{2} \quad \text{or} \quad a_c = 0.5a$$

That is the coefficient of contraction for a Borda's mouthpiece = 0.5.

However, if some loss of energy is considered and the flowing liquid is water, then

$$V_c = 0.98\sqrt{2gH} \quad ; \text{or } \frac{V_c^2}{2g} = 0.96\,H$$

and $$C_c = \frac{0.5}{0.96} = 0.52$$

This method cannot, however, be used for finding the value of C_c for a plain sharp-edged orifice. This is so because as the liquid approaches the orifice, the streamlines converge, thereby resulting in reducing the pressure below the full static head H in a zone immediately surrounding the orifice. As such the unbalanced force producing the change of momentum for the liquid flowing through the orifice is the static thrust waH, plus a reaction equivalent to the reduction in pressure in the zone surrounding the orifice, which is unknown and cannot be readily determined. On the other hand there is no such zone in the side of the tank containing the Borda's mouthpiece, since in this case still liquid is in contact with the tank walls, and pressure changes take place well towards the middle of the tank.

The value of C_v for a Borda's mouthpiece running free is equal to 1 and therefore the coefficient of discharge C_d equals C_c, that is

$$C_d = 0.5 \text{ (or } 0.52)$$

2. Borda's Mouthpiece Running Full. Consider a Borda's mouthpiece of cross-sectional area a discharging liquid under a constant head H above the centre of the mouthpiece. As shown in Fig. 9.15 (b), the jet of liquid as it enters the mouthpiece contracts and vena-contracta is formed, but since the mouthpiece is long enough the jet of liquid again expands to fill the mouthpiece completely, so that the mouthpiece is running full and the jet of liquid emerging from the mouthpiece has the same diameter

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

as that of the mouthpiece. The flow pattern in this case is thus same as in the case of an external cylindrical mouthpiece. Again since the issuing jet is of the same cross-sectional area as that of the mouthpiece the coefficient of contraction C_c at the outlet of the mouthpiece is equal to 1. The value of the coefficient of discharge for a Borda's mouthpiece running full may be determined as indicated below.

If a_c is the cross-sectional area of the jet at vena-contracta and C_c is the coefficient of contraction at the entrance to the mouthpiece then $a_c = C_c a$. At the entrace section of the mouthpiece since the flow pattern is same as that for the running free condition and hence

$$C_c = 0.5$$

or $$a_c = 0.5\, a$$

Further if V_c is the velocity of the jet at vena-contracta and V is the velocity of the jet at the outlet of the mouthpiece, then by continuity

$$a_c V_c = aV$$

or $$V_c = \frac{a}{a_c} \quad V = \frac{V}{0.5}$$

Consider section *aa* at the vena-contracta and section *bb* just outside the mouthpiece. If H_a is the atmospheric pressure head, then applying Bernoulli's equation between the free surface of the liquid in the tank and the section *bb*, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

where h_L represents the loss of head between the sections *aa* and *bb*. Between the section *aa* and *bb* the flow pattern is similar to that after a sudden enlargement and hence the loss of head h_L may be computed by Eq. 9.25. Thus

$$h_L = \frac{(V_c - V)^2}{2g} = \frac{\left(\frac{V}{0.5} - V\right)^2}{2g} = \frac{V^2}{2g}$$

By substituting the value of h_L, we have

$$H_a + H = H_a + \frac{V^2}{2g} + \frac{V^2}{2g}$$

or $$\frac{V^2}{g} = H \,;\, V = \sqrt{gH}$$

But theoretical velocity

$$V_{th} = \sqrt{2gH}$$

∴ Coefficient of velocity

$$C_v = \frac{\sqrt{gH}}{\sqrt{2gH}} = \frac{1}{\sqrt{2}} = 0.707$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since C_c at the outlet of the mouthpiece running full is equal to 1, the coefficient of discharge for the Borda's mouthpiece running full is

$$C_d = C_c \times C_v = (1 \times 0.707)$$
$$= 0.707$$

However, in practice the coefficient of discharge for a Borda's mouthpiece running full is found to be slightly more than this value and it is about 0.75. This is probably on account of the fact that in actual practice the loss of energy is less than that due to sudden enlargement as considered above.

The pressure at vena-contracta may be found by applying Bernoulli's equation between the free surface of the liquid in the tank and the section *cc*. Thus if H_c represents the absolute pressure head at vena-contracta, then

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

or
$$H_c = H_a + H - \frac{V_c^2}{2g}$$

But $V_c = 2V$

and $V^2 = gH$, so by substitution

$$H_c = H_a - H \qquad ...(9.38)$$

From Eq. 9.38 it is observed that at vena-contracta formed in a Borda's mouthpiece running full, the pressure is less than the atmospheric pressure by an amout equal to the head H of the liquid above the centre of the mouthpiece.

If it is assumed that the flowing liquid is water and the limiting value of the absolute pressure at vena-contracta is 2.5 m of water, then the maximum value of H upto which the flow through the Borda's mouthpiece will remain steady may be obtained from Eq. 9.38 as

$$2.5 = 10.3 - H$$

or
$$H = 7.8 \text{ m}$$

A further increase in the head H would cause the jet to come out clear of the walls of the mouthpiece thereby resulting in running free condition of discharge for the mouthpiece.

9.12 FLOW THROUGH AN ORIFICE OR A MOUTHPIECE UNDER VARIABLE HEADS

In the preceding analysis of flow through orifice and mouth-pieces it has been assumed that the head above the centre of the orifice or the mouthpiece is constant with time, which can be achieved by adjusting the inflow of liquid into the tank equal to the outflow of liquid through the orifice or the mouthpiece. The flow through an orifice or a mouthpiece under such conditions is steady. However, if the head on an orifice or a mouthpiece is not constant with time, the flow becomes unsteady. The variation of head on an orifice or a mouthpiece may be either due to the inflow of the liquid being completely cut off or the inflow of the liquid being different from the outflow through the orifice (or mouthpiece). Under both these conditions of flow the main problem is to determine the time required for the free liquid surface to change from its initial position to some other position. Both these conditions of flow are discussed below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*a*) *Time of emptying a tank with no inflow.* Consider a tank containing liquid upto a height H_1, above the centre of the orifice (or mouthpiece) provided in the side or bottom of the tank. It is required to find the time t for the liquid surface to fall from the height H_1 to a height H_2 above the centre of the opening. Since the discharge through an orifice or a mouthpiece depends on the head, which is changing in this case with time, the discharge also changes with the head. Therefore the time t can be determined only by integration as indicated below.

Let at any instant the liquid surface be at a height h above the orifice (or mouthpiece) and let the liquid surface fall by a small amount dh in time dt. Now if A is the horizontal cross-sectional area of the tank then the volume of liquid leaving the tank in time dt is Adh. Again if at this instant the discharge through the orifice (or mouthpiece) is Q, then the volume of liquid discharged during the interval of time dt is Qdt. As the volume of the liquid leaving the tank is equal to the volume of the liquid flowing through the orifice (or mouthpiece) during the same interval of time, we have

$$A(-dh) = Qdt$$

The negative sign is introduced becuase as the time increases the head decreases. Further if a is the cross-sectional area of the orifice (or mouthpiece) and C_d is its coefficient of discharge, then

$$Q = C_d a \sqrt{2gh}$$

and by substitution

$$-Adh = C_d a \left(\sqrt{2gh}\right) dt$$

or
$$dt = -\frac{Adh}{C_d a\sqrt{2gh}}$$

By integrating both the sides of the above expression, we get

$$\int_0^t dt = -\int_{H_1}^{H_2} \frac{Adh}{C_d a\sqrt{2gh}}$$

or
$$t = \int_{H_1}^{H_2} -\frac{Adh}{C_d a\sqrt{2gh}} \quad ...(9.39)$$

Equation 9.39 may be evaluated if the shape of the tank is known. The tanks of the following shapes are commonly found in practice.

(i) Cylindrical (or rectangular or prismatic, with constant horizontal cross-sectional area).
(ii) Conical.
(iii) Hemispherical.

(i) Cylindrical Tank. As shown in Fig. 9.16 (a) a cylindrical tank placed with its axis vertical has a constant horizontal cross-sectional area A, then from Eq. 9.39

$$t = -\frac{A}{C_d a\sqrt{2g}} \int_{H_1}^{H_2} h^{-1/2}\, dh$$

or
$$t = \frac{2A}{C_d a\sqrt{2g}} (H_1^{1/2} - H_2^{1/2}) \quad ...(9.40)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If the tank is completely emptied then $H_2 = 0$ and Eq. 9.40 becomes

$$t = \frac{2A(H_1^{1/2})}{C_d a\sqrt{2g}} \qquad ...(9.41)$$

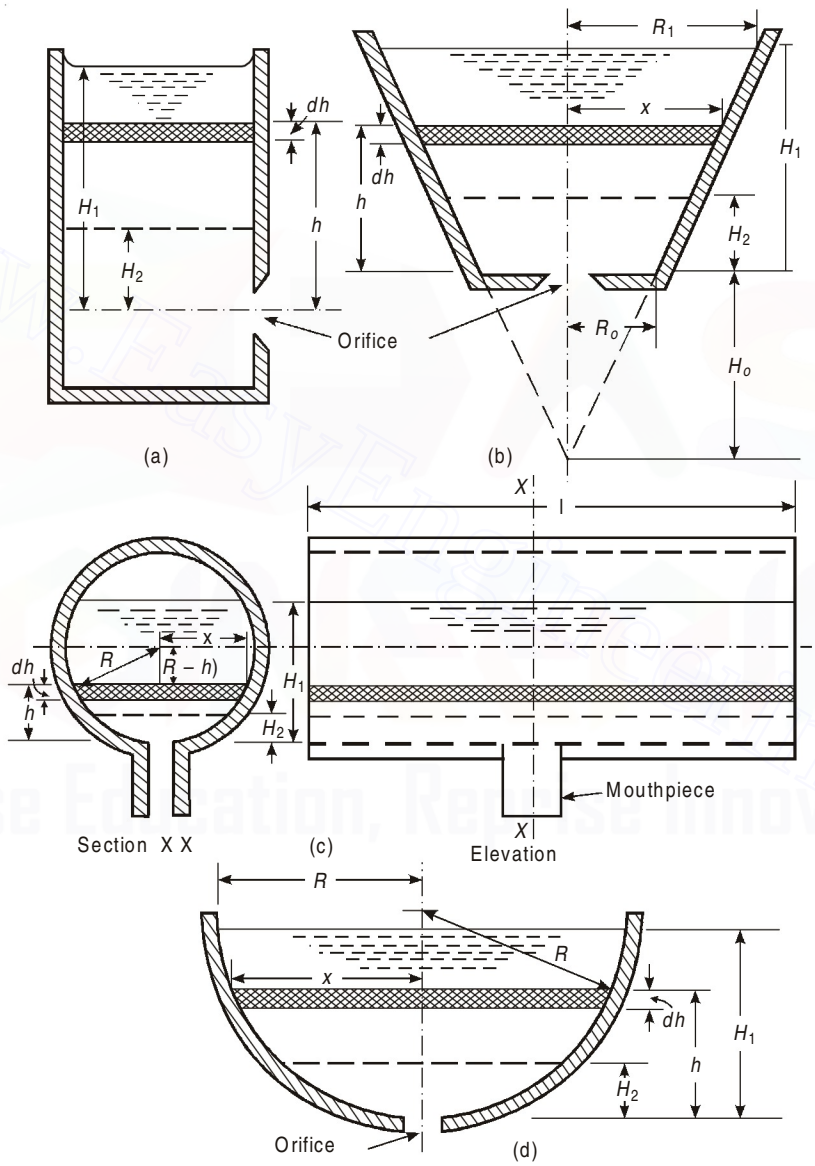


Figure 9.16 *Emptying of tanks (a) vertical cylindrical; (b) conical; (c) horizontal cylindrical; (d) hemispherical*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, if a cylindrical tank is placed with its axis horizontal then the horizontal cross-sectional area A of the tank will also vary with the head. As shown in Fig. 9.16 (c), in this case

$$A = 2l\sqrt{2Rh - h^2}$$

where l is the length and R is the radius of the tank. Then from Eq. 9.39

$$t = -\frac{2l}{C_d a\sqrt{2g}} \int_{H_1}^{H_2} \frac{\sqrt{2Rh - h^2}}{\sqrt{h}}\, dh$$

or
$$t = \frac{4l}{3C_d a\sqrt{2g}} \left[(2R - H_2)^{3/2} - (2R - H_1)^{3/2}\right] \quad ...(9.42)$$

(ii) Conical Tank. Generally a conical tank has a shape of a frustrum of a cone. In this case the horizontal cross-sectional area A varies. Thus as shown in Fig. 9.16 (b)

$$A = \pi x^2$$

where x is the radius of the cone at a height h above the bottom. From the similar triangles, we have

$$\frac{R_1}{(H_1 + H_0)} = \frac{x}{(h + H_0)}$$

or
$$x = \frac{R_1(h + H_0)}{(H_1 + H_0)}$$

Then from Eq. 9.39

$$t = -\frac{\pi R_1^2}{C_d a\sqrt{2g}(H_1 + H_0)^2} \int_{H_1}^{H_2} \frac{(H_0 + h)^2}{\sqrt{h}}\, dh$$

or
$$t = -\frac{\pi R_1^2}{C_d a\sqrt{2g}(H_1 + H_0)^2} \times \left[\frac{2}{5}h^{5/2} + 2H_0^2 h^{1/2} + \frac{4}{3}H_0 h^{3/2}\right]_{H_1}^{H_2} \quad ...(9.43)$$

In the above expression the value of H_0 may be obtained if the radius R_0 at the bottom of the vessel is known. Thus again by similar triangles

$$\frac{R_1}{H_1 + H_0} = \frac{R_0}{H_0}$$

or
$$H_0 = \frac{R_0 H_1}{(R_1 - R_0)}$$

(iii) Hemispherical Tank. In this case too the horizontal cross-sectional area is varying as shown in Fig. 9.16 (d)

$$A = \pi x^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $x = \sqrt{2Rh - h^2}$

where R is the radius of tank. Then from Eq. 9.39

$$t = -\frac{\pi}{C_d a\sqrt{2g}} \int_{H_1}^{H_2} \frac{(2Rh - h^2)}{\sqrt{h}}\, dh$$

or $$t = \frac{2\pi}{C_d a\sqrt{2g}} \left[\frac{2}{3}R\left(H_1^{3/2} - H_2^{3/2}\right) - \frac{1}{5}\left(H_1^{5/2} - H_2^{5/2}\right)\right] \quad ...(9.44)$$

Now if the tank (or vessel) was full at the beginning and it is completely emptied, then

$H_1 = R$

and $H_2 = 0$

Equation 9.44 then becomes

$$t = \frac{14\pi R^{5/2}}{15 C_d a\sqrt{2g}} \quad ...(9.45)$$

(*b*) *Time of emptying (or filling) a tank with inflow.* Consider a tank of constant horizontal cross-sectional area A, which is provided with an orifice (or mouthpiece) of cross-sectional area a. Let there be a constant inflow of liquid of Q m^3 per second and at the same time the liquid is discharging through the orifice (or mouthpiece). It is required to find the time t in which the height of the liquid surface changes from H_1 to H_2 above the centre of the orifice (or mouthpiece). As shown in Fig. 9.17 let at any instant liquid surface be at a height h above the centre of the orifice (or mouthpiece) and in time dt the level is increased by dh. Then volume of liquid added to the tank is Adh.

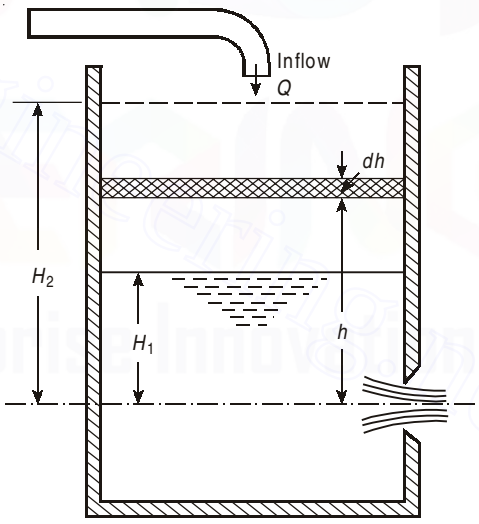


Figure 9.17 *Flow from a vessel with inflow*

Further in time dt the volume of the inflow of the liquid into the tank is Qdt, and during the same time the volume of the liquid discharged through the orifice (or mouthpiece) is qdt where q represents the discharge through the orifice (or mouthpiece) at that instant.

Since $q = C_d a\sqrt{2gh}$

$= K\sqrt{h}$

where $K = C_d a\sqrt{2g}$

$qdt = K\sqrt{h}\, dt$

Thus net volume of liquid added to the tank during time dt is

$$(Qdt - qdt) = \left(Q - K\sqrt{h}\right) dt$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus equating the two, we have

$$Adh = \left(Q - K\sqrt{h}\right) dt$$

or $$dt = \frac{Adh}{Q - K\sqrt{h}}$$

By integrating this equation the time required to raise the liquid surface from the height H_1 to H_2 may be obtained. Thus

$$\int_0^t dt = \int_{H_1}^{H_2} \frac{Adh}{Q - K\sqrt{h}}$$

or $$t = \int_{H_1}^{H_2} \frac{Adh}{Q - K\sqrt{h}} \quad ...(i)$$

Let $$Q - K\sqrt{h} = z$$

or $$h = \frac{(Q - z)^2}{K^2}$$

Differentiating with respect to z

$$dh = -\frac{2(Q - z)}{K^2} dz$$

Substituting this value of dh and h in Eq. (i), we have

$$t = -\frac{2A}{K^2} \int \left(\frac{Q - z}{z}\right) dz$$

or $$t = -\frac{2A}{K^2} [Q \log_e z - z]$$

or $$t = -\frac{2A}{K^2} \left[Q \log_e \left(Q - K\sqrt{h}\right) - \left(Q - K\sqrt{h}\right)\right]_{H_1}^{H_2}$$

or $$t = -\frac{2A}{K^2} \left[Q \log_e \left(\frac{Q - K\sqrt{H_2}}{Q - K\sqrt{H_1}}\right) + K(\sqrt{H_2} - \sqrt{H_1})\right] \quad ...(9.46)$$

Equation 9.46 can also be used to compute the time required to lower the liquid surface from the initial height H_1 to another height H_2, in which case Eq. 9.46 gives negative result.

9.13 FLOW OF LIQUID FROM ONE VESSEL TO ANOTHER

As shown in Fig. 9.18 consider a liquid flowing from one vessel to another, so that as the liquid surface falls in one vessel, it will rise by a corresponding amount in the other. In this case, the orifice will be drowned and therefore the head causing the flow at any instant will be the difference between the liquid surfaces in the two tanks at that instant. The problem in this case is to determine the time

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

required to reduce the difference between the liquid surfaces in the two tanks from H_1 to H_2.

Let the liquid flow from a vessel of cross-sectional area A_1 to a vessel of cross-sectional area A_2 and let a be the cross-sectional area of the orifice. Let the difference between the liquid surfaces in the two vessels be H_1 at the beginning and it is required to determine the time t, taken for the difference to be reduced to H_2.

At any instant let the difference between the liquid surfaces in the two vessels be h and let a small quantity Q flow through the orifice in time dt. This will cause the liquid surface to fall by dH in the upstream tank and the corresponding rise in the liquid surface in the downstream tanks will be $\frac{A_1}{A_2}$ dH.

$\therefore$ Difference of liquid levels after time dt

$$= h - \left(dH + dH \frac{A_1}{A_2} \right) = h - dH \left(1 + \frac{A_1}{A_2} \right)$$

Hence the change of liquid levels in time dt

$$= dH \left(1 + \frac{A_1}{A_2} \right)$$

It is this change of the liquid levels in time dt that equals the head which causes flow. Let this head be dh, then

$$dh = \left(1 + \frac{A_1}{A_2} \right) ; \text{ or } dH = \frac{dh}{\left(1 + \frac{A_1}{A_2} \right)}$$

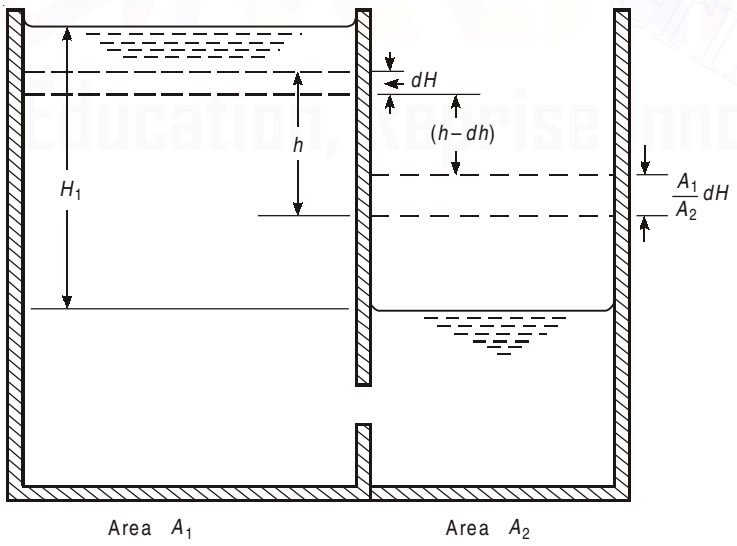


Figure 9.18 *Flow of liquid from one vessel to another*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again if Q is the discharge through the orifice at this instant then the total volume of liquid passing through the orifice in time dt is Qdt.

But $$Q = C_d a\sqrt{2gh}$$

$$\therefore \quad Qdt = C_d a\sqrt{2gh}\ dt$$

Further the quantity of liquid flowing through the orifice in time dt is also equal to the volume of liquid leaving the upstream tank which is equal to $A_1 dH$. Thus equating the two, we have

$$-A_1 dH = C_d a\sqrt{2gh}\ dt$$

The negative sign has been introduced because as the time dt increases the liquid surface in the upstream tank lowers down.

$$\therefore \quad dt = -\frac{A_1 dH}{C_d a\sqrt{2gh}}$$

Substituting the value of dH as derived above

$$dt = -\frac{A_1 dh}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2gh}}$$

By integrating both the sides of the above expression, we get

$$\int_0^t dt = -\int_{H_1}^{H_2} \frac{A_1 dh}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2gh}}$$

or $$t = \frac{2A_1(H_1^{1/2} - H_2^{1/2})}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2g}} \qquad ...(9.47)$$

If both the vessels have the same horizontal cross-sectional area i.e., $A_1 = A_2$, then

$$t = \frac{A_1(H_1^{1/2} - H_2^{1/2})}{C_d a\sqrt{2g}} \qquad ...(9.48)$$

It may however be noted that the time taken to reduce the difference of liquid level between two vessels of different cross-sectional areas is the same whether the liquid flows from a larger to smaller vessel or from a smaller to larger vessel, provided the reduction in liquid level is the same in each case.

9.14 TIME OF EMPTYING AND FILLING OF A CANAL LOCK

As discussed in Chapter 3 a canal lock is a rectangular chamber constructed at the junction of two canal reaches which are at different levels. It is constructed to facilitate the transfer of boats from a higher to a lower level and vice versa. The difference of level of the water surfaces in the two canal reaches is termed as the *lift* of the lock. The lock chamber is closed at each end by a pair of gates called

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

lock gates. When the lock is full it can be emptied by means of orifices (or sluices) placed in the lower gates below the water surface in the lower reach. When the lock is empty it can be filled by means of orifices (or sluices) placed in the upper gates. The orifices may be closed or opened as per requirements by means of sliding shutters. In designing locks, it is necessary to estimate the time required to empty or fill the lock which may be determined by using some of the equations derived earlier as indicated below.

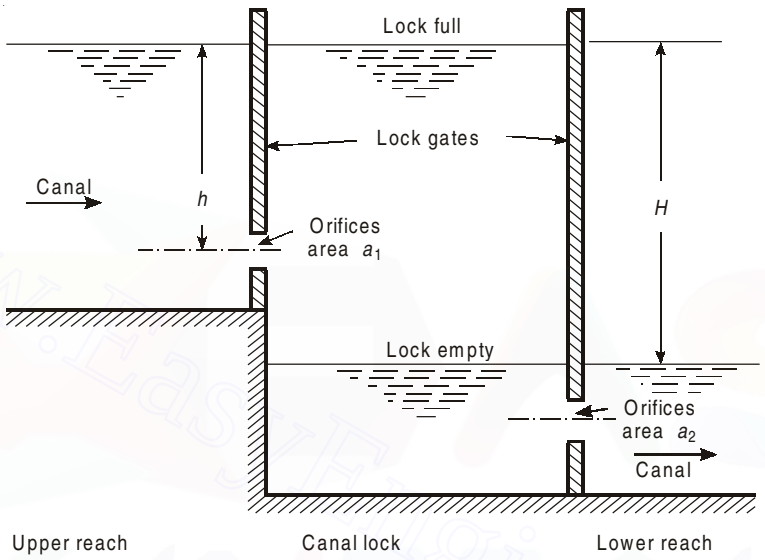


Figure 9.19 *Canal lock*

Let A be cross-sectional area of the water surface in the lock; H be the lift; and h be the depth from the water surface of the upper reach to the centre of the upper orifices. Further let a_1 and a_2 be the total areas of the upper and the lower orifices respectively.

(*i*) *To empty the lock*. The orifices being submerged, the head varies from H to 0. The time required to empty the lock may be obtained by Eq. 9.40 as

$$t = \frac{2A\sqrt{H}}{C_d a_2 \sqrt{2g}} = \frac{2AH}{C_d a_2 \sqrt{2gH}} \qquad ...(9.49)$$

(*ii*) *To fill the lock*. From the level of the lower reach to the centre of the upper orifices the head is constant, equal to h. Thus the time taken to fill the lock upto the centre of the upper orifices is

$$t_1 = \frac{A(H-h)}{C_d a_1 \sqrt{2gh}}$$

From the centre of the upper orifices to the level of the upper reach, the head varies from h to 0. Thus the time for filling this portion is obtained from Eq. 9.40 as

$$t_2 = \frac{2A\sqrt{h}}{C_d a_1 \sqrt{2g}} = \frac{2Ah}{C_d a_1 \sqrt{2gh}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence the total time t taken to fill the lock is

$$t = t_1 + t_2 = \frac{A(H+h)}{C_d a_1 \sqrt{2gh}} \quad \text{....(9.50)}$$

ILLUSTRATIVE EXAMPLES

Example 9.1. *A jet of water issues from a sharp edged vertical orifice under a constant head of 0.51 m. At a certain point of issuing jet, the horizontal and vertical coordinates measured from the vena-contracta are 0.406 m and 0.085 m respectively. Determine C_v. If $C_d = 0.62$, find C_c.*

Solution

From Eq. 9.10, we have

$$C_v = \sqrt{\frac{x^2}{4hy}}$$

$x = 0.406$ m; $y = 0.085$ m; and $h = 0.51$ m

$$\therefore \quad C_v = \sqrt{\frac{(0.406)^2}{4\times 0.51\times 0.085}} = 0.975$$

$$C_d = C_c \times C_v$$

$$\therefore \quad C_c = \frac{C_d}{C_v} = \frac{0.62}{0.975} = 0.636$$

Example 9.2. *Water discharges at the rate of 98 litres per second through a 0.12 m diameter vertical sharp-edged orifice placed under a constant head of 18 m. A point on the jet measured from the vena-contracta of the jet has coordinates 4.5 m horizontal and 0.54 m vertical. Find (a) the coefficients C_c, C_v, C_d and C_r for the orifice; and (b) the power lost at the orifice.*

Solution

(a) $$C_v = \sqrt{\frac{x^2}{4hy}}$$

$x = 4.5$; $h = 10$ m; and $y = 0.54$ m

$$\therefore \quad C_v = \sqrt{\frac{(4.5)^2}{4\times 10\times 0.54}} = 0.968$$

$$Q = 98\ l/\text{s} = 0.098\ \text{m}^3/\text{s}$$

Theoretical discharge

$$Q_{th} = a\sqrt{2gh}$$

$$= \frac{\pi}{4}(0.12)^2 \times \sqrt{2\times 9.81\times 10}$$

$$= 0.158\ \text{m}^3/\text{s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad C_d = \frac{0.098}{0.158} = 0.62$$

But $C_d = C_c \times C_v$

$$\therefore \quad C_c = \frac{C_d}{C_v} = \frac{0.62}{0.968} = 0.64$$

$$C_r = \left(\frac{1}{C_{v^2}} - 1\right) = \left[\frac{1}{(0.968)^2} - 1\right] = 0.067$$

(*b*) *SI Units*

$$\text{Loss of power} = wQH\,(1 - C_v^2)$$
$$= 9810 \times 0.098 \times 10\,[1 - (0.968)^2]$$
$$= 0.605 \text{ kW}$$

Metric Units

$$\text{Loss of power} = \frac{wQH(1 - C_V^2)}{75}$$
$$= \frac{1000 \times 0.098 \times 10[1 - (0.968)^2]}{75}$$
$$= 0.823 \text{ h.p.}$$

Example 9.3. *A closed tank partially filled with water discharges through an orifice of 12.5 mm diameter and has a coefficient of discharge of 0.65. If air is pumped into the upper part of the tank, determine the pressure required to produce a discharge or 36.6 litres/minute when the water surface is 1m above the outlet.*

Solution

$$Q = C_d\, a \sqrt{2gh}$$

$$Q = \frac{36.6 \times 10^{-3}}{60} = 6.1 \times 10^{-4} \text{ m}^3/\text{s}$$

$$a = \frac{\pi}{4} \times (12.5 \times 10^{-3}) = 1.23 \times 10^{-4} \text{ m}^2$$

Thus by substitution, we get

$$6.1 \times 10^{-4} = 0.65 \times 1.23 \times 10^{-4} \times \sqrt{2 \times 9.81 \times h}$$

$$\therefore \quad h = 2.97 \text{ m}$$

$\therefore$ Pressure head of the air to be pumped

$$= (2.97 - 1) = 1.97 \text{ m of water}$$

SI Units

Pressure intensity of air

$$= (9\,810 \times 1.97)$$
$$= 19\,326 \text{ N/m}^2 = 19.326 \text{ kN/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Metric Units

Pressure intensity of air

$$= (1000 \times 1.97)$$
$$= 1970 \text{ kg(f)/m}^2 = 0.197 \text{ kg(f)/cm}^2$$

Example 9.4. *A large tank having a circular orifice 6.45 ×10^{-4} m^2 in area in its vertical side rests on a smooth horizontal surface. When the depth of water in the tank is 1.22 m the discharge through the orifice is 1118.34 N per minute* [*114 kg(f) per minute*] *and a horizontal force of 9.123 N in line with the centre of the orifice is required to keep the tank at rest. From these data determine the coefficients C_v, C_c and C_d.*

Solution

Actual discharge $$Q = \frac{1118.34}{9810 \times 60} = 1.9 \times 10^{-3} \text{ m}^3/\text{s}$$

Theoretical discharge

$$Q_{th} = a\sqrt{2gh}$$
$$= (6.45 \times 10^{-4}) \times \sqrt{2 \times 9.81 \times 1.22}$$
$$= 3.16\ 10^{-3} \text{ m}^3/\text{s}$$

$$\therefore \quad C_d = \frac{1.9 \times 10^{-3}}{3.16 \times 10^{-3}} = 0.601$$

According to impulse-momentum principle

Force $$F = \frac{WV}{g}$$

or $$9.123 = \frac{1118.34}{60} \times \frac{V}{9.81}$$

$$\therefore \quad V = \frac{9.123 \times 60 \times 9.81}{1\,118.34} = 4.80 \text{ m/s}$$

Theoretical velocity

$$V_{th} = \sqrt{2gh}$$
$$= \sqrt{2 \times 9.81 \times 1.22} = 4.89 \text{ m/s}$$

$$\therefore \quad C_v = \frac{4.80}{4.89} = 0.982$$

Since $$C_d = C_c \times C_v$$

$$\therefore \quad C_c = \frac{C_d}{C_v} = \frac{0.601}{0.982} = 0.612$$

Example 9.5. *A circular sharp-edged orifice of 25 mm diameter is situated in a vertical side of a large oil tank and discharges to a region where the pressure is 82.404 kN/m^2* [*0.84 kg(f)/cm^2*] *absolute. The oil in the tank is maintained at a constant level and subject to an external pressure of 48.069 kg(f)/m^2* [*0.49 kg(f)/cm^2*) *gage.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The values of C_c and C_v for the orifice are 0.62 and 0.98 respectively, the specific gravity of the oil is 0.85 and the barometric pressure head is 760 mm of mercury absolutely.

If the jet drops 0.05 m in a horizontal distance of 1.375 m, calculate (a) the height of the oil surface in the tank above the axis of the orifice, (b) the volume rate of flow through the orifice.

Solution

(a) From Eq. 9.9, we have

$$V = \sqrt{\frac{gx^2}{2y}}$$

or $$\frac{V^2}{2g} = \frac{x^2}{4y}$$

$$x = 1.375 \text{ m; and } y = 0.05 \text{ m}$$

$\therefore$ $$\frac{V^2}{2g} = \frac{(1.375)^2}{4 \times 0.05}$$

$$= 9.453 \text{ m of oil}$$

But $$V = C_v \sqrt{2gh}$$

or $$h = \frac{1}{C_v^2} \times \frac{V^2}{2g}$$

$\therefore$ $$h = \frac{9.453}{(0.98)^2} = 9.843 \text{ m of oil}$$

Now if h' is the height of the oil surface in the tank above the axis of the orifice, then

In SI Units

$$h = 9.843 = \frac{760 \times 13.6}{1000 \times 0.85} + \frac{48.069 \times 10^3}{9810 \times 0.85} + h' - \frac{82.404 \times 10^3}{9810 \times 0.85}$$

$\therefore$ $$h' = 1.80 \text{ m}$$

In Metric Units

$$h = 9.843 = \frac{760 \times 13.6}{1000 \times 0.85} + \frac{0.49 \times 10^4}{1000 \times 0.85} + h' - \frac{0.84 \times 10^4}{1000 \times 0.85}$$

$\therefore$ $$h' = 1.80 \text{ m}$$

(b) $$Q = C_c C_v a \sqrt{2gh}$$

$$= (0.62 \times 0.98) \frac{\pi}{4} (25 \times 10^{-3})^2 \times \sqrt{2 \times 9.81 \times 9.843}$$

$$= 0.004\ 14 \text{ m}^3/\text{s} = 4.14 \text{ l/s}.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 9.6. *A circular orifice of area 6.45×10^{-4} m^2 is provided in the vertical side of a large tank. The tank is suspended from a knife edge 1.53 m above the level of the orifice. When the head of water is 1.22 m the discharge is 1161.5 N/min [118.4 kg(f)/min] and a turning moment of 14.421 N-m [1.47 kg(f)-m] has to be applied to the knife edges to keep the tank vertical. Determine C_v, C_d and C_c of the orifice.*

Solution

From Eq. 9.12, we have

$$C_v = \frac{Wlg}{W_l y\sqrt{2gH}}$$

$$(Wl) = 14.421 \text{ N-m}$$

$$W_l = \frac{1161.5}{60} = 19.358 \text{ N/s}$$

$$y = 1.53 \text{ m}; h = 1.22 \text{ m}$$

Thus by substitution, we get

$$C_v = \frac{14.421 \times 9.81}{19.358 \times 1.53 \times \sqrt{2 \times 9.81 \times 1.22}}$$

$$= 0.976$$

Actual discharge $$Q = \frac{1161.5}{9810 \times 60} = 1.973 \times 10^{-3} \text{ m}^3/\text{s}$$

Theoretical discharge

$$Q_{th} = a\sqrt{2gh}$$

$$= 6.45 \times 10^{-4} \times \sqrt{2 \times 9.81 \times 1.22}$$

$$= 3.156 \ \ 10^{-3} \text{ m}^3/\text{s}$$

$$\therefore \quad C_d = \frac{1.973 \times 10^{-3}}{3.156 \times 10^{-3}} = 0.625$$

Since $$C_d = C_c \times C_v$$

$$C_c = \frac{C_d}{C_v} = \frac{0.625}{0.976} = 0.640$$

Example 9.7. *A reservoir discharges through a sluice 0.915 m wide by 1.22 m deep. The top of the opening is 0.61 m below the water level in the reservoir and the downstream water level is below the bottom of the opening. Calculate (a) the discharge through the opening if $C_d = 0.60$; and (b) percentage error if the opening is treated as a small orifice.*

Solution

(a) From Eq. 9.15, we have

$$Q = \frac{2}{3} C_d B \sqrt{2g} \left[H_2^{3/2} - H_1^{3/2} \right]$$

$$C_d = 0.60; B = 0.915 \text{ m};$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_2 = (1.22 + 0.61) = 1.83 \text{ m}$$
$$H_1 = 0.61 \text{ m}$$

Thus by substitution, we get

$$Q = \frac{2}{3} \times 0.60 \times 0.915 \times \sqrt{2 \times 9.81} \left[(1.83)^{3/2} - (0.61)^{3/2} \right]$$
$$= 3.241 \text{ m}^3/\text{s}$$

(b) For a small orifice

$$Q = C_d \, a \sqrt{2gh}$$
$$a = (0.915 \times 1.22) = 1.116 \text{ m}^2$$
$$h = \left(0.61 + \frac{1.22}{2} \right) = 1.22 \text{ m}$$

Thus
$$Q = 0.60 \times 1.116 \times \sqrt{2 \times 9.81 \times 1.22}$$
$$= 3.276 \text{ m}^3/\text{s}$$

$\therefore$
$$\text{Error} = \frac{3.276 - 3.241}{3.241}$$
$$= 0.010\,8 \text{ or } 1.08\%$$

Example 9.8. *Water is supplied from a tank into a canal through a rectangular sluice 1 m wide and 1.75 m high. The water level in the tank is 2 m above the top edge of the opening and the canal water level is 0.3m below the top edge. If the coefficient of discharge is 0.62 for both the free and the submerged portions, calculate the discharge.*

Solution

In this case the orifice is partially submerged, thus the discharge through the free portion is

$$Q_1 = \frac{2}{3} C_d \, B \sqrt{2g} \left(H_2^{3/2} - H_1^{3/2} \right)$$
$$C_d = 0.62; \; B = 1 \text{ m}$$
$$H_2 = (2 + 0.3) = 2.3 \text{ m}$$

and
$$H_1 = 2 \text{ m}$$

Thus by substitution, we get

$$Q_1 = \frac{2}{3} \times 0.62 \times 1 \times \sqrt{2 \times 9.81} \; [(2.3)^{3/2} - (2)^{3/2}]$$
$$= 1.208 \text{ m}^3/\text{s}$$

Discharge through the submerged portion is

$$Q_2 = C_d \, a \sqrt{2gh}$$
$$C_d = 0.62$$
$$a = 1 \times (1.75 \times 0.3) = 1.45 \text{ m}^2$$
$$H = (2 + 0.3) = 2.3 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by substitution, we get

$$Q_2 = 0.62 \times 1.45 \times \sqrt{2 \times 9.81 \times 2.3}$$
$$= 6.039 \text{ m}^3/\text{s}$$

∴ Total discharge through the orifice

$$Q = (Q_1 + Q_2) = (1.208 + 6.039) = 7.247 \text{ m}^3/\text{s}$$

Example 9.9. *Water flows vertically downwards through a 0.15 m diameter pipe at the rate of 70 litres per second. The pipe suddenly enlarges to 0.3 m diameter. A is a point 0.5 m distant from and above the section of enlargement, and B is another point 0.5 m below the same. It was found that pressure gage connected to A gives a reading of 210.915 kN/m². Determine the pressure at point B. Neglect the friction loss between A and B.*

Recalculate the pressure at B, if the same discharge flows up the pipe, considering the pressure at A as unchanged. Assume that $C_c = 0.64$.

Solution

Area of pipe at $A = \frac{\pi}{4}(0.15)^2 = 0.017\,7 \text{ m}^2$

Area of pipe at $B = \frac{\pi}{4}(0.30)^2 = 0.0708 \text{ m}^2$

$$Q = 70\ l/\text{s} = 0.07 \text{ m}^3/\text{s}$$

$$V_A = \frac{0.07}{0.0177} = 3.96 \text{ m/s}$$

$$V_B = \frac{0.07}{0.0708} = 0.99 \text{ m m/s}$$

When the flow is vertically downwards, there is sudden enlargement and hence there is a loss of head due to sudden enlargement which equals

$$h_L = \frac{(V_A - V_B)^2}{2g}$$

$$= \frac{(3.96 - 0.99)^2}{2 \times 9.81} = 0.45 \text{ m}$$

Applying Bernoulli's equation between the points *A* and *B*, we have

$$\frac{p_A}{w} + \frac{V_A^2}{2g} + z_A = \frac{p_B}{w} + \frac{V_B^2}{2g} + z_B + h_L$$

$$p_A = 210.915 \text{ kN/m}^2 = 210.915 \times 10^3 \text{ N/m}^3$$
$$z_A = (0.50 + 0.50) = 1.0 \text{ m}$$
$$z_B = 0, \text{ assuming datum level at } B.$$

Thus by substitution, we have

$$\frac{210.915 \times 10^3}{9810} + \frac{(3.96)^2}{2 \times 9.81} + 1.0 = \frac{p_B}{w} + \frac{(0.99)^2}{2 \times 9.81} + 0 + 0.45$$

$$\therefore \quad p_B = 223.661 \text{ kN/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When the flow is vertically upwards there is a sudden contraction and hence the loss of head is due to sudden contraction which equals

$$h'_L = \frac{V_A^2}{2g}\left(\frac{1}{C_c}-1\right)^2$$

$$= \frac{(3.96)^2}{2\times 9.81}\left(\frac{1}{0.64}-1\right)^2 = 0.253 \text{ m}$$

Again applying Bernoulli's equation between the points B and A, we have

$$\frac{p_B}{w} + \frac{V_B^2}{2g} + Z_B = \frac{p_A}{w} + \frac{V_A^2}{2g} + Z_A + h'_L$$

or $$\frac{p_B}{w} + \frac{(0.99)^2}{2\times 9.81} + 0 = \frac{210.915\times 10^3}{9810} + \frac{(3.96)^2}{2\times 9.81} + 1.0 + 0.253$$

$$\therefore \quad p_B = 230.558 \text{ kN/m}^2$$

Example 9.10. *(a) For sudden expansion in pipe flow what is the optimum ratio between the diameter of the pipe before expansion and the diameter of the pipe after expansion so that the pressure rise may be maximum. What will be the corresponding pressure rise?*

(b) Oil of specific gravity 0.8 flows in a 0.08 m diameter pipeline of such diameter that the maximum pressure rise is obtained. If the rate of flow through the pipeline is 12.5 liters per second find (i) the loss of energy in the sudden expansion in m of oil, and (ii) the differential gage length h indicated on an oil mercury manometer connected between the two pipes.

Solution

(a) As indicated in Section 9.8, by applying the momentum equation the pressure rise between two sections on either side of the sudden expansion may be expressed as

$$\frac{p_2 - p_1}{w} = \frac{V_2(V_1 - V_2)}{g}$$

If Q is the discharge and d_1 and d_2 are the diameters of the pipe before and after the enlargement respectively, then

$$V_1 = \frac{Q}{(\pi/4)d_1^2} \text{ ; and } V_2 = \frac{Q}{(\pi/4)d_2^2}$$

and by substituting these values of V_1 and V_2, we get

$$\frac{p_2 - p_1}{w} = \frac{Q^2}{g(\pi/4)^2 d_1^4}\left[\left(\frac{d_1}{d_2}\right)^2 - \left(\frac{d_1}{d_2}\right)^4\right]$$

Let $\left(\frac{p_2 - p_1}{w}\right)$ be equal to H and $\frac{d_1}{d_2}$ be equal to x, then

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H = \frac{Q^2}{g(\pi/4)^2 d_1^4}(x^2 - x^4)$$

For a given discharge Q flowing through a pipe of given diameter d_1 the condition for maximum pressure rise at a sudden expansion may be obtained by equating (dH/dx) to zero. Thus

$$\frac{dH}{dx} = \frac{Q^2}{g(\pi/4)^2 d_1^4}(2x - 4x^3) = 0$$

from which the optimum value of the ratio (d_1/d_2) is obtained as

$$x = \frac{d_1}{d_2} = \frac{1}{\sqrt{2}} = 0.707$$

The corresponding pressure rise

$$H_{\max} = \frac{Q^2}{4g(\pi/4)^2 d_1^4} = \frac{V_1^2}{4g}$$

(b) (i) From Eq. 9.25 the head loss due to sudden expansion is given by

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

$$d_1 = 0.08 \text{ m; and } d_2 = \left(0.08 \times \sqrt{2}\right) = 0.113 \text{ m}$$

$$V_1 = \frac{12.5 \times 10^{-3}}{(\pi/4) \times (0.08)^2} = 2.487 \text{ m/s}$$

$$V_2 = \frac{12.5 \times 10^{-3}}{(\pi \times 4) \times (0.113)^2} = 1.246 \text{ m/s}$$

$$\therefore \quad h_L = \frac{(2.487 - 1.246)^2}{2 \times 9.81} = 0.0785 \text{ m of oil}$$

(ii) By substituting the given values in the above noted expression for the pressure rise at the sudden expansion and considering the optimum value of (d_1/d_2) as $(1/\sqrt{2})$, we get

$$\frac{p_2 - p_1}{w} = \frac{(12.5 \times 10^{-3})^2}{9.81 \times (\pi/4)^2 (0.08)^4}\left(\frac{1}{2} - \frac{1}{4}\right)$$

$$= 0.1576 \text{ m of oil}$$

When an oil mercury manometer is connected between the two pipes, then

$$\frac{p_2 - p_1}{w} = h\left(\frac{S_m}{S_{\text{oil}}} - 1\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$0.1576 = h\left(\frac{13.6}{0.8} - 1\right)$$

$$\therefore \quad h = 9.85 \times 10^{-3} \text{ m} = 9.85 \text{ mm}$$

Example 9.11. *At a sudden enlargement of a pipeline from a diameter of 0.3 m to 0.6 m, the hydraulic gradient line rises 0.125 m. Estimate the discharge if the pipe is horizontal.*

Solution

Let V_1 and V_2 be the velocities of flow in 0.3 m and 0.6 m pipes respectively. Then by continuity equation

$$\frac{\pi}{4}(0.3)^2 V_1 = \frac{\pi}{4}(0.6)^2 \times V_2$$

or $$V_1 = 4V_2$$

Applying Bernoulli's equation between the points 1 and 2 lying on either side of the enlargement, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_L$$

From the given data

$$z_1 = z_2 \text{ ; and } \left(\frac{p_2}{w} - \frac{p_1}{w}\right) = 0.125 \text{ m}$$

and $$h_L = \frac{(V_1 - V_2)^2}{2g} = \frac{(4V_2 - V_2)^2}{2g} = \frac{9V_2^2}{2g}$$

Thus by substitution, we have

$$\frac{(4V_2)^2}{2g} = 0.125 + \frac{V_2^2}{2g} + \frac{9V_2^2}{2g}$$

or $$V_2 = 0.639 \text{ m/s}$$

$$\therefore \quad Q = a_2 V_2 = \frac{\pi}{4}(0.6)^2 \times 0.639$$

$$= \frac{\pi}{4}(0.6)^2 \times 0.639$$

$$= 0.181 \text{ m}^3/\text{s} = 181 \text{ } l/\text{s}$$

Example 9.12. *Water under a constant head of 4.5 m discharges through an external cylindrical mouthpiece 50 mm diameter and 150 mm long. If C_c for the orifice is 0.60, find (a) the discharge in litres per second; (b) the coefficient of discharge; and (c) the absolute pressure at vena-contracta. Assume atmospheric pressure to be 10.3 m of water.*

Solution

Refer Fig. 9.13. Applying Bernoulli's equation between the free surface and the outlet section of the mouthpiece, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

The loss of head is due to sudden enlargement between the vena-contracta and the end sections and hence

$$h_L = \frac{(V_c - V)^2}{2g}$$

By continuity $a_c V_c = aV$; or $V_c = \frac{V}{C_c}$

$$\therefore \quad h_L = \frac{V^2}{2g}\left(\frac{1}{C_c} - 1\right)^2$$

$$= \frac{V^2}{2g}\left(\frac{1}{0.6} - 1\right)^2 = 0.444\ \frac{V^2}{2g}$$

Thus by substitution, we get

$$4.5 = \frac{V^2}{2g} + 0.444\ \frac{V^2}{2g}$$

$$\therefore \quad V = 7.819 \text{ m/s}$$

$$Q = aV$$

$$= \left[\frac{\pi}{4} \times (0.05)^2 \times 7.819\right]$$

$$= 0.015\ 4 \text{ m}^3/\text{s} = 15.4\ l/\text{s}$$

$$C_v = \frac{V}{\sqrt{2gH}}$$

$$= \frac{7.819}{\sqrt{2 \times 9.81 \times 4.5}} = 0.832$$

Since for the outlet end of the mouthpiece $C_v = 1$

$$\therefore \quad C_d = C_c \times C_v = (1 \times 0.832) = 0.832$$

For finding the absolute pressure at vena-contracta, apply Bernoulli's equation between free surface of liquid and the vena-contracta. Thus

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

$$V_c = \frac{V}{C_c} = \frac{7.819}{0.6} = 13.03 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substitution, we get

$$10.3 + 4.5 = h_c + \frac{(13.03)^2}{2\times 9.81}$$

$$\therefore \quad h_c = 6.147 \text{ m of water absolute.}$$

Example 9.13. *Water under a constant head of 3 m discharges through an external cylindrical mouthpiece 50 mm diameter, for which C_v = 0.82, determine (a) the discharge in cumec; (b) absolute pressure at vena-contracta; and the maximum head for the mouthpiece to flow full.*

Solution

For the mouthpiece at the outlet end $C_c = 1$.

Thus
$$C_d = C_c \times C_v = (1 \times 0.82) = 0.82$$

$$Q = C_d\, a\, \sqrt{2gh}$$

$$= 0.82 \times \frac{\pi}{4}(0.05)^2 \times \sqrt{2\times 9.81\times 3}$$

$$= 0.0124 \text{ m}^3/\text{s}$$

If the head loss in the mouthpiece is expressed as $k\,\frac{V^2}{2g}$, then applying Bernoulli's equation between the free surface and the outlet section of the mouthpiece, we have

$$H_a + 3 = H_a + \frac{V^2}{2g} + k\,\frac{V^2}{2g}$$

or
$$V = \sqrt{\frac{2g\times 3}{(1+k)}}$$

But
$$V = C_v\sqrt{2g\times 3} = 0.82\sqrt{2g\times 3}$$

$$\therefore \quad C_v = \sqrt{\frac{1}{1+k}} = 0.82$$

or
$$k = 0.487$$

Further the loss of head in the mouthpiece is due to sudden enlargement, which equals

$$h_L = \frac{(V_c - V)^2}{2g} = \frac{V^2}{2g}\left(\frac{1}{C_c} - 1\right)^2$$

From which
$$k = \left(\frac{1}{C_c} - 1\right)^2$$

or
$$0.487 = \left(\frac{1}{C_c} - 1\right)^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore \quad C_c = 0.589$

By continuity $\quad V_c = V\frac{a}{a_c} = \frac{V}{C_c}$

$\therefore \quad V_c = \frac{0.82\sqrt{2\times 9.81\times 3}}{0.589} = 10.68 \text{ m/s}$

Again applying Bernoulli's equation between the free surface and the vena-contracta, we have

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

or $\quad 10.3 + 3 = H_c + \frac{(10.68)^2}{2\times 9.81}$

$\therefore \quad H_c = 7.486$ m of water absolute

Let H be the maximum head upto which the mouthpiece flows full and let the corresponding minimum pressure at vena-contracta be 2.5 m of water absolute. Then

$$10.3 + H = 2.5 + \frac{V_c^2}{2g}$$

But $\quad V_c = \frac{V}{C_c}$

$$= \frac{C_v\sqrt{2gH}}{C_c} = \frac{0.82\sqrt{2gH}}{0.589}$$

or $\quad 10.3 + H = 2.5 + \left(\frac{0.82}{0.589}\right)^2 H$

$\therefore \quad H = \frac{7.8}{0.938} = 8.316$ m

However, if the minimum pressure at vena-contracta is taken as zero, then

$$H = \frac{10.3}{0.938} = 10.98 \text{ m}$$

Example 9.14. *A Borda's mouthpiece 40 mm diameter discharges under a constant head of 1.5 m. If the coefficient of velocity for the entrance section of the mouthpiece is 0.95, find (a) the coefficient of contraction and the discharge when the mouthpiece is running free, and (b) the discharge when the mouth piece is running, full, assuming that vena-contracta is formed in the mouthpiece and the coefficient of contraction being the same as in (a) and allowing for the loss of energy between the vena-contracta and the outlet.*

Solution

(a) For the Borda's mouthpiece running free as indicated in Section 9.11.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{a_c}{a} = \frac{gH}{V_c^2}$$

But $V_c = C_v \sqrt{2gH}$

$$\therefore \quad \frac{a_c}{a} = C_c = \frac{1}{2C_v^2} = \frac{1}{2\times(0.95)^2} = 0.554$$

Discharge $Q = a_c V_c = C_c a \times C_v \sqrt{2gH}$

$$\therefore \quad Q = 0.554 \times \frac{\pi}{4}(0.04)^2 \times 0.95 \times \sqrt{2\times 9.81\times 1.5}$$

$$= 3.588 \times 10^{-3} \text{ m}^3/\text{s} = 3.588 \ l/\text{s}.$$

(b) For the Borda's mouthpiece running full, the loss of head between the vena-contracta and the outlet is

$$h_L = \frac{(V_c^2 - V)^2}{2g}$$

$$= \frac{V^2}{2g}\left(\frac{1}{C_c} - 1\right)^2$$

$$= \frac{V^2}{2g}\left(\frac{1}{0.554} - 1\right)^2 = 0.648\frac{V^2}{2g}$$

Applying Bernoulli's equation between the free surface and the outlet, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

or $$H = \frac{V^2}{2g} + 0.648\ \frac{V^2}{2g}$$

or $$1.5 = 1.648\ \frac{V^2}{2g}$$

$$\therefore \quad V = \sqrt{\frac{2\times 9.81\times 1.5}{1.648}} = 4.226 \text{ m/s}$$

$$\therefore \quad \text{Discharge } Q = aV$$

$$= \frac{\pi}{4}(0.04)^2 \times 4.226$$

$$= 5.311 \times 10^{-3} \text{ m}^3/\text{s} = 5.311 \ l/\text{s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 9.15. *A convergent-divergent nozzle is fitted into the side of a tank containing water. Assuming that there are no losses in the convergent part of the nozzle, that the losses in the divergent part are equivalent to 0.18 times the velocity head at exit, and that the minimum absolute pressure at the throat is 2.5 m of water for a barometric pressure of 10.3 m of water, determine the throat and exit diameters of the nozzle to discharge 0.0042 m^3/s for a head of 1.5 m above the centre-line of the nozzle.*

Solution

Refer Fig. 9.14 (c). Applying Bernoulli's equation between the free surface and the throat, we have

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

H_a = 10.3; H = 1.5 m; and H_c = 2.5 m

Thus by substitution, we have

$$10.3 + 1.5 = 2.5 + \frac{V_c^2}{2g}$$

or $V_c = 13.508$ m/s

$Q = a_c V_c$

and $Q = 0.004\,2$ m^3/s

Thus $0.0042 = \frac{\pi}{4}(d_c)^2 \times 13.508$

$$\therefore \quad d_c = \sqrt{\frac{4 \times 0.0042}{\pi \times 13.508}}$$

$= 0.019\,9$ m or 19.9 mm

i.e., throat diameter = 19.9 mm

Applying Bernoulli's equation between the free surface and exit, allowing for the loss of energy in the divergent portion, we have

$$H_a + H = H_a + \frac{V^2}{2g} + 0.18\frac{V^2}{2g}$$

or $$V = \sqrt{\frac{2 \times 9.81 \times 1.5}{1.18}} = 4.994 \text{ m/s}$$

Now if d is the diameter at the exit, then

$$Q = 0.004\,2 = \frac{\pi}{4}(d)^2 \times 4.994$$

$$\therefore \quad d = \sqrt{\frac{4 \times 0.0042}{\pi \times 4.994}} = 0.032\,7 \text{ m} = 32.7 \text{ mm}$$

Example 9.16. *In a laboratory experiment, on finding the coefficient of discharge of an orifice 25 mm*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

diameter, provided in a tank of constant cross-sectional area of 0.305 m by 0.305 m, it was found that it took $1\frac{1}{4}$ *minutes for the head over the centre of the orifice to fall from 1.2 m to 0.3 m. Find the coefficient of dishcarge of the orifice.*

Solution

From Eq. 9.40, we have

$$t = \frac{2A}{C_d a\sqrt{2g}}\left(H_1^{1/2} - H_2^{1/2}\right)$$

$$t = 1\frac{1}{4} \text{ min} = 75 \text{ s}$$

$$A = (0.305 \times 0.305) \text{ m}^2;$$

$$a = \frac{\pi}{4}(0.025)^2$$

$$H_1 = 1.2 \text{ m; and } H_2 = 0.3 \text{ m}$$

Thus by substitution, we have

$$75 = \frac{2\times(0.305\times0.305)}{C_d \times \frac{\pi}{4}(0.025)^2 \times\sqrt{2\times9.81}}[(1.2)^{1/2} - (0.3)^{1/2}]$$

$$\therefore \quad C_d = 0.625$$

Example 9.17. *A tank of constant cross-sectional area of 2.8 m² has two orifices each* 9.3×10^{-4} *m² in area in one of its vertical sides at heights of 6 m and 1.5 m respectively above the bottom of the tank. Calculate the time taken to lower the water level from 9 m to 3.6 m above the bottom of the tank. Assume* $C_d = 0.625$.

Solution

In the beginning both the orifices will be discharging till the water level falls to 6 m, then the top orifice will be ineffective and the discharge will take place only through the lower orifice. Thus the total time taken to lower the water level may be divided into two parts viz., time taken to lower the water level from 9 m to 6 m when both the orifices are discharging, and the time taken to lower the water level from 6 m to 3.6 m when only the lower orifice is discharging.

Let t_1 and t_2 be these times, then the total time t is equal to $(t_1 + t_2)$. Let at any instant water level be at a height h above the lower orifice and let the water level be lowered by dh in time dt, then

$$Q\,dt = -A\,dh$$

in which $Q = (Q_1 + Q_2)$, where Q_1 and Q_2 are the discharges through the upper and lower orifices respectively.

Then $$Q_1 = C_d \times (9.3 \times 10^{-4})\sqrt{2g(h - 4.5)}$$

and $$Q_2 = C_d \times (9.3 \times 10^{-4})\sqrt{2gh}$$

Since the vertical distance between the two orifices

$$= (6 - 1.5) = 4.5 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substitution, we have

$$\left[C_d \times (9.3\times10^{-4}) \times \sqrt{2g}\left(\sqrt{h-4.5}+\sqrt{h}\right)\right]dt = -A\,dh$$

$C_d = 0.625$; $A = 2.8$ m^2

$H_1 = (9 - 1.5) = 7.5$ m

and $H_2 = (6 - 1.5) = 4.5$ m

$$\therefore \quad dt = -\frac{2.8}{0.625\times(9.3\times10^{-4})\times\sqrt{2g}} \times \frac{dh}{(\sqrt{h-4.5}+\sqrt{h})}$$

By integrating both sides of the above equation, we get

$$\int_0^{t_1} dt = -\frac{2.8}{0.625\times(9.3\times10^{-4})\times\sqrt{2g}} \times \int_{7.5}^{4.5} \frac{dh}{\left(\sqrt{h-4.5}+\sqrt{h}\right)}$$

or $$t_1 = -1087.54 \int_{7.5}^{4.5} \left[\frac{(\sqrt{h-4.5}-\sqrt{h})}{-4.5}\right] dh$$

$$= 241.68\left[\frac{2}{3}(h-4.5)^{3/2} - \frac{2}{3}h^{3/2}\right]_{7.5}^{4.5}$$

$$= \frac{241.68\times2}{3}\left[0-(4.5)^{3/2}-(3)^{3/2}+(7.5)^{3/2}\right]$$

$$= 934 \text{ s}$$

The value of t_2 may be obtained directly by Eq. 9.40 as

$$t_2 = \frac{2A}{C_d a\sqrt{2g}}(H_1^{1/2} - H_2^{1/2})$$

$H_1 = (6 - 1.5) = 4.5$ m

and $H_2 = (3.6 - 1.5) = 2.1$ m

$$\therefore \quad t_2 = \frac{2\times2.8\left[(4.5)^{1/2}-(2.1)^{1/2}\right]}{0.625\times(9.3\times10^{-4})\times\sqrt{(2\times9.81)}}$$

$$= 1462 \text{ s}$$

$\therefore$ Total time $t = (934 + 1462)$

$= 2\,396$ s $= 39.93$ min.

Example 9.18. *A cylindrical boiler, diameter 2 m and length 10 m is lying horizontally and is half full of water. It is to be emptied by an external cylindrical mouthpiece of diameter 75 mm fixed at the bottom of the boiler. How long will the mouthpiece take to empty the boiler? Take the coefficient of discharge for the mouthpiece as 0.82.*

Solution

From Eq. 9.42, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$t = \frac{4l}{3C_d a\sqrt{2g}}[(2R - H_2)^{3/2} - (2R - H_1)^{3/2}]$$

l = 10 m; $C_d = 0.82$; $a = \frac{\pi}{4}(0.075)^2 = 4.42 \times 10^{-3}$ m^2;

$(2R)$ = 2 m; $H_1 = 1$ m; and $H_2 = 0$

Thus by substitution, we get

$$t = \frac{4\times10}{3\times0.82\times4.42\times10^{-3}\times\sqrt{2\times9.81}}[(2)^{3/2} - 1]$$

$$= 1518.56 \text{ s} = 25.31 \text{ min.}$$

Example 9.19. *A swimming bath 36 m long and 12 m wide has vertical sides, and the bottom slopes uniformly from a depth of 1.2 m at the shallow end to 2.1 m at the deep end. There are two outlets at the bottom of the deeper end, one is an orifice 0.225 m diameter and the other a small pipe 0.75 m long fitted on the outside of a similar opening of 0.225 m diameter. Find the time taken to empty the tank if the values of C_d for the orifice and the mouthpiece are 0.62 and 0.85 respectively.*

Solution

As shown in Fig. Ex. 9.19 upto the shallow end the tank has a constant cross-sectional area but below the shallow end the cross-sectional area varies. Thus the total time t taken to empty may be divided into two parts t_1 and t_2 required to empty these two portions, which may be computed as indicated below.

Let at any instant water level be at a height h above the centre of the openings and in time dt, let the water level be lowered by dh.

Then $Q dt = -A\, dh$

$$Q = \left[0.62\times\frac{\pi}{4}(0.225)^2\times\sqrt{2gh}\right] + \left[0.85\times\frac{\pi}{4}(0.225)^2\times\sqrt{2gh}\right]$$

$$= 0.259\sqrt{h}$$

$A = (36 \times 12) = 432$ m^2; $H_2 = 2.1$ m;

$H_1 = (2.1 - 1.2) = 0.9$ m

Thus by substitution, we have

$$dt = -\frac{432\,dh}{0.259\sqrt{h}}$$

By integrating both sides, we have

$$\int_0^{t_1} dt = -\int_{2.1}^{0.9}\frac{432\,dh}{0.259\sqrt{h}}$$

$$\therefore \quad t_1 = \frac{432\times2}{0.259}\left[(2.1)^{1/2} - (0.9)^{1/2}\right]$$

$$= 1669.5 \text{ s}$$

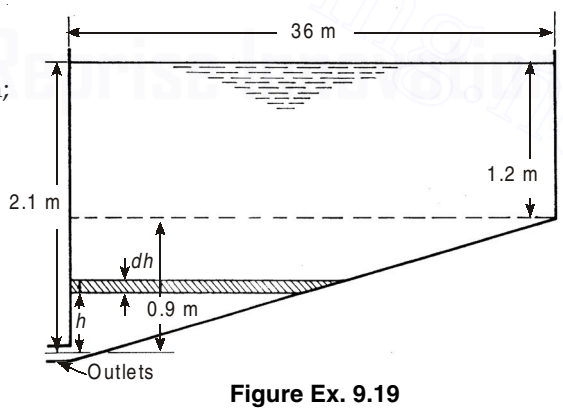


Figure Ex. 9.19

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As shown in Fig. Ex. 9.19 if h is the height of the water level above the centre of the opening at any instant in the lower portion of the tank then cross-sectional area of the tank at this level is

$$A = \left(\frac{36 \times h \times 12}{0.9}\right) = 480\ h$$

Again if in time dt the water level is lowered by dh, then

$$Q\,dt = -A\,dh$$

or $$dt = -\frac{A\,dh}{Q} = \frac{-480\ h\ dh}{0.259\sqrt{h}}$$

h varies from $H_1 = 0.9$ m to $H_2 = 0$

Thus by integrating both sides, we have

$$\int_0^{t_2} dt = -\frac{480}{0.259}\int_{0.9}^{0} h^{1/2}\,dh$$

or $$t_2 = \frac{480}{0.259} \times \frac{2}{3}\left[(0.9)^{3/2} - 0\right]$$

$$= 1054.9 \text{ s}$$

Therefore total time,

$$t = (t_1 + t_1)$$
$$= (1669.5 + 1054.9)$$
$$= 2724.4 \text{ s} = 45.41 \text{ min}$$

Example 9.20. *A tank 3 m high and filled with water has the form of frustrum of a cone with the smaller end pointing downwards. The upper end is 2.4 m diameter and lower end 1.2 m diameter. The bottom end contains a central orifice whose average coefficient of discharge is 0.6, which is to empty the tank in 6 minutes. Find the size of the orifice.*

Solution

Refer Fig. 9.16 (c)

$$R_1 = \frac{2.4}{2} = 1.2 \text{ m}; \quad R_0 = \frac{1.2}{2} = 0.6 \text{ m}$$

$$H_1 = 3 \text{ m}$$

$\therefore$ $$H_0 = \frac{R_0 H_1}{R_1 - R_0} = \frac{0.6 \times 3}{1.2 - 0.6} = 3 \text{ m}$$

From Eq. 9.43, we have

$$t = -\frac{\pi R_1^2}{C_d a\sqrt{2g}(H_1 + H_0)^2}$$

$$\times\left[\frac{2}{5}h^{5/2} + 2H_0^2 h^{1/2} + \frac{4}{3}H_0 h^{3/2}\right]_{H_1}^{H_2}$$

$t = (6 \times 60) = 360$ s; $C_d = 0.6$; $H_2 = 0$; and $H_1 = 3$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Let d be the diameter of the orifice. Thus by substitution, we have

$$360 = \frac{\pi \times (1.2)^2}{0.6 \times \frac{\pi}{4} d^2 \times \sqrt{2 \times 9.81} \times (3+3)^2} \times \left[\frac{2}{5}(3)^{5/2} + 2 \times (3)^2 \times (3)^{1/2} + \frac{3}{4} \times (3) \times (3)^{3/2} \right]$$

$$\therefore \quad d = 0.098\ 65 \text{ m} = 98.65 \text{ mm}.$$

Example 9.21. *A rectangular tank 6 m long by 1.5 m wide is divided into two parts by a partition so that one part is 4 times the other part. The water level in the large portion is 3 m above that in the smaller. Find the time required for the difference of water levels in the two portions to be reduced to 1.2 m, if the water flows through an orifice in the bottom of the partition having an area of 58× 10^{-4} m^2 and $C_d = 0.6$.*

Solution

From Eq. 9.47, we have

$$t = \frac{2A_1A_2(H_1^{1/2} - H_2^{1/2})}{C_d a (A_1 + A_2)\sqrt{2g}}$$

$$A_1 = 4A_2$$

and $$(A_1 + A_2) = (6 \times 1.5) = 9 \text{ m}^2$$

$$\therefore \quad A_1 = 7.2 \text{ m}^2; \text{ and } A_2 = 1.8 \text{ m}^2$$

$$H_1 = 3 \text{ m}; H_2 = 1.2 \text{ m}$$

$$C_d = 0.6; a = 58 \times 10^{-4} \text{ m}^2$$

Thus by substitution, we have

$$t = \frac{2 \times (7.2 \times 1.8)\left[(3)^{1/2} - (1.2)^{1/2} \right]}{0.6 \times 58 \times 10^{-4} \times (7.2 + 1.8) \times \sqrt{2 \times 9.81}}$$

$$= 119 \text{ s} = 1 \text{ min } 59 \text{ s}$$

Example 9.22. *A reservoir 180 m × 120 m at bottom has slopes 1 to 1 and a depth of 7.5 m of water. Find the time taken to reduce the water level by 2.5 m by a short tube 1m diameter at the bottom having $C_d = 0.70$.*

Solution

Let at any instant the water level be at a height h above the orifice and let in time dt the water level be reduced by dh. Then

$$Q\,dt = -A\,dh$$

$$Q = C_d a \sqrt{2gh} = 0.70 \times \frac{\pi}{4}(1)^2 \times \sqrt{2 \times 9.81 \times h}$$

$$A = (180 + 2h)\ (120 + 2h)$$

Thus by substitution, we have

$$dt = -\frac{(180 + 2h)(120 + 2h)dh}{0.70 \times \frac{\pi}{4}(1)^2 \times \sqrt{2 \times 9.81 \times h}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By integrating both sides, we have

$$\int_0^t dt = -\frac{4}{0.70\times\pi\times\sqrt{2\times9.81}}\int_{H_1}^{H_2}\frac{(180+2h)(120+2h)dh}{\sqrt{h}}$$

H_1 = 7.5 m; and $H_2 = (7.5 - 2.5) = 5$ m

$$\therefore \quad t = \frac{-4\times4}{0.7\times\pi\times\sqrt{2\times9.81}}\times\int_{7.5}^{5}\frac{(90+h)(60+h)dh}{\sqrt{h}}$$

$$= 10\,517 \text{ s} = 2 \text{ h } 55 \text{ min}$$

Example 9.23. *A cylindrical tank of internal diameter 0.6 m, length 1.5 m and axis vertical has a 50 mm diameter sharp-edged orifice ($C_d = 0.6$) in the bottom opening to atmosphere. The tank is open at the top and is empty. If water is admitted into the tank from above at a constant rate of 14 litres per second how long will it take to just fill the tank? How much water will escape through the orifice during that period?*

Solution

From Eq. 9.46, we have

$$t = \frac{2A}{K^2}\left[Q\log_e\frac{Q-K\sqrt{H_2}}{Q-K\sqrt{H_1}} + K(\sqrt{H_2}-\sqrt{H_1})\right]$$

$$A = \frac{\pi}{4}\times(0.6)^2 = 0.2827$$

$$K = C_d\, a\sqrt{2g} = 0.6\times\frac{\pi}{4}(0.05)^2\times\sqrt{2\times9.81} = 0.00522$$

Q = 14 l/s = 0.014 m^3/s

H_1 = 0; and H_2 = 1.5 m

Thus by substitution, we have

$$t = \frac{2\times0.2827}{(0.005\,22)^2}$$

$$\times\left[0.014\log_e\left\{\frac{0.014-0.00522(1.5)^{½}}{0.014}\right\}+0.005\,22(1.5)^{½}\right]$$

$$= 44.55 \text{ s}$$

Total inflow = $(0.014 \times 44.55) = 0.6237$ m^3

Water contained in the tank

$$= \frac{\pi}{4}(0.6)^2\times1.5 = 0.4241 \text{ m}^3$$

$\therefore$ Water escaped through the orifice

$$= (0.6237 - 0.4241)$$

$$= 0.199\,6 \text{ m}^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 9.24. *A cylindrical tank is placed with its axis vertical and is provided with a circular orifice 40 mm in diameter in its bottom. Water flows into the tank at a uniform rate and is discharged through the orifice. It is found that it takes 107 seconds for the head in the tank to rise from 0.60 m to 0.76 m and takes 120 seconds for it to rise from 1.20 m to 1.28 m. Find the rate of inflow in cumec and the cross-sectional area of the tank. Take $C_d = 0.62$.*

Solution

In Eq. 9.46

$$K = C_d\, a \sqrt{2g}$$

$$= 0.62 \times \frac{\pi}{4}(0.04)^2 \times \sqrt{2 \times 9.81}$$

$$= 0.00345$$

Two values of (dh/dt) are given as indicated below.

For an average head

$$h = \frac{0.60 + 0.76}{2} = 0.68 \text{ m}$$

$$dh = 0.16 \text{ m}$$

and $dt = 107$ s

or $$\frac{dh}{dt} = \frac{0.16}{107}$$

For an average head

$$h = \frac{1.20 + 1.28}{2} = 1.24 \text{ m}$$

$$dh = 0.08 \text{ m}$$

and $dt = 120$s

or $$\frac{dh}{dt} = \frac{0.08}{120}$$

Since $$Adh = \left(Q - K\sqrt{h}\right) dt$$

$$\frac{dh}{dt} = \frac{Q - K\sqrt{h}}{A}$$

Thus by substitution, we get

$$\frac{0.16}{107} = \frac{1}{A}(Q - 0.00345\sqrt{0.68}) \quad \text{...(i)}$$

$$\frac{0.08}{120} = \frac{1}{A}(Q - 0.00345\sqrt{1.24}) \quad \text{...(ii)}$$

Solving Eqs (i) and (ii), we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = 0.0046 \text{ cumec}$$
$$A = 1.204 \text{ m}^2$$

Example 9.25. *A tank one metre square in plan area at all levels is provided with an orifice ($C_d = 0.677$) of area 0.01 m^2 in its bottom. The tank receives from the top a constant supply of water which would maintain a constant water level in the tank at a height of 4 m above the orifice. Calculate the rate of rise of the water level in the tank (a) when the tank is empty and water just enters the tank, and (b) when the head over the orifice is 2 m.*

Solution

When a constant head of 4 m is maintained, then inflow

$$= Q = \text{outflow through orifice}$$
$$= C_d\, a\, \sqrt{2gh}$$
$$= 0.677 \times (0.01) \times \sqrt{2 \times 9.81 \times 4}$$
$$= 0.06 \text{ m}^3/\text{s}$$

When the inflow and outflow are not balanced, then

$$Adh = (Q - q)\, dt$$

(a) When the tank is empty and the water just enters the tank, $q = 0$ as $h = 0$, and hence

$$\frac{dh}{dt} = \frac{Q}{A}$$
$$= \frac{0.06}{1 \times 1} = 0.06 \text{ m/s}$$

(b) When the head over the orifice is 2 m, then

$$q = 0.677 \times 0.01 \times \sqrt{2 \times 9.81 \times 2}$$
$$= 0.042\ 4 \text{ m}^3/\text{s}$$

$$\therefore \quad \frac{dh}{dt} = \frac{Q - q}{A}$$
$$= \frac{0.06 - 0.0424}{1 \times 1} = 0.0176 \text{ m/s}$$

Example 9.26. *A streamlined nozzle of diameter d is supplied at a constant head, the magnitude of the head being large compared to d. The nozzle discharges directly into the atmosphere and so shaped that the issuing jet is paralled at the nozzle exit. To increase the flow rate a shroud of diameter D is firmly secured to the nozzle as shown. The jet expands to fill the shroud and the shroud is long enough to ensure that the flow leaving it is steady and parallel.*

Determine what the diameter of the shroud should be to maximize the flow rate. Neglect shear stresses at the walls of the shroud.

Also obtain the expression for the maximum discharge.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

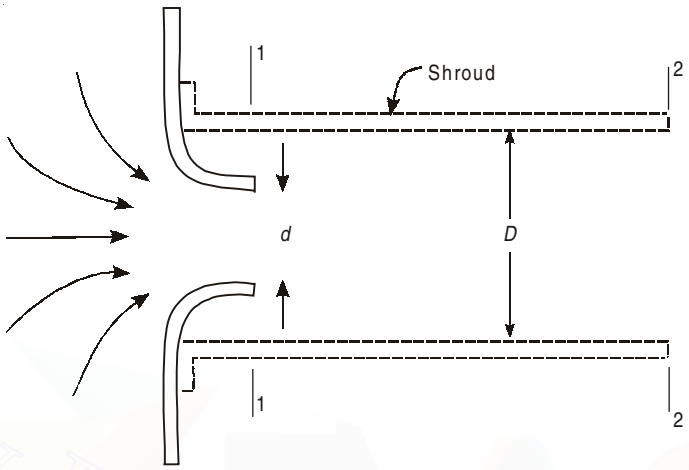


Figure Ex. 9.26

Solution

The head loss between sections 1–1 and 2–2 is due to sudden enlargement and is given by equation 9.25 as

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

If H is the constant head above the centre line of the nozzle, then applying Bernoulli's equation between the liquid surface in the supplying tank and section 2–2 at the exit end of the shroud, we get

$$H = \frac{V_2^2}{2g} + \frac{(V_1 - V_2)^2}{2g}$$

If Q is the discharge, then by continuity, we have

$$Q = \frac{\pi d^2}{4} \times V_1 = \frac{\pi D^2}{4} V_2$$

or $$V_1 = \frac{4Q}{\pi d^2}; \text{ and } V_2 = \frac{4Q}{\pi D^2}$$

$\therefore$ $$H = \frac{1}{2g}\left[\left(\frac{4Q}{\pi D^2}\right)^2 + \left\{\left(\frac{4Q}{\pi d^2}\right) - \left(\frac{4Q}{\pi D^2}\right)\right\}^2\right]$$

$$= \frac{8Q^2}{\pi^2 g}\left[\frac{1}{D^4} + \frac{1}{d^4} + \frac{1}{D^4} - \frac{2}{D^2 d^2}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$Q^2 = \frac{\pi^2 gH}{8}\left[\frac{D^4 d^4}{2d^4 - 2d^2D^2 + D^4}\right]$$

For Q to be maximum, $\frac{dQ}{dD} = 0$, which gives

$$D = \sqrt{2}\, d$$

Substituting this value of D in the equation for Q, the expression for maximum discharge Q_{max} is obtained as

$$Q_{max}^2 = \frac{\pi^2 gH}{8}\left[\frac{4d^8}{2d^4 - 4d^4 + 4d^4}\right]$$

or $$Q_{max} = \frac{\pi d^2}{2\sqrt{2}}\sqrt{2gH}$$

SUMMARY OF MAIN POINTS

1. An orifice is an opening having a closed perimeter, made in the walls or the bottom of a tank or a vessel containing fluid, through which the fluid may be discharged.

   According to the size, the orifices may be classified as small and large orifices. According to the shape, the orifices may be classified as circular, rectangular, square and triangular. According to the shape of the upstream edge the orifices may be classified as sharp-edged orifices and bell-mouthed orifices. According to the discharge conditions, the orifices may be classified as orifices discharging free and drowned or submerged orifices. The drowned or submerged orifices may be further classified as fully submerged orifices and partially submerged orifices.
2. A mouthpiece is a short tube of length not more than two to three times its diameter, which is fitted to a circular opening or orfice of the same diameter, provided in a tank or a vessel containing fluid, such that it is an extension of the orifice and through which also the fluid may be discharged.

   According to the shape the mouthpiece may be classified as cylindrical, convergent and convergent-divergent. According to the position the mouthpieces may be classified as external and internal mouthpieces. An internal mouthpiece is also called re-entrant or Borda's mouthpiece. According to the discharge conditions the mouthpieces may be classified as running full and running free mouthpieces. The running free condition of discharge may be developed only in the case of internal mouthpieces.
3. Theoretical velocity V_{th} of jet of liquid emerging from an orifice under a head h is given as

   $$V_{th} = \sqrt{2gh}$$
4. The various coefficients for an orifice are defined as

   (i) Coefficient of velocity

   $$C_v = \frac{\text{Actual velocity at vena contracta}}{\text{Theoretical velocity}}$$

   $$C_v = \sqrt{\frac{x^2}{4hy}}$$

   where x and y are the coordinates of a point in the jet measured from the vena contracta.

   (ii) Cofficient of contraction

   $$C_c = \frac{\text{Area of jet at vena contracta}}{\text{Area of orifice}}$$

   (iii) Cofficient of discharge

   $$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$C_d = C_c \times C_v$$

(iv) Coefficient of resistance

$$C_r = \frac{\text{Loss of kinetic energy}}{\text{Actual kinetic energy}}$$

$$C_r = \left(\frac{1}{C_v^2} - 1\right)$$

Depending on the shape and size of the orifice and on the head of liquid under which the flow takes place, C_c varies from 0.61 to 0.69, C_v varies from 0.95 to 0.99, and C_d varies from 0.61 to 0.65. For a small sharp edged circular orifice discharging water and other liquids of similar viscosity the average values of these coefficients may be taken as

C_c =0.64 or 0.65 ; C_v = 0.97 or 0.98 ; C_d = 0.62 or 0.63

6. Discharge through a small orifice of area a under a head h measured above the centre of the orifice is given by

$$Q = C_d a \sqrt{2gh}$$

7. Discharge through a large rectangular orifice is given by

$$Q = \frac{2}{3} C_d b \sqrt{2g} \left[H_2^{3/2} - H_1^{3/2}\right]$$

where

b = width of orifice ;
C_d = coefficient of discharge for the orifice, usually taken as 0.6;
H_1 = height of the liquid surface above the top edge of the orifice;
H_2 = height of the liquid surface above the bottom edge of the orifice; and
g = acceleration due to gravity.

8. Discharge through a totally submerged orifice is given by

$$Q = C_d a \sqrt{2g(H_1 - H_2)}$$

where

a = area of the orifice;
C_d = coefficient of discharge for the orifice ;
H_1 = height of the liquid surface on the upstream side above the centre of the orifice ;
H_2 = height of the liquid surface on the downstream side above the centre of the orifice; and
g = acceleration due to gravity.

9. Discharge through a partially submerged orifice is given by

$$Q = Q_1 + Q_2$$

where Q_1 and Q_2 are respectively the discharges through the free and the submerged portions of the orifice, which are given as

$$Q_1 = \frac{2}{3} C_{d_1} b \sqrt{2g} \left[H^{3/2} - H_2^{3/2}\right]$$

$$Q_2 = C_{d_2} b (H_1 - H) \sqrt{2gh}$$

where

b = width of the orifice;
H = difference between the liquid surfaces on the upstream and the downstream sides of the orifice
H_1 = height of the liquid surface on the upstream side above the bottom edge of the orifice;
H_2 = height of the liquid surface on the upstream side above the top edge of the orifice;
C_{d_1} = coefficient of discharge for the free portion of the orifice;
C_{d_2} = coefficient of discharge for the submerged portion of the orifice; and
g = acceleration due to gravity.

10. Discharge through a mouthpiece of area a under a head h measured above the centre of the mouthpiece is given by

$$Q = C_d a \sqrt{2gh}$$

(i) For external mouthpiece, C_d = 0.855
(ii) For internal mouthpiece,
(a) running full, C_d = 0.707
(b) running free, C_d =0.50
(iii) For convergent, divergent and convergent-divergent mouthpieces, C_d = 0.975 to 1.0

11. For external mouthpiece absoulte pressure head H_c at vena-contracta is given as

$$H_c = H_a - 0.89\, H$$

where

H_a = atmosphereic pressure head =10.3 m of water; and
H = height of liquid surface above the centre of the mouthpiece.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For internal mouthpiece absolute pressure head H_c at vena-contracta is given as

$$H_c = H_a - H$$

where H_a and H are same as indicated above.

12. For convergent-divergent mouthpiece the ratio of the area a at the outlet end the area a_c at the vena-contracta is given as

$$\frac{a}{a_c} = \sqrt{1+\frac{H_a - H_c}{H}}$$

where

H_a = atmospheric pressure head;
H_c = absoulute pressure head at vena-contracta; and
H = height of liquid surface above the centre of the mouthpiece.

13. Time of emptying a cylindrical tank with its axis vertical, through an orifice at its bottom is given by

$$t = \frac{2A\left(H_1^{1/2} - H_2^{1/2}\right)}{C_d a\sqrt{2g}}$$

where

H_1 = initial height of liquid surface above the centre of the orifice;
H_2 = final height of liquid surface above the centre of the orifice;
A = cross-sectional area of the tank;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

If the tank is to be completely emptied then time t is given by

$$t = \frac{2A\left(H_1^{1/2}\right)}{C_d a\sqrt{2g}} \qquad (\text{Since } H_2 = 0)$$

14. Time of emptying a cylindrical tank with its axis horizontal, through an orifice at its bottom is given by

$$t = \frac{4l}{3C_d a\sqrt{2g}}\left[(2R - H_2)^{3/2} - (2R - H_1)^{3/2}\right]$$

where

l = length of the tank;
R = radius of the tank;
H_1 = initial height of liquid surface above the centre of orifice;
H_2 = final height of liquid surface above the centre of orifice;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

15. Time of emptying a hemispherical tank through an orifice at its bottom is given by

$$t = \frac{2\pi}{C_d a\sqrt{2g}}\left[\frac{2}{3}R\left(H_1^{3/2} - H_2^{3/2}\right) - \frac{1}{5}\left(H_1^{5/2} - H_2^{5/2}\right)\right]$$

where

R = radius of the tank;
H_1 = initial height of liquid surface above the centre of the orifice;
H_2 = final height of liquid surface above the centre of the orifice;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

If the tank was full at the beginning and it is to be completely emptied then the time t is given by

$$t = \frac{14\pi R^{5/2}}{C_d a\sqrt{2g}}$$

(Since $H_1 = R$, and $H_2 = 0$)

16. Time required to reduce the difference between the liquid surfaces in the two tanks is given by

$$t = \frac{2A_1\left(H_1^{1/2} - H_2^{1/2}\right)}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2g}}$$

where

A_1 = cross-sectional area of tank from which the liquid flows;
A_2 = cross-sectional area of tank into which the liquid flows;
H_1 = initial difference between the liquid surfaces in the two tanks;
H_2 = final difference between the liquid surfaces in the two tanks;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

If both the tanks are of the same cross-sectional areas, i.e., $A_1 = A_2$, then

$$t = \frac{A_1\left(H_1^{1/2} - H_2^{1/2}\right)}{C_d a\sqrt{2g}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

17. Time required to empty a canallock is given by

$$t = \frac{2AH}{C_d a_2 \sqrt{2gh}}$$

Time required to fill a canal lock is given by

$$t = \frac{A(H+h)}{C_d a_1 \sqrt{2gh}}$$

where

A = cross-sectional area of the lock;
H = difference of water levels on the upstream reach and the downstream reach (also known as lift) ;
h = height of the water surface of the upper reach above the centre of the upper orifices;
a_1 = area of the upper orifices;
a_2 = area of the lower orifices;
C_d = coefficient of discharge for the orifices; and
g = acceleration due to gravity.

PROBLEMS

9.1 Define 'vena-contracta'. Explain how is it developed.

9.2 Define the various coefficients for an orifice.

9.3 Describe the different methods for the determination of the various coefficients for an orifice.

9.4 What is a mouthpiece? What is the advantage of providing a mouthpiece?

9.5 Show that discharge of water through a sharp-edged orifice shall be increased by about 38% if a short cylindrical mouthpiece of the same diameter is fitted to it on the outside of the tank. Take coefficient of contraction as 0.62 and neglect friction.

9.6 A jet of water issuing from an orifice 25 mm diameter under a constant head of 1.5 m falls 0.915 m vertically before it strikes the ground at a distance of 2.288 m measured horizontally from the vena-contracta. The discharge was found to be 102 litres per minute. Determine C_d, C_v and C_c for the orifice.

[*Ans.* 0.638; 0.976; 0.654]

9.7 A sharp-edged orifice of 50 mm diameter discharges water under a head of 4.5 m. Find the coefficient of discharge if the measured rate of flow is 11.76 litres per second. If there is an average pressure within the jet in the plane of the orifice of 39.044 kN/m² [0.398 kg(f)/cm²] above atmospheric pressure, calculate the coefficient of contraction. Neglect loss of energy due to friction. [*Ans.* 0.637; 0.677]

9.8 Oil of specific gravity 0.75 flows through a 80 mm diameter orifice whose coefficients of velocity and contraction are 0.95 and 0.65 respectively. At what pressure the air be pumped in the space above the oil surface so that the power in the jet may be 7.356 kW [10 hp] . Take height of oil above the centre of the orifice as equal to 2.7 m.

[*Ans.* 117.329 kN/m² {1.197 kg(f)/cm²}]

9.9 A vessel containing water has a vertical circular orifice 25 mm diameter which is at first plugged up. The tank is suspended in such a way that any displacing force can be accurately measured. On the removal of the plug the horizontal force required to keep it in place, applied opposite to the orifice is 16 N [1.631 kg (f)]. The discharge was found to be 141 litres per minute. The level of the water being maintained at a constant height of 2.745 m above the orifice. Determine C_c, C_v and C_d for the orifice. [*Ans.* 0.703; 0.928; 0.652]

9.10 Compensation water is to be discharged by two circular orifices under a constant head of 0.9 m measured to the centres of the orifices. What diameter will be required to give a discharge of 1510^6 litres per day? Assume $C_d = 0.66$.

[*Ans.* 0.2 m]

9.11 A large tank has a rectangular sharp-edged orifice 1 m broad and 0.75 m deep, the top edge of which is 0.45 m below the level of water in the tank. Find the quantity of water flowing through the orifice in cumec, if the coefficient of discharge is 0.62. [*Ans.* 1.854 cumec]

9.12 An orifice in the side of a large tank is rectangular in shape 1.2 m wide and 0.7 m deep. The water level on one side of the orifice is 1.2 m above the top edge and the water level on the other side of the orifice is 0.25 m below the top edge. Compute the discharge if $C_d = 0.62$.

[*Ans.* 2.734 m³/s]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

9.13 A horizontal boiler 2.4 m diameter and 7.5 m long contains water to a depth of 1.5 m. Find the time of emptying the boiler through a short vertical pipe 75 mm in diameter attached to the bottom of the boiler. Take $C_d = 0.8$.

[*Ans.* 0.508 *h*]

9.14 A tank has two indentical orifices, one vertically above the other and 1m apart, in one of its vertical sides. The water surface is 1.22 m above the higher orifice and is maintained at a constant level. It is found that the jets intercept each other at a horizontal distance of 2.65 m from the vena-contracta. Determine the C_v for the orifices.

[*Ans.* 0.805]

9.15 Water flows vertically downward through a 0.15 m diameter pipe at a rate of 50 litres per second. The pipe suddenly enlarges to 0.3 m diameter. Find the loss of head due to this sudden enlargement. Also find the loss of head if the same discharge passes vertically upward through the pipe, the coefficient of contraction being 0.62. If *A* is a point 0.3 m above the enlargement and *B* is a point 0.3 m below it, find the excess of pressure head at *B* over that at *A* in both cases.

[*Ans.* 0.23 m; 0.153 m; 0.7524 m; 1.1354 m]

9.16 A vertical cylindrical tank is 0.6 m in diameter and water flows in it at a constant rate of 0.0042 m^3/s. There is a sharp-edged orifice in the bottom having a cross-sectional area of 12.9 m^2. If $C_d = 0.6$, how long will it take for the depth in the tank to increase from 0.3 m to 0.6 m.

[*Ans.* 45 s]

9.17 A Borda's mouthpiece of 0.15 m diameter discharges water under a head of 3 m. Calculate (a) the discharge; and (b) the diameter of the jet at the vena-contracta for both conditions of flow.

[*Ans.* (a) Running free 68 *l*/s; Running full 95.85 *l*/s; (b) 0.106 1 to 0.108 2 m]

9.18 A convergent-divergent nozzle is fitted into the side of a tank containing water and, under a constant head *H* m above the centre line of the nozzle, discharges into the atmosphere. Obtain an expression for the ratio of the exit area of the nozzle to the area of the throat for maximum discharge, making the following assumptions: (i) the barometric pressure is 10.3 m of water, (ii) separation will occur at an absolute pressure head of 1.83 m of water, (iii) the only hydraulic losses occur in the divergent portion of the nozzle and amount to 25 per cent of the head lost at a sudden enlargement for the same change of area.

If the supply head *H* is 2.75 m and the throat diameter is 0.05 m, calculate the maximum discharge.

$$\left[\frac{\text{Exit area}}{\text{Throat area}} = \frac{4\{\sqrt{(H+8.47)(H-2.1175)}\}-(H+8.47)}{(3H-8.47)}\right]$$

[*Ans.* $Q = 0.0291\ m^3/s$]

9.19 Find the time required to empty a swimming pool through an opening provided in the bottom of the deep end. Depth of water at the deep end is 2.4 m and at the shallow end it is 0.9 m. Length of the pool is 27 m and the width of the pool is 9 m. Area of the opening is 9.2 m^2 and its $C_d = 0.6$. [*Ans.* 11.166 min]

9.20 A reservoir is circular in plan, the diameter of the top water level is 90 m. At a depth of 1.5 m the diameter is 75 m. The mouth of the outlet pipe which is 0.6 m in diameter is 3.6 m below top water level. How long will it take to lower the depth of the water level by 1.5 m. Take $C_d = 0.87$. [*Ans.* 72.91 min]

9.21 A cylindrical tank 12 m high, containing water upto brink is completely emptied through a hole located in the bottom in 8.4 minutes. How long would it take for the water level to drop from the top of the tank when full to 4m from the top. [*Ans.* 92.49 s]

9.22 A tank has upper cylindrical portion of 3m diameter and 4 m high with a hemispherical base. If the tank is full of water, determine the time taken to empty it through an orifice 0.1 m diameter at its bottom. Take $C_d = 0.62$.

[*Ans.* 18.48 min]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Flow Over Notches and Weirs

Chapter 10

10.1 INTRODUCTION

A *notch* may be defined as an opening provided in the side of a tank (or vessel) such that the liquid surface in the tank is below the top edge of the opening. Notches made of metallic plates are also provided in narrow channels (particularly in laboratory channels) in order to measure the rate of flow of liquid. As such in general notches are used for measuring the rate of flow of liquid from a tank or in a channel.

A *weir* is the name given to a concrete or masonry structure built across a river (or stream) in order to raise the level of water on the upstream side and to allow the excess water to flow over its entire length to the downstream side. Thus a weir is similar to a small dam constructed across a river, with the difference that whereas in the case of a dam excess water flows to the downstream side, only through a small portion called spillway, the same in the case of a weir flows over its entire length. Weirs may also be used for measuring the rate of flow of water in rivers or streams.

The sheet of water flowing through a notch or a weir is known as the *nappe* (French term meaning sheet) or *vein*. The bottom edge of a notch or the top of a weir over which the water flows is known as the *sill* or *crest*, and its height above the bottom of the tank or channel is known as the *crest height*.

10.2 CLASSIFICATION OF NOTCHES AND WEIRS

The notches are usually classified according to the shape of the opening as *rectangular notch, triangular notch* (*or V-notch*), *trapezoidal notch, parabolic notch* and *stepped notch.* The edges of all these notches are bevelled on the downstream side so as to have sharp-edged sides and crest, resulting in minimum contact with the flowing liquid. Since the liquid surface is always below the top edge of the notch, a notch is usually provided with only a crest and sides with no top edge.

Notches may also be classified according to the effect of the sides on the nappe emerging from a notch, as *notch with end contraction* and *notch without end contraction* or *suppressed notch.* If the sides of a notch cause the contraction of nappe, then it is said to be notch with end contraction (or contracted notch). On the other hand if there is no contraction of the nappe due to the sides (or in other words the end contractions are suppressed) then it is known as a notch without end contraction. Notches provided in the sides of tanks or vessels are essentially the notches with end contraction. However, in a channel if the crest length of the notch is less than the width of the channel then it is a notch with end contraction. But if the crest length of the notch is equal to the width of the channel then it is a notch without end contraction or a suppressed notch.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Weirs may be classified according to the shape of the opening, the shape of the crest, the effect of the sides on the issuing nappe and the discharge conditions. According to the shape of the opening, the weirs may be classified as *rectangular, triangular and trapezoidal weirs.* A particular type of a trapezoidal weir is known as *Cipolletti weir.* According to the shape of the crest, the weirs may be classified as *thin-plate or sharp-edged weir, narrow-crested weir, broad-crested weir* and *Ogee shaped weir.* A sharp-edged weir may be formed by means of a metallic plate which is embedded in the concrete or masonry and therefore it is same as a notch of the similar shape. Alike notches weirs may also be classified according to the effect of the sides on the issuing nappe as *weir with end contraction* and *weir without end contraction.* If water is allowed to flow over a portion of the weir so that the crest length of the weir over which water flows is less than the width of the channel then the nappe emerging from the weir is affected by the sides and it gets contracted. Such a weir is known as weir with end contraction (or *contracted weir).* However, if water is allowed to flow over the entire length of the weir so that the crest length of the weir is equal to the width of the channel, then the nappe emerging from the weir is not affected by the sides and therefore it does not undergo contraction. The weir is then known as weir without end contraction (or *suppressed weir* because the end contractions are suppressed). In some cases on the weir crest, piers are constructed to support a foot-bridge, on account of which the effective crest length of the weir over which water flows is reduced. Moreover, the construction of the piers on the weir crest results in dividing the entire weir into a number of small weirs with end contraction (due to the piers). According to the discharge conditions weirs may be classified as *freely discharging weir* and *submerged* (or *drowned*) *weir.* If the water level on the downstream of the weir is well below the weir crest then the nappe emerges freely in the atmosphere and it is known as freely discharging weir. On the other hand if the water level on the downstream of the weir is above the crest of the weir then it is known as submerged weir.

There exists a considerable similarity between the pattern of flow over a sharp (or narrow) crested weir and a notch of the same shape. As such the same expressions as derived below may be adopted for computing the rate of flow of water over a sharp (or narrow) crested weir and a notch of the same shape. Moreover, on account of the similarity a notch is more often termed as sharp-crested weir.

10.3 FLOW OVER A RECTANGULAR SHARP-CRESTED WEIR OR NOTCH

Consider a rectangular sharp-crested weir or notch provided in a channel carrying water as shown in Fig. 10.1. Let L be the length of the crest of the weir or notch and H be the height of the water surface above its crest, which is known as the head causing the flow over the weir or notch. As water flows over weir or notch, the surface of water over the crest and immediately upstream of it becomes curved. The head H above the crest is therefore measured at a certain distance upstream of the weir or notch where the water surface may be assumed to be unaffected by the curvature effect. According to IS: 9108–1979,* the head H above the crest should be measured on the upstream of the weir or notch at a distance of 4 to 5 times the maximum head (H_{max}) above the crest.

For computing the discharge of water flowing over the weir or notch consider an elementary horizontal strip of water of thickness dh and length L at depth h below the water surface as shown in Fig. 10.1.

Area of strip is ($L \times dh$) and the theoretical velocity of the water flowing through the strip will be $\sqrt{2gh}$. Thus if dQ is the discharge through the strip, then

IS : 9108–1979 provides detailed specifications for thin plate weirs or notches.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$dQ = C_d \times L \times dh \times \sqrt{2gh}$$

where C_d is the coefficient of discharge. The total discharge Q for the entire weir or notch may be detrmined by integrating the above expression within limits 0 to H. Thus

$$Q = \int_0^H C_d L \sqrt{2gh}\, dh$$

Assuming the coefficient of discharge C_d to be constant for the entire weir or notch we obtained

$$Q = \frac{2}{3} C_d \sqrt{2g}\, LH^{3/2} \qquad \ldots(10.1)$$

In the above equation for the discharge over the weir or notch the *velocity of approach* has not been taken into account. The velocity of approach may be defined as the velocity with which water approaches or reaches the weir or notch before it passes over it. Thus if V_a is the velocity of approach, then the water flowing over the weir or notch possesses an additional head h_a equal to $\left(\frac{V_a^2}{2g}\right)$ due to the velocity of approach. Thus the limits of integration for the above equation become h_a and $(H + h_a)$ instead of 0 to H. The discharge Q passing over the weir or notch is then given by

$$Q = \int_{h_a}^{(H+h_a)} C_d L \sqrt{2gh}\, dh$$

or

$$Q = \frac{2}{3} C_d \sqrt{2g} L \left[(H + h_a)^{3/2} - h_a^{3/2} \right] \qquad \ldots(10.2)$$

Equation 10.2 gives the discharge of water flowing over a rectangular weir or notch when the velocity of approach is taken into account. The head $(H + h_a)$ is usually known as *still water head.*

Equations 10.1 and 10.2 are applicable to a suppressed rectangular weir or notch only, for which the crest length is equal to the width of the channel. For a rectangular weir or notch with end contraction these equations are to be modified as indicated below, in order to take into account the effect of the end contraction on the discharge over such a weir or notch. It has been indicated by J.B. Francis on the basis of his experiments that the end contraction has the effect of decreasing the effective length of the crest of the weir or notch, due to the contraction of the nappe, which results in decreasing the discharge. Further it has been proposed by Francis that the amount by which the crest length is reduced depends on the head H and for each end contraction the reduction in the crest length may be taken to be equal to $(0.1\,H)$ or $(H/10)$. Thus if there are n end contractions for a weir or notch of the crest length L then the effective length of the crest of the weir or notch will be $(L - 0_1\, n\, H)$. The discharge Q of water flowing over a weir or notch with n end contractions when the velocity of approach is notconsidered, is then given by

$$Q = \frac{3}{2} C_d \sqrt{2g}\, (L - 0.1\, n\, H)^{3/2} \qquad \ldots(10.3)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

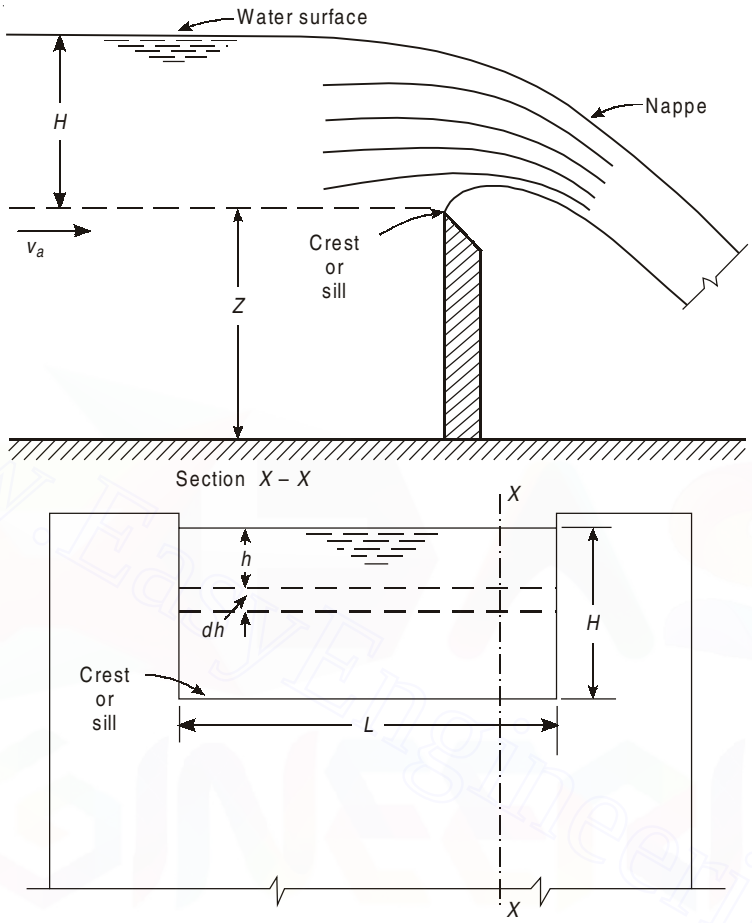


Figure10.1 *Flow over rectangular sharp crested weir or notch*

However, for such a weir or notch if the velocity of approach is taken into account, then the discharge Q is given by

$$Q = \frac{2}{3} C_d \sqrt{2g} (L - 0.1\, n\, H_1)\left(H_1^{3/2} - h_a^{3/2}\right) \quad \ldots(10.4)$$

in which $$H_1 = (H + h_a) = \left[H + \left(V_a^2 / 2g\right)\right]$$

The mean value of the velocity of approach to be used in Eqs 10.2 and 10.4 may be determined by dividing the discharge Q by the area of the flow section of the channel. Thus if the height of the crest of the weir or notch above the bottom of the channel is Z, then the depth of the flow of water in the channel is $(H + Z)$. Now if the width of the channel is equal to B then the area of the flow section of the channel becomes $B(H + Z)$ and the velocity of approach is given by

$$V_a = \frac{Q}{B(H + Z)} \quad \ldots(10.5)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, in case of suppressed weir or notch, the width of the channel equals the crest length L, in which case

$$V_a = \frac{Q}{L(H+Z)} \qquad ...(10.5a)$$

It may be observed from Eq.. 10.5 that for the determination of the velocity of approach the discharge Q of water flowing in the channel should be known. Since Q is not known initially, an approximate value of Q is computed by neglecting the velocity of approach and using Eq. 10.1 or Eq. 10.3, depending on whether the weir or notch is suppressed or not. Using this approximate value of Q, an approximate value of V_a is then computed which is substituted in Eq. 10.2 or Eq. 10.4 to get another value of discharge Q. The new value of Q obtained will be a better approximation than its previously calculated value. The process is, therefore, repeated until the final value of the discharge Q obtained is within 1% of the preceding value of the calculated discharge.

10.4 CALIBRATION OF RECTANGULAR WEIR OR NOTCH

A rectangular weir or notch is generally used for measuring the quantity of water flowing in a channel, in which case it must be calibrated previously. For this purpose the weir or notch which is to be calibrated is installed in a laboratory channel and the discharge Q and head H are measured. The discharge Q may be expressed as

$$Q = kH^n$$

By taking log of both sides of this equation

$$\log Q = \log k + n \log H \qquad ... (10.6)$$

which is a straight line law in terms of log Q and log H.

From the experimental data a plot of log Q as ordinate and log H as abscissa may be prepared, which will be a straight line in accordance with Eq. 10.6. The values of k and n may be determined from the plot as described below.

For $H = 1$, $\log H = 0$ then from Eq. 10.6, $\log Q = \log k$, hence k can be evaluated. Further from Eq. 10.6,

$$n = \frac{\log Q - \log k}{\log H}$$

Therefore by choosing any convenient point on the straight line, and substituting the corresponding values of log Q and log H in the above equation the value of n can be calculated.

An alternative method is to assume $Q = kH^{3/2}$ where $k = \frac{2}{3} C_d \left(\sqrt{2g}\right) L$. From the experimental data a plot of Q and $H^{3/2}$ may be prepared which is accordance with the above expression will be a straight line, the slope of which will give the value of k. However, in this method a perfect straight line may not be obtained because the value of C_d though assumed to be constant, varies slightly with the head. From the value of k evaluated by either of the methods the coefficient of discharge C_d for the rectangular weir or notch may be determined.

The value of C_d for a rectangular weir or notch is approximately 0.6. However the value of C_d depends on head H and the ratio (H/Z) where Z is the height of the crest above the bed of the channel. Further if the head is small the value of C_d is also affected by viscosity and surface tension.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

10.5 EMPIRICAL FOMULA FOR DISCHARGE OVER RECTANGULAR WEIRS

A rectangular weir is frequently used for measuring the rate of flow of water in channels. As such various investigators have conducted experiments with rectangular weirs and on the basis of the experimental results they have proposed a number of empirical formulae for computing the discharge over rectangular weirs. Some of these formulae which are commonly used are described below:

1. Francis' Formula. It is one of the most commonly used formula for computing the discharge over sharp or narrow crested weirs with or without end contraction. This formula was proposed by J. B. Francis on the basis of his extensive series of experiments with rectangular weirs during the years 1848 to 1852 at Lowell, Massachusetts (USA). The length of the crest varied from 1.07 m to 5.19 m, but in most of the experiments the crest length was 3.05 m. The head H varied from 0.18 m to 0.49 m. On the basis of the results obtained from more than eighty experiments. Francis proposed the following formulae for the discharge over rectangular weir.

When the velocity of approach is not considered

$$Q = 1.84\left(L - 0.1\ nH\right)H^{3/2} \quad \text{...(10.7)}$$

With the velocity of approach taken into account

$$Q = 1.84\left(L - 0.1\ nH_1\right)\left(H_1^{3/2} - h_a^{3/2}\right) \quad \text{...(10.8)}$$

where $H_1 = (H + h_a) = \left(H + \frac{V_a^2}{2g}\right)$, H is the height of water surface above the crest of the weir, and n is the number of end contractions.

By comparing the above noted Francis' formulae with the Eqs. 10.3 and 10.4, we obtained

$$\frac{2}{3}C_d\sqrt{2g} = 1.84$$

from which $C_d = 0.623$

i.e., according to Francis' formula the value of C_d for a rectangular weir is 0.623.

If the end contractions are suppressed i.e., if the crest length of the weir over which water flows is equal to the width of the channel, then $n = 0$ and Eq. 10.7 and 10.8 reduce to

$$Q = 1.84LH^{3/2} \quad \text{... (10.9)}$$

and

$$Q = 1.84L\left(H_1^{3/2} - h_a^{3/2}\right) \quad \text{... (10.10)}$$

Figure 10.2 shows the typical cases of flow over rectangular weirs for which the different values of n as indicated thereon may be adopted.

2. Bazin's Formula. In 1886 H. Bazin of France undertook a series of experiments with rectangular weirs. On the basis of the results of these experiments Bazin proposed the following formulae for the discharge over rectangular weir.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

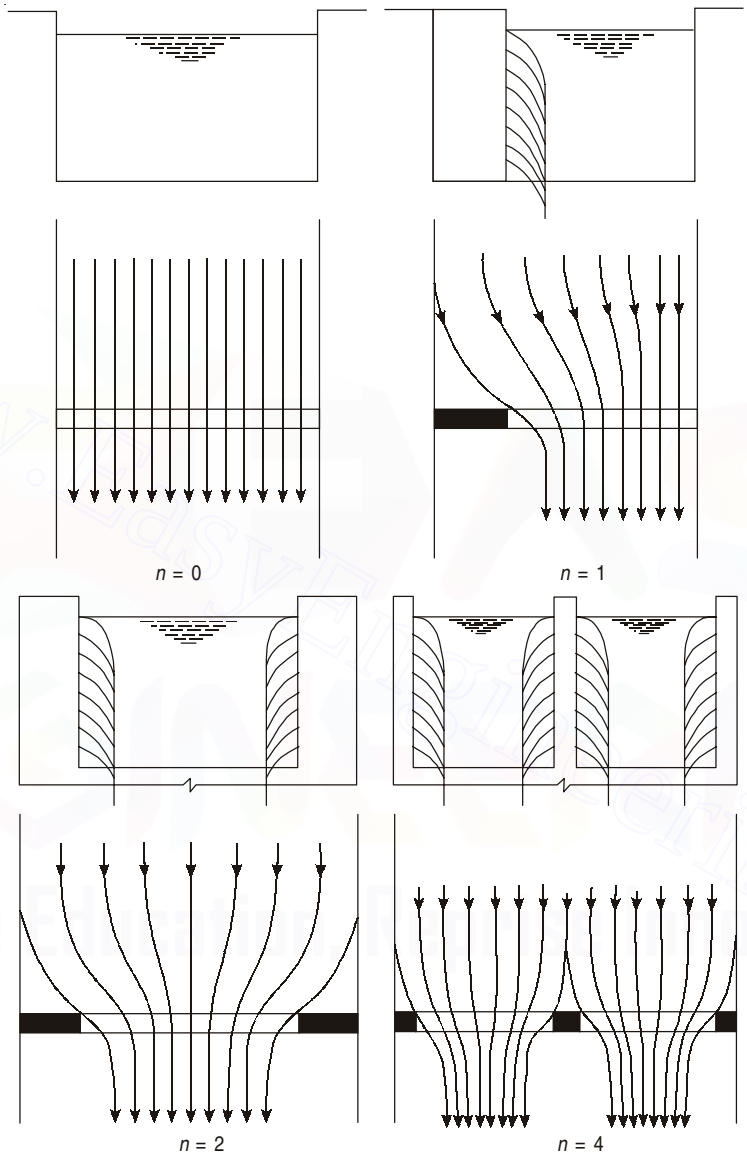


Figure 10.2 *End contractions for rectangular weirs*

For the velocity of approach not considered.

$$Q = m\sqrt{2g}\, LH^{3/2} \qquad ...(10.11)$$

in which according to Bazin the value of the coefficient m is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$m = \left(0.405 + \frac{0.003}{H}\right)$$

With the velocity of approach taken into account

$$Q = m_1 \sqrt{2g}\ LH_1^{3/2} \quad \ldots (10.12)$$

in which the coefficient m_1 is given by

$$m_1 = \left(0.405 + \frac{0.003}{H_1}\right)$$

and H_1, which is called the still water head is given by

$$H_1 = \left(H + \alpha \frac{V_a^2}{2g}\right)$$

where α is a constant, the mean value of which has been given by Bazin as 1.6.

3. Rehbock's Formula. In 1929 T. Rehbock on the basis of his experiments with suppressed rectangular weirs proposed the following empirical formula for the coefficient of discharge C_d for a suppressed weir

$$C_d = \left(0.605 + 0.08\frac{H}{Z} + \frac{0.001}{H}\right) \quad \ldots (10.13)$$

where H is the head in metres and Z is the crest height in metres. The Rehbock's formula for the discharge over rectangular weir then becomes

$$Q = \frac{2}{3}\left(0.605 + 0.08\frac{H}{Z} + \frac{0.001}{H}\right)\sqrt{2g}\ LH^{3/2} \quad \ldots (10.14)$$

10.6 VENTILATION OF WEIRS

In the case of suppressed weir, since the crest-length is equal to the width of the channel, the nappe emerging from the weir touches the side walls of the channel. On account of this, air is trapped in the space between the side walls of the channel, the falling nappe, the weir and the bottom of the channel as shown in Fig. 10.3. This air is gradually carried away with the flowing water, thereby reducing the pressure in the space below the nappe, which may even become negative i.e., below atmospheric pressure or vacuum pressure. The negative pressure so developed in the space draws the lower nappe more and more towards the downstream surface of the weir as shown in Fig. 10.3 (b). Such a nappe is known as *depressed nappe*, which results in drawing more water thereby increasing the actual discharge over theweir. With a further withdrawal of the air from the space when no air is left below the nappe, the nappe may adhere to the downstream surface of the weir as shown in Fig. 10.3 (c). Such a nappe is called *adhering or clinging nappe*, which further results in drawing more water, thereby further increasing the actual discharge over the weir.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

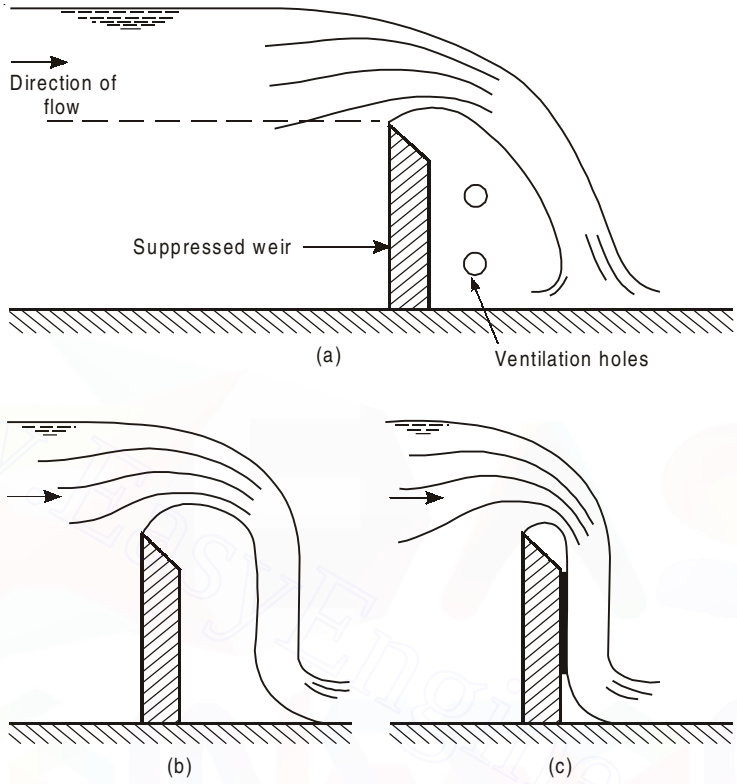


Figure 10.3 *Different types of nappe over a suppressed weir* : (*a*) *Free nappe;* (*b*) *Depressed nappe;* (*c*) *Adhering or clinging nappe*

The various formulae both analytical and empirical as indicated earlier for computing the discharge over suppressed rectangular weirs are based on the assumption that in the space below the emerging nappe there is atmospheric presure. Such a nappe is known as *free nappe* [Fig. 10.3 (a)]. A free nappe may be developed for a suppressed weir if the pressure in the space below the nappe is always atmospheric and it is not allowed to be reduced. For this purpose just downstream of the weir, holes are made in the side walls of the channel in the space below the lower nappe, so that this region is connected to the atmospheric air outside and the air which is carried away with the flowing water from this region is recouped from the atmosphere through these holes. These holes are thus called ventilation holes and the weirs are called ventilated weirs. Experimentally it has been found that the minimum cross-sectional area of the ventilation holes may be about 0.5 per cent of $(L \times Z_1)$ where L is the crest length and Z_1 is the depth of water surface on the downstream side below the crest of the weir. Generally two 25 mm diameter holes are sufficient for discharges upto 100 litres per second. However, if a suppressed weir is not properly ventilated then as stated earlier a partial withdrawal of the air takes place which may lead to the development of a depressed nappe. Moreover, if a suppressed weir is not ventilated then the air from the space below the nappe may be completely removed which may lead to the development of an adhering or clinging nappe. It has been observed in actual practice that

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the discharge over a weir with depressed nappe is about 6 to 7% more than that obtained with a free nappe. Further the discharge over a weir with adhering nappe is about 25 to 39% more than that obtained with a free nappe.

When a suppressed weir is used as measuring device, then it should be properly ventilated so that a free nappe is developed and the actual discharge of water flowing over the weir is same as that obtained by using any of the formulae indicated in the preceding section. On the other hand if a weir with either depressed or adhering nappe is used for the discharge measurement then the actual discharge of water flowing over the weir being more, the use of any of the formulae for the computation of the discharge will give wrong results.

10.7 FLOW OVER A TRIANGULAR WEIR (V-NOTCH WEIR) OR TRIANGULAR NOTCH (V-NOTCH)

A triangular weir is an ordinary weir which is having a triangular or V-shaped opening or notch provided in its body, so that water is discharged through this opening only. Thus a triangular weir and a triangular notch are the same and the discharge of water flowing over a triangular weir or notch may be computed by using the same expression as described below. Generally a triangular weir or notch is preferred to a rectangular weir or notch for measuring the low discharges. This is so because with low discharges if rectangular weirs or notches are used for measuring the discharges then the head over the crest of the weir or notch may be so small that it may not be possible to measure it accurately. For such cases a triangular weir or notch may be used because the crest length for a triangular weir or notch is equal to zero and therefore even for a low discharge the head over the crest is fairly large which can be measured more accurately.

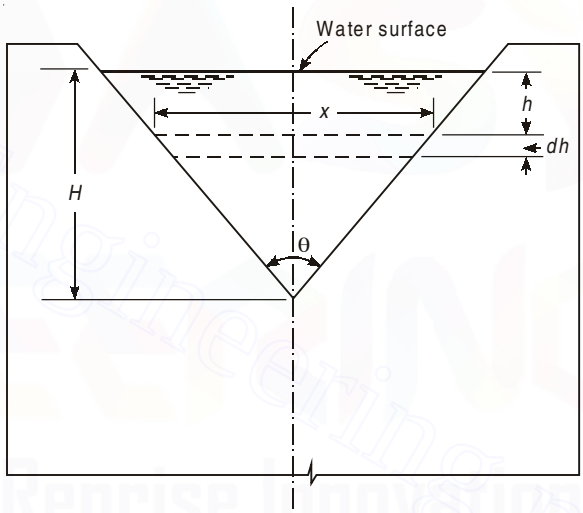


Figure 10.4 *Flow over triangular weir or notch*

Figure 10.4 shows a triangular weir or notch with a vertex angle equal to θ. Let H be the head above the crest of the weir. Consider a horizontal elementary strip of thickness dh at a depth h below the water surface. If x is the width of the strip then

$$\frac{x}{2(H-h)} = \tan\frac{\theta}{2}; \text{ or } x = 2(H-h)\tan\frac{\theta}{2}$$

The area of the strip is $(x\,dh)$ or $[2(H-h)(\tan\theta/2)dh]$ and the theoretical velocity of water flowing through the strip will be $\sqrt{2gh}$.

Thus if dQ is the discharge through the strip then

$$dQ = C_d \times 2(H-h)(\tan\theta/2)dh \times \sqrt{2gh}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where C_d is the coefficient of discharge. The total discharge Q for the entire triangular weir or notch may be determined by integrating the above expression within limits 0 to H. Thus

$$Q = \int_0^H C_d \times 2(H-h)\tan\frac{\theta}{2}\sqrt{2gh}\ dh$$

Assuming the coefficient of discharge C_d to be constant for the entire weir or notch, we obtain

$$Q = 2C_d\sqrt{2g}\tan\frac{\theta}{2}\int_0^H (H-h)h^{1/2}dh$$

or
$$Q = 2C_d\sqrt{2g}\tan\frac{\theta}{2}\left[\frac{2}{3}Hh^{3/2} - \frac{2}{5}h^{5/2}\right]_0^H$$

or
$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2} \quad ...(10.15)$$

If the vertex angle θ equals $90°$ then for a right-angled triangular weir or notch

$$Q = \frac{8}{15}C_d\sqrt{2g}\ H^{5/2} \quad ...(10.16)$$

since $\theta/2 = 45°$ and $\tan\theta/2 = 1$. Further for a right angled weir or notch if C_d is assumed to be 0.6, then Eq. 10.16 becomes

$$Q = 1.418H^{5/2} \quad ...(10.17)$$

It is thus observed that for a right-angled triangular weir or notch the expression for the disharge is very much simplified. However, for any triangular weir or notch since the vertex angle θ is constant, the equation for the discharge may be expressed as

$$Q = KH^{5/2} \quad ...(10.18)$$

where K is a constant for the weir or notch given by

$$K = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2} \quad ...(10.19)$$

The value of K will, however, depend on the vertex angle θ and the coefficient of discharge C_d for the triangular weir or notch.

In deriving Eq. 10.15 for the discharge over a triangular weir or notch the velocity of approach has been neglected. In general for a triangular weir or notch the cross-section area of the approach channel is usually so much greater than that of the notch that the velocity of approach may be neglected. However, if the velocity of approach is to be taken into account then Eq. 10.15 for discharge Q over a triangular weir or notch may be modified as

$$Q = \frac{8}{15}C_d\sqrt{2g}\ \tan\frac{\theta}{2}\left[(H+h_a)^{5/2} - h_a^{5/2}\right] \quad ...(10.20)$$

where h_a is the head due to velocity of approach V_a and is equal to $\left(V_a^2/2g\right)$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Advantages of Triangular Weir over Rectangular Weir. A triangular weir or notch has several advantages over a rectangular weir or notch which are as noted below:

(1) The nappe emerging from a triangular weir or notch has the same shape for all the heads. As such the value of the coefficient of discharge for a triangular weir or notch is fairly constant for all the heads. On the other hand for a rectangular weir or notch the shape of the nappe is affected by the head and therefore the coefficient of discharge varies with the head.

(2) For measuring low discharges a triangular weir or notch is very useful. This is because with a triangular weir or notch even with a low discharge the head over the crest is comparatively large which can be measured accurately. However if a rectangular weir is used to measure low discharge, the head over the crest may be very small which may not be measured accurately. Moreover, when the head over th crest is small, the flow is markedly affected by surface tension and viscosity which may also lead to inaccurate measurement of head.

(3) For a right-angled triangular weir or notch the expression for the computation of the discharge is very much simplified.

(4) In most of the cases of flow over a triangular weir or notch the velocity of approach may be neglected without introducing an appreciable error.

(5) Ventilation of a triangular weir is not necessary.

On account of the above mentioned advantages a triangular weir or notch is preferred to a rectangular weir or notch for measuring the discharges. For laboratory channels since the discharges are usually low, a right-angled triangular weir or notch is preferred to other types. The calibration of triangular weirs or notches may also be carried out in the same manner as in the case of rectangular weirs or notches as described in Section 10.4.

10.8 FLOW OVER A TRAPEZOIDAL WEIR OR NOTCH

As shown in Fig. 10.5 a trapezoidal weir or notch is a combination of a rectangular and a triangular weir or notch. As such the discharge over such a weir or notch may be determined by adding the discharges over the two different types. Thus if L is the crest length of a trapezoidal weir or notch and $\theta/2$ is the angle of inclination of its sides with the vertical then for such a weir

$$Q = \left[\frac{2}{3}C_{d_1}\sqrt{2g}\ LH^{3/2} + \frac{8}{15}C_{d_2}\sqrt{2g}\ \tan\frac{\theta}{2}H^{5/2}\right] \quad ...(10.21)$$

where C_{d_1} and C_{d_2} are respectively the coefficients of discharge for the rectangular and the triangular portions of the trapezoidal weir or notch. However, if for whole of the trapezoidal weir or notch the coefficient of discharge is assumed to be C_d then the expression for the discharge becomes

$$Q = C_d\sqrt{2g}\ H^{3/2}\left[\frac{2}{3}L + \frac{8}{15}H\tan\frac{\theta}{2}\right] \quad ...(10.22)$$

Cipolletti Weir (or Notch). A Cipolletti weir is particular type of trapezoidal weir, the sloping sides of which have an inclination of 1 horizontal to 4 vertical, $\left(\text{i.e., } \frac{\theta}{2} = 14°\right)$. This weir was invented by an Italian engineer Cipolletti in 1887. As explained below the slope of 1 in 4 provided for the sides of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

this weir results in making the decrease in the discharge over a rectangular weir due to two end contractions just equal to the increase in the discharge trough the two triangular portions. So that

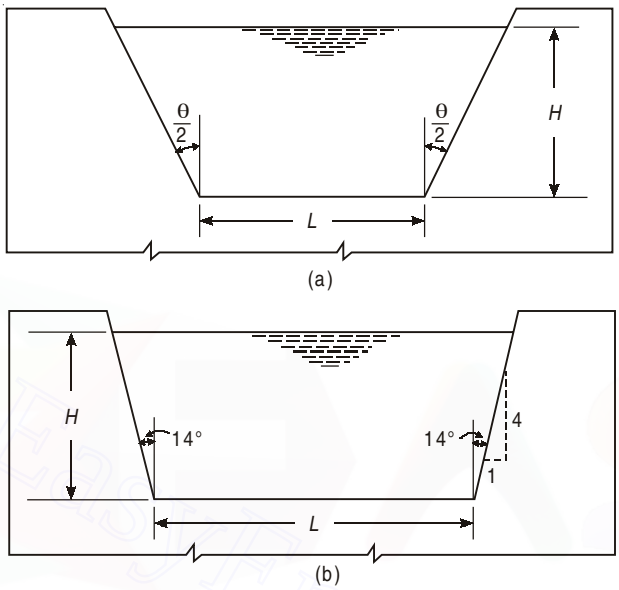


Figure 10.5 (*a*) *Trapezoidal weir;* (*b*) *Cipolletti weir*

the discharge over a Cipolletti weir may be computed by using the formula for a suppressed rectangular weir.

As in the case of a trapezoidal weir, for a Cipolletti weir also the discharge is given by

$$Q = (Q_1 + Q_2)$$

where Q_1 is the discharge through the rectangular portion and Q_2 is the discharge through the two triangular portions on either side. For a rectangular weir with two end contractions

$$Q_1 = \frac{2}{3} C_d \sqrt{2g}\,(L - 0.2H) H^{3/2}$$

and for a triangular weir

$$Q_2 = \frac{8}{15} C_d \sqrt{2g}\, \tan\frac{\theta}{2} H^{5/2}$$

For $\tan\frac{\theta}{2} = \frac{1}{4}$

$$Q_2 = \frac{2}{15} C_d \sqrt{2g}\, H^{5/2}$$

Thus adding the two, the discharge for the Cipolletti weir is given by

$$Q = \frac{2}{3} C_d \sqrt{2g}\,(L - 0.2H) H^{3/2} + \frac{2}{15} C_d \sqrt{2g}\, H^{5/2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$Q = \frac{2}{3} C_d \sqrt{2g}\, H^{3/2} \left[L - \frac{H}{5} + \frac{H}{5} \right]$$

or $$Q = \frac{2}{3} C_d \sqrt{2g}\, LH^{3/2} \quad ...(10.23)$$

which is same as the Equation for the discharge over a suppressed rectangular weir. On the basis of his own experiments and those of Francis, Cipolletti proposed the following equation for the discharge over a Cipolletti weir

$$Q = 1.86 LH^{3/2} \quad ...(10.24)$$

By comparing Eqs. 10.23 and 10.24 the value of the coefficient of discharge C_d for a Cipolletti weir is obtained as 0.632. Again Eq. 10.24 is applicable only if the velocity of approach is neglected. However, if the velocity of approach V_a is to be considered then Eq. 10.23 may be modified as

$$Q = 1.86 L \left(H_1^{3/2} - h_a^{3/2} \right) \quad ...(10.25)$$

where $H_1 = (H + h_a) = H + \left(V_a^2 / 2g \right)$.

10.9 TIME REQUIRED TO EMPTY A RESERVOIR WITH RECTANGULAR WEIR

Consider a reservoir of uniform cross-sectional area A, which is provided with a rectangular weir or notch in one of its sides. Let L be the crest length of the weir or notch and C_d be its coefficient of discharge. It is required to determine the time taken to reduce the level of liquid in the reservoir from H_1 to H_2 above the crest of the weir or notch.

Let at any instant the height of the liquid surface above the crest of the weir or notch be h, and in a small time dt let the liquid surface in the reservoir be lowered by dh. Now if Q is the discharge of liquid flowing over the weir or notch during the interval of time dt, then

$$-Adh = Qdt$$

But $$Q = \frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2}$$

Thus

$$-Adh = \frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2} dt$$

or $$dt = -\frac{Adh}{\frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2}}$$

By integrating both sides of the above equation, the time t required to lower the liquid surface from H_1 to H_2 above the crest of the weir or notch is obtained as

$$\int_0^t dt = t = \int_{H_1}^{H_2} -\frac{Adh}{\frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$t = -\frac{A}{\frac{2}{3}C_d\sqrt{2g}\,L}\left[-2h^{-1/2}\right]_{H_1}^{H_2}$$

or $$t = \frac{2A}{\frac{2}{3}C_d\sqrt{2g}\,L}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right] \quad ...(10.26)$$

If Bazin's formula is used for the computation of discharge then Eq. 10.26 becomes

$$t = \frac{2A}{m\sqrt{2g}}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right] \quad ...(10.27)$$

Similarly if Francis formula is used for the computation of discharge, then Eq. 10.26 becomes

$$t = \frac{2A}{1.84(L-0.1n\,H)}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right] \quad ...(10.28)$$

in which the value of H being taken as a mean of H_1 and H_2.

In order to determine the time required to lower the liquid surface to the crest of the weir or notch, H_2 must be considered to be equal to zero, in which case the time t in Eq. 10.26 becomes infinity. It therefore means that the liquid surface can never be reduced to the level of the crest of the weir or notch. This is however not true because the liquid in the reservoir is not at rest but it has some velocity with which it is approaching the weir or notch. This causes the time of emptying the reservoir upto the crest of the weir or notch to be finite, that is less than that given by Eq. 10.26. Moreover as $H_2 \to 0$ the effect of surface tension becomes significant, in which case Eq. 10.25 is not applicable.

If instead of a rectangular weir or notch, a triangular weir or notch is provided in the side of the tank in order to reduce the level of liquid in the tank from H_1 to H_2 above the crest of the weir or notch, then as in the previous case

$$-Adh = Qdt$$

But in this case

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}h^{5/2}$$

Thus $$-Adh = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}h^{5/2}dt$$

or $$dt = \frac{-Adh}{\frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}h^{5/2}}$$

By integrating both sides

$$t = \frac{\frac{2}{3}A}{\frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}}\times\left[\frac{1}{(H_2)^{3/2}}-\frac{1}{(H_1)^{3/2}}\right] \quad ...(10.29)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

10.10 EFFECT ON COMPUTED DISCHARGE OVER A WEIR OR NOTCH DUE TO ERROR IN THE MEASUREMENT OF HEAD

The discharge over a rectangular weir or notch is proportional to $(H)^{3/2}$ and over a triangular weir or notch it is proportional to $(H)^{5/2}$, where H is the height of the liquid surface above the crest of the weir or notch. As such the accurate measurement of the head H is quite essential in order to obtain an accurate value of the discharge over the weir or notch. However, if an error is introduced in the measurement of the head, it will affect the computed discharge. It is, therefore, desired to determine upto what extent an error in measuring the head will affect the computed discharge over the weir or notch. For the rectangular and triangular weir or notch this may be computed as follows:

(*a*) *Rectangular weir or notch.* The discharge over a rectangular weir or notch is given by

$$Q = \frac{2}{3} C_d \sqrt{2g}\, LH^{3/2} = KH^{3/2}$$

where $$K = \left(\frac{2}{3} C_d \sqrt{2g}\, L \right)$$

Differentiating the above equation

$$dQ = \frac{3}{2} KH^{1/2} dH$$

Then $$\frac{dQ}{Q} = \frac{\frac{3}{2} KH^{1/2} dH}{KH^{3/2}} = \frac{2}{3} \frac{dH}{H}$$

That is, a percentage change in H produces 1.5 times the same percentage change in Q, or an error of 1 per cent in measuring H will produce 1.5 per cent error in the computed Q over a rectangular weir or notch.

(*b*) *Triangular weir or notch.* The discharge over a triangular weir or notch is given by

$$Q = \frac{8}{15} C_d \sqrt{2g}\, \tan\frac{\theta}{2} H^{5/2} = KH^{5/2}$$

where $$K = \left(\frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} \right)$$

Differentiating the above equation

$$dQ = \frac{5}{2} KH^{3/2} dH$$

Then $$\frac{dQ}{Q} = \frac{\frac{5}{2} KH^{3/2} dH}{KH^{5/2}} = \frac{5}{2} \frac{dH}{H}$$

Therefore a small change or error of 1 per cent in the head produces an error of 2.5 per cent in the computed discharge over a triangular weir or notch.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

10.11 BROAD CRESTED WEIR

A weir having a wide crest is known as a broad crested weir. Such a weir differs from thin-plate and narrow-crested weirs in this respect that as water flows over it a different flow pattern is developed. Experiments have shown that if the width of the crest of the weir $B < 0.625\,H$ [i.e., $(H/B) > 1.6$] then it is known as thin-plate weir in which case the jet of water touches only the upstream edge of the weir and it flows clear of the downstream edge. If the width of the crest of the weir $B \geq 0.625\ H$ and $\leq 2.5H$ [i.e., $0.4 \leq (H/B) \leq 1.6$] it is known as narrow-crested weir in which case as the water flows over it the jet of water remains in contact with the entire crest. On the other hand when the width of the crest of the weir $B \geq 2.5H$ and $\leq 10H$ [i.e., $0.1 \leq (H/B) \leq 0.4$] it is known as *broad-crested weir* in which case as shown in Fig. 10.6 as water flows over it there occurs a drop in the water surface over the crest of the weir. There are two types of broad crested weirs namely (i) with a sharp corner at the upstream end [Fig. 10.6 (a)] and (ii) with a round corner at the upstream end [Fig. 10.6 (b)]. In the case of a broad crested weir

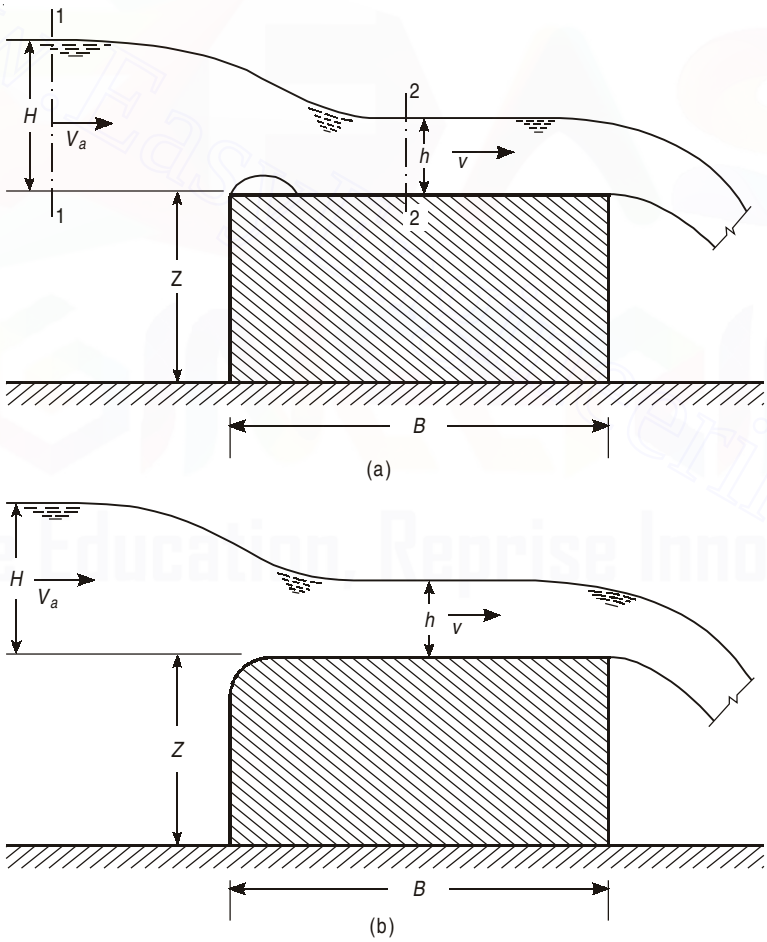


Figure 10.6 *Broad crested weir : (a) with sharp corner at the upstream end (b) with round corner at the upstream end*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

with sharp corner at the upstream end a small air pocket is developed on the crest of the weir close to the upstream edge. This is due to the lower nappe being slightly raised by the upstream sharp edge and then it drops down and flows over the surface of the weir crest. From the air pocket so developed the air is gradually carried away by the flowing stream of water, thereby resulting in the reduction of the pressure in this portion, by which the cavitation may occur at the corner and consequently the weir may be damaged. On the other hand, in the case of a broad crested weir with the upstream corner rounded, no air pocket is formed and consequent possibility of cavitation in avoided. Therefore for most of the broad crested weirs the upstream edges are rounded in order to avoid cavitation.

Figure 10.6 shows the flow pattern over a broad crested weir. As water flows over a broad crested weir the water surface drops from H on the upstream of the weir to h over the weir crest, due to the reduction in the area of flow section and consequent increase in the velocity of flow of water. If the velocity of approach is neglected then just upstream from the crest the total head above the weir crest may be represented by H. Further over the middle portion of the crest of the weir the depth of flow h may be assumed to be uniform, where let the velocity of flow be v. Then applying Bernoulli's equation between section 1–1 just upstream of the weir and section 2–2 at the middle of the weir crest, we have

$$H = h + \frac{v^2}{2g}$$

or $$v = \sqrt{2g(H-h)}$$

Now if L is the length of the crest of the weir then the discharge over the broad crested weir is given by

$$Q = Lh\sqrt{2g(H-h)}$$

In deriving the above expression the loss of head between section 1–1 and 2–2 has not been considered. As such the above expression gives only the theoretical discharge over a broad crested weir. The actual discharge over a broad crested weir is therefore obtained by introducing the coefficient of discharge C_d and hence

$$Q = C_d Lh\sqrt{2g(H-h)} \qquad ...(10.30)$$

Equation 10.30 shows that for the computation of discharge over a broad crested weir, two heads H and h need be measured. However, experiments have shown that in the case of broad crested weirs the flow adjusts itself to have maximum discharge for the available head H. The condition for the discharge Q over a broad crested weir to be maximum for constant H may be obtained by differentiating Eq. 10.30 with respect to h, and equating $\left(\frac{dQ}{dh}\right)$ to zero.

Thus

$$\frac{dQ}{dh} = C_d L\sqrt{2g}\left[\sqrt{H-h} - \frac{h}{2\sqrt{H-h}}\right] = 0$$

or $$h = \frac{2}{3}H.$$

This value of h is known as the *critical depth.* In other words, the discharge over a broad crested weir

IS: 6059–1971 provides detailed specifications for weirs of finite crest width (or broad crested weirs)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

is maximum when the critical depth of flow occurs over the surface of the weir crest. Introducing this value of h in Eq. 10.30 the discharge over a broad crested weir is given by

$$Q = C_d L\left(\frac{2}{3}H\right)\sqrt{2g}\left(\frac{1}{3}H\right)^{1/2} = 1.70\ C_d L H^{3/2} \quad \ldots (10.31)$$

Equation 10.31 gives maximum discharge over a broad crested weir, but as stated earlier for broad crested weirs the flow adjusts itself to discharge at the maximum rate. Therefore the discharge over a broad crested weir may be computed by using Eq. 10.31 in which only the head H on the upstream of the weir is required to be measured. The value of C_d for a broad crested weir may also be determined by calibrating it previously with the liquid the discharge of which it will measure. In general the value of C_d for a broad crested weir varies from 0.85 to 1.0.

In deriving the above equation for discharge over a broad crested weir the velocity of approach has been neglected. However if the velocity of approach V_a is to be considered then Eq. 10.30 may be modified as

$$Q = 1.70\ C_d L H_1^{3/2} \quad \ldots (10.32)$$

in which $$H_1 = (H + h_a) = \left[H + \left(\frac{V_a^2}{2g}\right)\right]$$

10.12 SUBMERGED WEIRS

When the water level on the downstream of the weir is above the crest of the weir then the weir is said to be a submerged weir. During floods often the weirs constructed across rivers become submerged. Submerged weirs have larger discharging capacity as compared with freely discharging weirs, thereby indicating that during floods when river carries huge quantity of water, the flow adjusts itself by setting the weir in the state of submergence, so that the discharge over the weir is increased and the flood water is quickly discharged to the downstream side.

As shown in Fig. 10.7 the discharge over a submerged weir may be obtained by dividing it into two parts. The portion between the upstream and downstream water surfaces may be treated as a free weir and the portion between the downstream water surface and the crest of the weir may be treated as a drowned orifice. Thus if Q_1 and Q_2 are the discharges through the free and the drowned portions respectively then

$$Q_1 = \frac{2}{3} C_{d_1} L \sqrt{2g}\,(H_1 - H_2)^{3/2}$$

and $$Q_2 = C_{d_2}(L \times H_2)\sqrt{2g(H_1 - H_2)}$$

where H_1 and H_2 are respectively the heads on the upstream and the downstream of the weir; L is the length of the weir; and C_{d_1} and C_{d_2} are the coefficients of discharge for the free and the drowned portions respectively.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The total discharge over a submerged weir is then

$$Q = Q_1 + Q_2$$

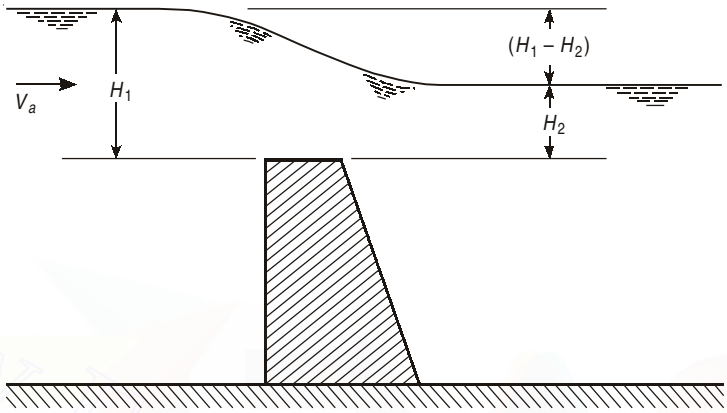


Figure 10.7 *Submerged weir*

If the velocity of approach V_a is to be considered then the values of Q_1 and Q_2 may be expressed as

$$Q_1 = \frac{2}{3} C_{d_1} \sqrt{2g}\, L \left[\left\{ (H_1 - H_2) + \frac{V_a^2}{2g} \right\}^{3/2} - \left(\frac{V_a^2}{2g} \right)^{3/2} \right]$$

$$Q_2 = C_{d_2} (L \times H_2) \left[\sqrt{2g(H_1 - H_2) + V_a^2} \right]$$

A sharp crested weir is more susceptible to submergence than a broad crested weir. But even a sharp crested weir behaves as a free weir only, for a submergence ratio $\left[\dfrac{H_2}{H_1 + \left(\dfrac{V_a^2}{2g} \right)} \right]$ as high as 0.66.

The corresponding value of the submergence ratio upto which a broad crested weir behaves as a free weir is as high as 0.83 to 0.85. This is attributed to the fact that the flow conditions are such that the downstream water level is held away from the crest and hence it does not affect the flow conditions upstream. The limiting value of the submergence ratio upto which any submerged weir may behave as a free weir is known as its *modular limit*.

10.13 SPILLWAY AND SIPHON SPILLWAY

A spillway is a portion of a dam over which the excess water, which cannot be stored in the reservoir formed on the upstream of the dam, flows to the downstream side. In general the shape of the spillway profile is made to follow the profiles of a lower nappe of a well-ventilated sharp crested weir. In other words, the spillway is formed by filling the space between the sharp crested weir and the lower nappe with concrete or masonry as shown in Fig. 10.8 (a). Such a spillway is known as *Ogee spillway* (or Ogee

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

weir). The main advantage of providing such a shape for the spillway is that the flowing sheet of water remains in contact with the surface of the spillway, thereby preventing the negative pressure being

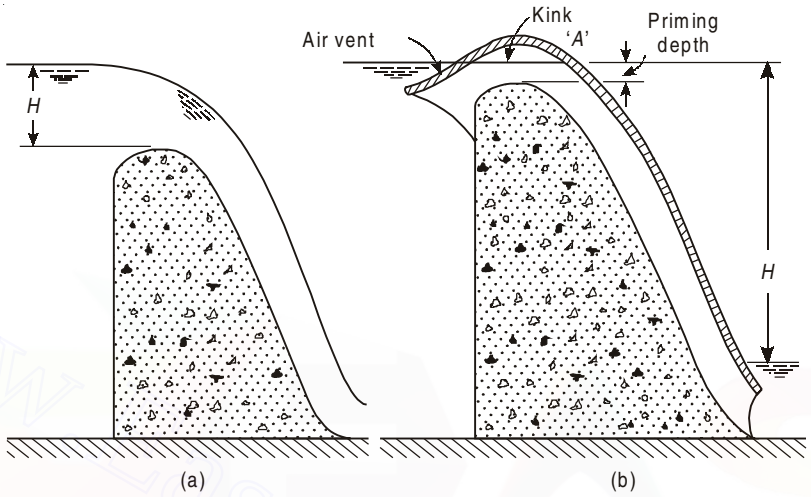


Figure 10.8 (*a*) *Ogee spillway*, (*b*) *Siphon spillway*

developed on the downstream side. However this condition will be fulfilled only as long as the head over crest of the spillway is equal to or less than the designed head. For the heads larger than the designed head the sheet of water may separate from the spillway, resulting in the development of the negative pressure on some portion of the spillway. As such, the actual shape of a spillway is usually determined by conducting the experiments on the models of the actual spillways. The discharge over an Ogee spillway may be expressed as

$$Q = CLH^{3/2} \qquad ...(10.33)$$

where C is the coefficient of the spillway, L is the length of the spillway and H is the head above the crest of the spillway. The coefficient C of the spillway may be determined by calibration.

Siphon Spillway. A siphon spillway consists of an Ogee weir which is provided with an air-tight cover as shown in Fig. 10.8 (b), thus converting the discharge face of the spillway into a large rectangular sectioned pipe connecting the upstream and the downstream water surfaces. In the case of a siphon spillway the head H under which the water flows is equal to the difference between the water surfaces on the upstream and the downstream sides. As such as compared with an ordinary spillway the head in the case of a siphon spillway is more, on account of which the siphon spillway has a much greater discharge for a given length than an ordinary open spillway. The working of a siphon spillway is automatic which is as explained below.

When the water level in the upstream reservoir rises above the crest of the spillway, water begins to flow over the crest through the siphon. The jet of flowing water strikes the inside of the cover, thus enclosing a small space A called kink, in the upper portion of the cover, in which air is trapped. As more and more water flows down, it sucks the trapped air from the kink, thus creating a partial vacuum in this portion which sucks up the water from the reservoir and completely fills the pipe. The siphonic action is thus started and the effective head causing the flow of water becomes equal to H, the difference between the water surfaces on the upstream and the downstream sides. When the level of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

water in the reservoir reduces and becomes almost equal to the crest level of the spillway, then the siphonic action should stop, otherwise the water in the reservoir may be lowered to a level much below the crest of the spillway. For this purpose air vents are provided in the cover at a level slightly above the crest of the spillway so that when the water in the reservoir drops below this level the air vents get exposed and air enters the upper portion thereby stopping the siphonic action. By such an arrangement the working of a siphon spillway may be made automatic and hence often it is called *automatic siphon spillway.*

A siphon spillway has the following advantages over an ordinary open spillway.

(i) The operating head is increased considerably due to which the discharge is increased.

(ii) The depth of water (called priming depth) required for the commencement of the siphonic action is only a few centimetres above the crest of the spillway. As such the crest of a siphon spillway can be raised, thereby allowing a greater amount of water to be stored in the reservoir.

10.14 PROPORTINAL WEIR OR SUTRO WEIR

In general for the weirs discussed earlier the discharge Q may be expressed as $Q \propto H^n$, where $n = \frac{3}{2}$ for a rectangular weir and $n = \frac{5}{2}$ for a triangular weir. However, it is possible to design a weir of such a shape for which $Q \propto H$ i.e., discharge Q varies linearly with the head H over the weir crest. Such a weir is called a proportional weir or Sutro weir in honour of its first inventor. Figure 10.9 shows a proportional weir.

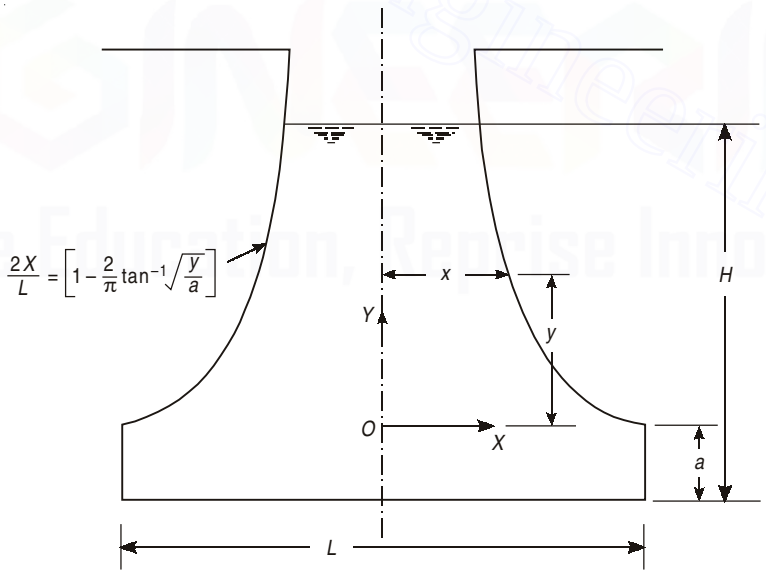


Figure 10.9 *Proportional weir*

Sutro analytically obtained the relationship for the shape of the proportional weir profile having $Q \propto H$, as

$$x \propto y^{-1/2} \qquad ...(10.34)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, it may be noted from the above relationship that as $y \to 0, x \to \infty$, which means that the width of the weir aperture becomes infinity at the crest. The infinity tending profile is however not practicable. In order to overcome this practical limitation Sutro modified the shape of the weir profile so that a finite width at the weir crest may be provided. The modified proportional weir profile as given by Sutro is shown in Fig. 10.9, which has its sides diverging downward in the form of hyperbolic curves having the equation

$$\frac{2x}{L} = \left[1 - \frac{2}{\pi}\tan^{-1}\sqrt{\frac{y}{a}}\right] \quad ...(10.35)$$

where a and L are respectively the height and width of the small rectangular shaped aperture which forms the base of the weir. The discharge through this weir is given by

$$Q = k\left(H - \frac{a}{3}\right) \quad ...(10.36)$$

where $k = C_d L (2ga)^{1/2}$

The coefficient of discharge C_d for such weirs varies from 0.60 to 0.65. Proportional weir is very useful as a control device, especially in chemical dosing and sampling.

ILLUSTRATIVE EXAMPLES

Example 10.1. *Find the discharge over a suppressed rectangular weir 4 m long with a head over the crest as 0.35 m.*

Solution

Using Francis formula

$$Q = 1.84\, LH^{3/2}$$
$$L = 4 \text{ m and } H = 0.35 \text{ m}$$
$$Q = 1.84 \times 4 \times (0.35)^{3/2} = 1.524 \text{ m}^3/s.$$

Example 10.2. *A rectangular weir 6 m long is divided into 3 bays by two vertical posts each 0.3 m wide. Find the discharge when the head is 0.45 m.*

Solution

Using Francis formula

$$Q = 1.84(L - 0.1nH)\, H^{3/2}$$
$$L = (6 - 2 \times 0.30) = 5.4 \text{ m}$$
$$n = 6\ ;\ H = 0.45 \text{ m}$$
$$\therefore \quad Q = 1.84\,[5.4 - (0.1 \times 6) \times 0.45] \times (0.45)^{3/2}$$
$$= 2.85 \text{ m}^3/\text{sec}.$$

Example 10.3. *The maximum flow through a rectangular flume 1.8 m wide and 1.2 m deep is 1.65 m^3/sec. It is proposed to install a suppressed sharp crested rectangular weir across the flume to measure flow. Find the maximum height at which the weir crest can be placed in order that water may not overflow the sides of the flume. Assume $C_d = 0.6$.*

Solution

Neglecting the velocity of approach, the discharge over a rectangular weir is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = \frac{2}{3} C_d \sqrt{2g}\ LH^{3/2}$$

$$Q = 1.65 \text{ m}^3/\text{sec}; L = 1.8 \text{ m}$$

$$1.65 = \frac{2}{3} \times 0.6 \times \sqrt{2 \times 9.81} \times 1.8 \times H^{3/2}$$

$$H = 0.644 \text{ m.}$$

If Z is the height of weir crest above the bottom of the flume, then

$$Z + 0.644 = 1.2$$

$$\therefore \quad Z = 0.556 \text{ m}$$

If the velocity of approach is taken into account then

$$Q = \frac{2}{3} C_d \sqrt{2gL} \left[(H + h_a)^{3/2} - h_a^{3/2} \right]$$

$$V_a = \frac{1.65}{1.8 \times 1.2} = 0.764 \text{ m/sec}$$

$$h_a = \frac{V_a^2}{2g} = \frac{(0.764)^2}{2 \times 9.81} = 0.029\ 8 \text{ m}$$

Thus $1.65 = \frac{2}{3} \times 0.6 \times \sqrt{2 \times 9.81} \times 1.8 \times \left[(H + 0.0298)^{3/2} - (0.0298)^{3/2} \right]$

or $H = 0.619 \text{ m}$

$$\therefore \quad Z = (1.2 - 0.619) = 0.581 \text{ m.}$$

Example 10.4. *A rectangular notch of crest width 0.4 m is used to measure the flow of water in a rectangular channel 0.6 m wide and 0.45 m deep. If the water level in the channel is 0.225 m above the weir crest, find the discharge in the channel. For the notch assume $C_d = 0.63$ and take velocity of approach into account.*

Solution

If the velocity of approach is neglected

$$Q = \frac{2}{3} C_d \sqrt{2g} (L - 0.1n\ H) H^{3/2}$$

$$C_d = 0.63;\ L = 0.40$$

$$H = 0.225 \text{ m};\ n = 2$$

Thus

$$Q = \frac{2}{3} \times 0.63 \times \sqrt{2 \times 9.81}\quad (0.40 - 0.2 \times 0.225) \times (0.225)^{3/2}$$

$$= 0.0705 \text{ m}^3/\text{s}$$

For this discharge the velocity of approach

$$V_a = \frac{0.0705}{0.6 \times 0.45}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= 0.261$ m/sec

Then

$$h_a = \frac{V_a^2}{2g} = \frac{(0.261)^2}{2\times 9.81} = 0.0035 \text{ m}$$

and $H_1 = (H + h_a) = (0.225 + 0.003\,5) = 0.2285$ m

Thus

$$Q = \frac{2}{3}\times 0.63\times\sqrt{2\times 9.81}\ (0.40 - 0.2\times 0.2285)\times[(0.2285)^{3/2} - (0.0035)^{3/2}$$

$$= 0.07186 \text{ m}^3/\text{s}$$

New velocity of approach

$$V_a = \frac{0.07186}{0.6\times 0.45} = 0.266 \text{ m/s}$$

$$h_a = \frac{V_a^2}{2g} = \frac{(0.266)^2}{2\times 9.81} = 0.0036 \text{ m}$$

$$H_1 = (H + h_a) = (0.225 + 0.0036) = 0.2286 \text{ m}$$

Thus

$$Q = \frac{2}{3}\times 0.63\times\sqrt{2\times 9.81}\ (0.40 - 0.2\times 0.2286)\times[(0.2286)^{3/2} - (0.0036)^{3/2}]$$

$$= 0.0719 \text{ m}^3/\text{s}$$

The new value of discharge is quite close to the previous value and hence it may be considered as the correct value of the discharge.

Example 10.5. *Find the depth and top width of a V-notch capable of discharging a maximum of 0.7 m^3/sec and such that the head shall be 75 mm for a discharge of 5.6 litres per second. Its C_d is the same as that of a similar (in material and sharpness of edges only) right-angled V-notch for which*

$$Q = 1.407^{5/2}$$

Solution

For a triangular notch

$$Q = \frac{8}{15} C_d \sqrt{2g}\ \tan\frac{\theta}{2} H^{5/2}$$

in which $\left(\frac{8}{15} C_d \sqrt{2g}\right) = 1.407$, as given.

Thus $Q = 1.407 \tan\frac{\theta}{2} H^{5/2}$

When $Q = 5.6 \times 10^{-3} = 0.0056$ m^3/s

$H = 75 \times 10^{-3} = 0.075$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by substitution, we get

$$0.0056 = 1.407(0.075)^{5/2} \tan\frac{\theta}{2}$$

or $$\tan\frac{\theta}{2} = \frac{0.0056}{1.407\times(0.075)^{5/2}} = 2.584$$

Now if H is the required head when $Q = 0.7\ m^3/s$, then

$$0.7 = 1.407 \times H^{5/2} \times 2.584$$

$$H = \left(\frac{0.7}{1.407\times 2.584}\right)^{2/5} = 0.517\ m$$

$$\text{Top width} = 2H\tan\frac{\theta}{2}$$

$$= (2 \times 0.517 \times 2.5840) = 2.672\ m$$

Example 10.6. *The following observations of head and the corresponding discharge were made in connection with a weir 0.277 m wide.*

Head in mm	*30*	*150*	*300*	*450*	*600*	*750*	*900*	*1050*	*1200*
Discharge in litres / second	*4.60*	*32.4*	*90.45*	*164.7*	*251.64*	*351.81*	*459.81*	*581.6*	*712.8*

Assuming $Q = CLH^n$, *find C and n.*

Solution

$$Q = CLH^n$$

$$L = 0.277\ m$$

so $$Q = 0.277\ CH^n$$

Taking log of both sides, we get

$$\log Q = \log(0.277\ C) + n \log H$$

The values of log Q and log H (taking H in metres and Q in m^3/sec) for the given data are as tabulated below:

$\log H$	$\bar{2}.477$	$\bar{1}.176$	$\bar{1}.477$	$\bar{1}.653$	$\bar{1}.778$	$\bar{1}.875$	$\bar{1}.954$	$\bar{0}.021$	$\bar{0}.079$
$\log Q$	$\bar{3}.663$	$\bar{2}.511$	$\bar{2}.956$	$\bar{1}.217$	$\bar{1}.401$	$\bar{1}546$	$\bar{1}.663$	$\bar{1}.765$	$\bar{1}.853$

The plot of log Q *v/s* log H is as shown in the Fig. Ex. 10.6

Since for $H = 1$

$$\log H = 0$$

$$\log Q = \log(0.277\ C)$$

From the graph, for $\log H = 0$

$$\log Q = \bar{1}.70$$

Thus $$\log(0.277\ C) = \bar{1}.70$$

$$\therefore \quad C = \frac{0.5012}{0.277} = 1.81$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further $$n = \frac{\log Q - \log(0.277C)}{\log H}$$

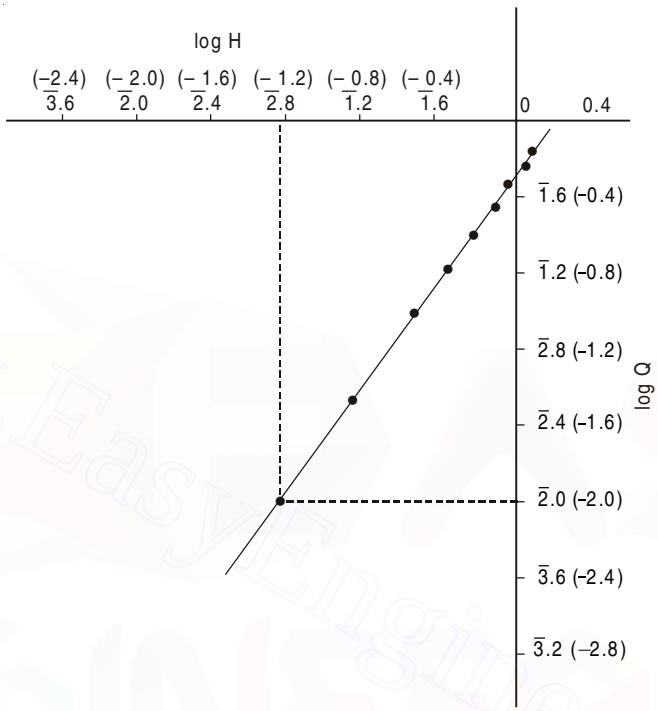


Figure Ex. 10.6

Choosing any point on the curve

$$\log H = \bar{2}.78$$
$$\log Q = \bar{2}.0$$
$$n = \frac{\bar{2}.0 - \bar{1}.70}{\bar{2}.78} = 1.39$$
$$\therefore \quad Q = 1.817H^{1.39}$$

Example 10.7. *A triangular notch is used to measure flow in a channel under a head of 0.2 m. If the discharge is to be measured within 3% accuracy, what is the maximum velocity of approach that can be neglected.*

Solution

Let Q be the discharge when the velocity of approach is neglected, and Q_1 be the discharge when the velocity of approach is taken into account.

Thus

$$Q = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \qquad ...(i)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$Q_1 = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2}\left[(H + h_a)^{5/2} - h_a^{5/2}\right] \quad ...(ii)$$

From Eq. (i) and (ii), we have

$$\frac{(Q_1 - Q)}{Q} = \frac{\left[\left(H + h_a^{5/2}\right) - h_a^{5/2}\right] - H^{5/2}}{H^{5/2}}$$

By substituting the given values, we get

$$0.03 = \frac{\left[(0.2 + h_a)^{5/2} - h_a^{5/2}\right]}{(0.2)^{5/2}} - 1$$

or $$1.03 = \frac{\left[(0.2 + h_a)^{5/2} - h_a^{5/2}\right]}{(0.2)^{5/2}}$$

Solving for h_a by trial and error, we get

$$h_a = 2.38 \times 10^{-3} \text{ m}$$

or $$\frac{V_a^2}{2g} = 2.38 \times 10^{-3}$$

$$\therefore \quad V_a = 0.216 \text{ m/s}$$

Example 10.8. *A stream approaching a water fall having a fall of 36 m is gaged by a weir. The measured head over the weir is 0.2575 m and the length of the weir is 3 m. The velocity of approach is 1.2 m/s. Determine the power available at the water fall. Use Bazin's formula with* $\alpha = 1.5$ *for flow over the weir.*

Solution

According to Bazin's formula

$$Q = m_1 \sqrt{2g} L H_1^{3/2}$$

$$m_1 = \left(0.405 + \frac{0.003}{H_1}\right)$$

$$H_1 = \left(H + \alpha \frac{V_a^2}{2g}\right)$$

$$= \left[0.275 + 1.5 \times \frac{(1.2)^2}{2 \times 9.81}\right]$$

$$= 0.385 \text{ m}$$

$$m_1 = \left(0.405 + \frac{0.003}{0.385}\right) = 0.413$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $Q = 0.413 \times \sqrt{2 \times 9.81} \times 3 \times (0.385)^{3/2}$

$= 1.311 \text{ m}^3/\text{sec}$

Power available at the fall

$$= \frac{wQH}{1000} \text{kW}$$

$$= \frac{9810 \times 1.311 \times 36}{1000} = 463 \text{ kW}$$

Example 10.9. *Rain falls over a catchment area of 26 sq km at the rate of 1 mm per hour. The rain water flows over a weir with a free length of 12 m constructed in 8 bays each 1.5 m long. Using Francis formula, find the head over weir crest.*

Solution

The discharge over the weir is

$$Q = \frac{26 \times 10^6 \times 1}{1000 \times 60 \times 60} = 7.22 \text{ m}^3/\text{sec.}$$

Using Francis formula

$$Q = 1.84(L - 0.1nH)H^{3/2}$$

$L = 12$ m and $n = (2 \times 8) = 16$

Thus $7.22 = 1.84(12 - 1.6H)H^{3/2}$

Solving by trial and error

$H = 0.5$ m.

Example 10.10. *Water flows through a rectangular channel 1 m wide and 0.5 m deep, and then over a sharp crested Cipolletti weir of crest length 0.6 m. If the water level in the channel is 0.225 m above the weir crest, calculate the discharge over the weir. Take $C_d = 0.6$ and make correction for velocity of approach.*

Solution

If the velocity of approach is neglected then for a Cipolletti weir

$$Q = \frac{2}{3} C_d \sqrt{2g} L H^{3/2}$$

$C_d = 0.6$, $L = 0.60$ m, $H = 0.225$ m

Then $Q = \frac{2}{3} \times 0.6 \times \sqrt{2 \times 9.81} \times 0.6 \times (0.225)^{3/2}$

$= 0.1135 \text{ m}^3/\text{s}$

Velocity of approach

$$V_a = \frac{0.1135}{1 \times 0.5} = 0.227 \text{ m/s}$$

Thus $h_a = \frac{V_a^2}{2g} = \frac{(0.227)^2}{2 \times 9.81} = 0.0026$ m

and $H_1 = (H + h_a) = (0.225 + 0.0026) = 0.2276$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For velocity of approach being taken in account

$$Q = \frac{2}{3} C_d \sqrt{2g} \quad L\left[H_1^{3/2} - h_a^{3/2}\right]$$

$$= \frac{2}{3} \times 0.6 \times \sqrt{2 \times 9.81} \times 0.6 \times \left[(0.2276)^{3/2} - (0.0026)^{3/2}\right]$$

$$= 0.1153 \text{ m}^3/\text{s}.$$

Since the new value of the discharge is quite close to the previous value the correct discharge over the Cipolletti weir may be taken as 0.115 3 m^3/s.

Example 10.11. *A rectangular channel 6 m wide carries 2800 litres per second at a depth of 0.9 m. What height of a broad crested rectangular weir must be installed to double the depth? Assume a weir coefficient of 0.86.*

Solution

From Eq. 10.31 for a broad crested weir

$$Q = 1.70\, C_d L H^{3/2}$$

$$Q = 280 \times 10^{-3} = 2.8 \text{ m}^3/s$$

$$C_d = 0.86; L = 6 \text{ m}$$

Thus by substitution, we get

$$2.8 = 1.70 \times 0.86 \times 6 \times H^{3/2}$$

$$\therefore \quad H = 0.467 \text{ m}$$

The depth of flow required to be developed in the channel

$$Z = (2 \times 0.90) = 1.80 \text{ m}$$

$\therefore$ Height of the broad crested weir

$$= (1.80 - 0.467) = 1.333 \text{ m}$$

If the velocity of approach is taken into account then using Eq. 10.32, we get

$$H_1 = (H + h_a) = 0.467 \text{ m}$$

The velocity of approach

$$V_a = \frac{2.8}{6 \times 1.8} = 0.26 \text{ m/s}$$

and

$$h_a = \frac{V_a^2}{2g} = \frac{(0.26)^2}{2 \times 9.81} = 0.0034 \text{ m}$$

Thus $H = (0.467 - 0.0034) = 0.4636$ m

$\therefore$ $Z = (1.80 - 0.4636) = 1.3364$ m

Example 10.12. *A Cipolletti weir, with coefficient of discharge $C_d = 0.625$ and crest length $L = 30$ m is provided in the side of a reservoir. Area of water surface in the reservoir A, in sq. metres is given by the expression*

$$A = 9.3 \times 10^3 (15 + \alpha h^2)$$

where α is a constant whose value may be taken as 0.5 and h is the head of water over the weir crest in metres. Calculate the time taken to lower the water surface in the reservoir from a head of 1.2 m above the weir crest to a head of 0.3 m.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

As described in Section 10.9

$$-A\,dh = Q\,dt$$

$$Q = \frac{2}{3} C_d \sqrt{2g}\; L H^{3/2}$$

and $$A = (9.3 \times 10^3)\,(15 + 0.5h^2)$$

On substitution,

$$-(9.3 \times 10^3)\,(15 + 0.5h^2)\,dh = \left(\frac{2}{3} \times 0.625 \times \sqrt{2 \times 9.81} \times 30 \times h^{3/2}\right) dt$$

or $$dt = \frac{-(9.3 \times 10^3)\;\;(15 + 0.5h^2)\,dh}{\frac{2}{3} \times 0.625 \times \sqrt{2 \times 9.81} \times 30 \times h^{3/2}}$$

By integrating both sides, we get

$$t = \frac{-(9.3 \times 10^3)}{\frac{2}{3} \times 0.625 \times \sqrt{2 \times 9.81} \times 30} \int_{1.2}^{0.3} \frac{(15 + 0.5h^2)}{h^{3/2}}\,dh$$

$$= -168\left[-\frac{30}{h^{1/2}} + \frac{h^{3/2}}{3}\right]_{1.2}^{0.3}$$

$$= 4665 \text{ s} = 1.296 \text{ h}$$

Example 10.13. *A reservoir 4.65 × 10⁴ m² in area is to be controlled by a rectangular weir with its crest level at El.30. It is intended to provide such a length of weir that will lower the water level from El. 31.2 to El. 30.6 in half an hour time. Determine the length of weir. The discharge over the weir is given by the formula $Q = 1.9\,LH^{3/2}$ where Q is discharge in cumec, L is crest length in metres and H is head over the weir in metres.*

Solution

From Eq. 10.25

$$t = \frac{2A}{\frac{2}{3} C_d \sqrt{2g}\;\; L\left[\frac{1}{\sqrt{H_2}} - \frac{1}{\sqrt{H_1}}\right]}$$

In this case $$\left(\frac{2}{3} C_d \sqrt{2g}\right) = 1.9$$

Thus $$t = \frac{2A}{1.9L}\left[\frac{1}{\sqrt{H_2}} - \frac{1}{\sqrt{H_1}}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$t = \left(\frac{1}{2}\times 60\times 60\right) = 1800 \text{ s}$$

$$A = 4.65 \times 10^4 \text{ m}^2$$

$$H_1 = (31.2 - 30.0) = 1.2 \text{ m}$$

$$H_2 = (30.6 - 30.0) = 0.6 \text{ m}$$

Thus by substitution, we get

$$1800 = \frac{2\times 4.65\times 10^4}{1.9\times L}\left[\frac{1}{(0.6)^{1/2}} - \frac{1}{(1.2)^{1/2}}\right]$$

$$\therefore \quad L = 10.28 \text{ m}$$

Example 10.14. *A right-angled triangular notch is provided in the vertical side of a tank having plan area of 0.93 m^2 uniform at all levels. When the head over the notch is 75 mm, it is found that the water surface in the tank is falling down at a rate of 2.54 mm per second. Calculate the coefficient of discharge of the notch.*

Solution

As described in Section 10.9

$$-Adh = Qdt$$

$$A = 0.93 \text{ m}^2; \; Q = \frac{8}{15} C_d \sqrt{2g} \quad H^{5/2}$$

When $H = 0.075$ m, $-\left(\frac{dh}{dt}\right) = 2.54\times 10^{-3}$ m/s

Thus by substitution

$$0.93\times 2.54\times 10^{-3} = \frac{8}{15} C_d \times \sqrt{2\times 9.81}\times (0.075)^{5/2}$$

$$C_d = 0.649.$$

Example 10.15. *A submerged sharp crested weir 0.81 m high stands clear across a channel having vertical sides and width of 3.15 m. The depth of water in the channel of approach is 1.26 m, and 10.5 m downstream from the weir the depth of water is 0.93 m. Determine the discharge in litres per minute. Assume* $C_{d_1} = 0.58$ *and* $C_{d_2} = 0.80$.

Solution

Discharge over a submerged weir is given by

$$Q = Q_1 + Q_2$$

where neglecting the velocity of approach

$$Q_1 = \frac{2}{3} C_{d_1} \sqrt{2g}\; L\; (H_1 - H_2)^{3/2}$$

and

$$Q_2 = C_{d_2} \times (L\times H_2)\times \sqrt{2g(H_1 - H_2)}$$

$$L = 3.15 \text{ m}, \; H_1 = (1.26 - 0.81) = 0.45 \text{ m}$$

and

$$H_2 = (0.93 - 0.81) = 0.12 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $Q_1 = \frac{2}{3} \times 0.58 \times \sqrt{2 \times 9.81} \times 3.15(0.45 - 0.12)^{3/2}$

$= 1.023 \text{ m}^3/\text{s}$

$Q_2 = 0.80 \times (3.15 \times 0.12)\sqrt{2 \times 9.81(0.45 - 0.12)}$

$= 0.769 \text{ m}^3/\text{s}$

$\therefore \quad Q = (1.023 + 0.769) \text{ m}^3/\text{s}$

$= 1.792 \text{ m}^3/\text{s}$

$= 1.792 \times 10^3 \, l/\text{s}$

$= 1.792 \times 10^3 \times 60 \, l/\text{min}$

$= 10.752 \times 10^4 \, l/\text{min}$

Note. If velocity of approach is to be considered, necessary correction may be applied.

Example 10.16. *A tank with vertical sides and a horizontal cross-sectional area 1.86 m^2 is provided with a notch cut at the top of one of the sides. Water flowing into the tank at constant rate was discharged over the notch, the head over the crest of which was 0.225 m. The supply of water was suddenly stopped and it was observed that the head over the notch started to fall at the rate of 5.33 mm/s. When the head had fallen to 0.1125 m it was observed that the head over the notch was falling at the rate of 1.905 mm/s. Estimate the rate of inflow to the tank when there is a steady head of 0.125 m over the notch. Assume for the notch $Q = kH^n$.*

Solution

As mentioned in Section 10.9

$$-Adh = Qdt$$

or $$-\frac{dh}{dt} = \frac{Q}{A} = \frac{KH^n}{A}$$

when $H = 0.225 \text{ m} ; -\frac{dH}{dt} = 5.33 \times 10^{-3} \text{ m/s}$

and when $H = 0.1125 \text{ m} ; -\frac{dH}{dt} = 1.905 \times 10^{-3} \text{ m/s}$

Thus by substitution, we get

$$5.33 \times 10^{-3} = \frac{K \times (0.225)^n}{1.86} \quad \ldots(1)$$

and $$1.905 \times 10^{-3} = \frac{K \times (0.1125)^n}{1.86} \quad \ldots(2)$$

Dividing Eq. 1 by Eq. 2, we have

$$\frac{5.33}{1.905} = \left(\frac{0.225}{0.1125}\right)^n$$

$\therefore \quad n = 1.484$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Then $K = \dfrac{5.33\times10^{-3}\times1.86}{(0.225)^{1.484}} = 0.090\,7$

Therefore $Q = 0.0907H^{1.484}$

For steady head $H = 0.125$ m, the rate of inflow to the tank will be equal to discharge Q over the notch. Thus rate of inflow to the tank is

$$Q = 0.0907\ (0.125)^{1.484} = 0.004\ \text{m}^3/\text{s}$$

Example 10.17. *A river 30 m wide and 3 m deep has a mean velocity of 1.2 m per second. Find the height of an anicut (weir) to raise the water level by 1 m.*

Solution

Width of river = 30 m

Depth of flow = 3 m

∴ Area of flow section

$= (30 \times 3) = 90\ \text{m}^2$

Mean velocity of flow

$= 1.2$ m/sec

∴ Discharge $Q = (90 \times 1.2) = 108\ \text{m}^3/\text{sec}.$

Since the anicut is constructed to raise the water level by 1 m, the depth of flow on the upstream of the anicut becomes

$(3 + 1) = 4$ m

∴ Velocity of approach

$$V_a = \frac{108}{30\times4} = 0.9\ \text{m/s}$$

and $$h_a = \frac{V_a^2}{2g} = \frac{(0.9)^2}{2\times9.81} = 0.0413\ \text{m}$$

Assuming that the anicut is discharging free, then

$$Q = \frac{2}{3}C_{d_1}L\sqrt{2g}\ \left[(H+h_a)^{3/2} - h_a^{3/2}\right]$$

Assume $C_{d_1} = 0.58.$

Thus by substitution, we get

$$108 = \frac{2}{3}\times0.58\times30\times\sqrt{2\times9.81}\ [(H + 0.0413)^{3/2} - (0.0413)^{3/2}]$$

or $H = 1.604$ m

The height of the anicut is then

$Z = (4 - 1.604) = 2.396$ m

Since the depth of water in the channel on the downstream of the anicut will also be 3 m, the anicut will be drowned.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For a drowned anicut the discharge is given by

$$Q = \frac{2}{3} C_{d_1} L \sqrt{2g}\ [(H_1 - H_2 + h_a)^{3/2} - h_a^{3/2}] + C_{d_2} \times (L \times H_2) \left[\sqrt{2g(H_1 - H_2) + V_a^2}\right]$$

$$(H_1 - H_2) = 1 \text{ m}$$

Assume $C_{d_1} = 0.58$; and $C_{d_2} = 0.80$

Thus by substitution, we get

$$108 = \frac{2}{3} \times 0.58 \times 30 \times \sqrt{2 \times 9.81} \left[(1 + 0.0413)^{3/2} - (0.0413)^{3/2}\right] + 0.80 \times (30 \times H_2) \left[\sqrt{(2 \times 9.81 \times 1) + (0.9)^2}\right]$$

or $$108 = 54.17 + 108.5\ H_2$$

$$\therefore \quad H_2 = \frac{108 - 54.17}{108.5} = 0.496 \text{ m}$$

Therefore the height of the anicut

$$= (3 - 0.496) = 2.504 \text{ m}$$

Example 10.18. *A spillway 45 m long having discharge coefficient 1.8 permits a maximum discharge of 90 m^3/s from a storage reservoir. It is proposed to replace the spillway by a siphon spillway of section 0.75 m × 1.5 m with operating head 8 m and discharge coefficient 0.64. Find the number of siphons required and the amount of extra water stored, if the siphons have a priming depth of 0.15 m; the average surface area of the reservoir being 5×10^5 m^2.*

Solution

For each siphon of the spillway operating head

$$H = 8 \text{ m}$$

$$a = (0.75 \times 1.5) = 1.125 \text{ m}^2$$

$\therefore$ Discharge through each siphon

$$= C_d a \sqrt{2gH}$$

$$= 0.64 \times 1.125 \times \sqrt{2 \times 9.81 \times 8}$$

$$= 9.02 \text{ m}^3/\text{s}$$

Maximum discharge $= 90 \text{ m}^3/\text{s}$

No. of siphons $= \frac{90}{9.02} = 10$

Discharge through open spillway is

$$Q = \frac{2}{3} C_d \sqrt{2g}\ LH^{3/2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = 90 \text{ m}^3/\text{s}; \left(\frac{2}{3}C_d\sqrt{2g}\right) = 1.8$$

$$L = 45 \text{ m}$$

Thus $90 = 1.8 \times 45 \times H^{3/2}$

$$\therefore \quad H = \left(\frac{90}{1.8\times 45}\right)^{2/3} = 1.073 \text{ m}$$

Priming depth for siphons

$$= 0.15$$

$\therefore$ Height of crest raised when siphons are installed

$$= (1.073 - 0.15) = 0.923 \text{ m}$$

Area of reservoir $= 5 \times 105 \text{ m}^2$

$\therefore$ Volume of extra water stored

$$= (5 \times 10^5 \times 0.923) = 46.15 \times 10^4 \text{ m}^3$$

SUMMARY OF MAIN POINTS

1. A notch may be defined as an opening provided in the side of a tank such that the liquid surface in the tank is below the top edge of the opening. Notches made of metallic plates are also provided in narrow channels in order to measure the rate of flow of liquid. As such in general notches are used for measuring the rate of flow of liquid from a tank or in a channel.

   According to the shape of the opening the notches are classified as rectangular notch, triangular notch, (or *V*-notch), trapezoidal notch and stepped notch. According to the effect of the sides on the nappe (or the sheet of water) emerging from a notch, the notches are classified as 'notch with end contraction' and 'notch without end contraction' or 'suppressed notch'. Notches provided in the sides of tanks are essentially the notches with end contraction. Further in a channel if the crest length (length of the bottom edge) of the notch is less than the width of the channel then it is also a notch with end contraction. However, if the crest length of the notch is equal to the width of the channel then it is a notch without end contraction.
2. A weir is a concrete or masonry structure built across a river (or stream) in order to raise the level of water on the upstream side and to allow the excess water to flow over its entire length to the downstream side. Weirs may also be used for measuring the rate of flow of water in rivers or streams.

   According to the shape of the opening, the weirs are classified as rectangular and trapezoidal weirs. A particular type of trapezoidal weir is known as 'Cipolletti weir'. According to the shape of the crest, the weirs are classified as thin-plate or sharp-edged weir, narrow-crested weir, broad-crested weir and Ogee-shaped weir. According to the effect of the sides on the issuing nappe as wier with end contraction and wier without end contraction. According to the discharge conditions weirs are classified as freely discharging weir and submerged (or drowned) weir.
3. Discharge through a rectangular notch or weir without end contractions is given by:

$$Q = \frac{2}{3}C_d\sqrt{2g}\, LH^{3/2}$$

where

L = lenght of the crest of the notch or weir ;
H = height of the water surface or head of water above the crest of the notch or weir;
C_d = coefficient of discharge for the notch or weir ; and
g = acceleration due to gravity.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

4. The velocity with which water approaches a notch or weir is called velocity of approach denoted by V_a and is given by

$$V_a = \frac{\text{Discharge through the notch or weir}}{\text{Cross-sectional area of channel}}$$

The head due to velocity of approach h_a is given by

$$h_a = \frac{V_a^2}{2g}$$

5. Discharge through a rectangular notch or weir, when head due to velocity of approach is considered, is given by

$$Q = \frac{2}{3} C_d \sqrt{2g} L \left[(H + h_a)^{3/2} - (h_a)^{3/2} \right]$$

6. For a rectangular notch or weir with end contractions discharge is given by :

When velocity of approach is not considered

$$Q = \frac{2}{3} C_d \sqrt{2g} (L - 0.1nH) H^{3/2}$$

When velocity of approach is considered

$$Q = \frac{2}{3} C_d \sqrt{2g} (L - .1nH_1) \left(H_1^{3/2} - h_a^{3/2} \right)$$

where

n = number of end contractions ; and

$$H_1 = (H + h_a) = \left(H + \frac{V_a^2}{2g} \right)$$

7. According to Francis formula discharge through a rectangular weir is given by : when the velocity of apporach is not considered

$$Q = 1.84 (L - 0.1nH) H^{3/2}$$

When the velocity of approach is considered

$$Q = 1.84 (L - 0.1nH_1)(H_1^{3/2} - h_a^{3/2})$$

When end contractions are suppressed and velocity of approach is not considered

$$Q = 1.84 LH^{3/2}$$

When end contractions are suppressed but velocity of approach is considered

$$Q = 1.84 L(H_1^{3/2} - h_a^{3/2})$$

8. According to Bazin's formula discharge through a rectangular weir is given by :

When velocity of approach is not taken into account

$$Q = m\sqrt{2g}\, LH^{3/2}$$

where

$$m = \left(0.405 + \frac{0.003}{H} \right)$$

When velocity of approach is taken into account

$$Q = m_1 \sqrt{2g} LH_1^{3/2}$$

where

$$m_1 = \left(0.405 + \frac{0.003}{H} \right);$$

$$H_1 = \left(H + \alpha \frac{V_a^2}{2g} \right); \text{ and}$$

α = a constant with a mean value equal to 1.6

9. Discharge through a triangular notch or weir is given by:

When velocity of approach is not taken into account

$$Q = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2}$$

When velocity of approach is not taken into account

$$Q = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} \left[(H + h_a)^{5/2} - (h_a)^{5/2} \right]$$

where

θ = vertex angle of the notch or weir

10. Discharge through a trapezoidal notch or weir is given by

$$Q = \left[\frac{2}{3} C_{d_1} \sqrt{2g} LH^{3/2} + \frac{8}{15} C_{d_2} \sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \right]$$

where

C_{d_1} = coefficient of discharge for the rectangular portion of the notch or weir ;

C_{d_2} = coefficient of discharge for the triangular portion of the notch or weir ;

L = crest length of the notch or weir; and

$\theta/2$ = the angle of inclination of the sides of the notch or weir with the vertical.

If C_d is the coefficeient of discharge for whole of the trapezoidal notch or weir then the discharge is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = C_d\sqrt{2g}H^{3/2}\left[\frac{2}{3}L + \frac{8}{15}H\tan\frac{\theta}{2}\right]$$

11. A Cipolletti weir or notch is a trapezoidal weir or notch having inclination of slopping sides as 1 horizontal to 4 vertical, $\left(\text{i.e., } \frac{\theta}{2} = 14°\right)$. The discharge through a Cipolletti weir is given by :
When velocity of apporach is not taken into account

$$Q = \frac{2}{3}C_d\sqrt{2g}\ L\ H^{3/2}$$

When velocity of approach is not taken into account

$$Q = \frac{2}{3}C_d\sqrt{2g}\ L\left[(H+h_a)^{3/2} - (h_a)^{3/2}\right]$$

12. The error in discharge Q due to the error in the measurement or triangular of head H over a rectangular or triangular notch or weir is given by

$\frac{dQ}{Q} = \frac{3}{2}\frac{dH}{H}$ .... For a rectangular notch or weir

$\frac{dQ}{Q} = \frac{5}{2}\frac{dH}{H}$ .... For a triangular notch or weir

13. Time required to empty a reservoir or tank by a rectangular or a triangular weir is given by

$$t = \frac{3A}{C_d\sqrt{2g}L}\left[\frac{1}{\sqrt{H_2}} - \frac{1}{\sqrt{H_1}}\right] \text{ ....By a rectangular weir}$$

$$t = \frac{5A}{4C_d\sqrt{2g}\tan\frac{\theta}{2}}\left[\frac{1}{(H_2)^{3/2}} - \frac{1}{(H_1)^{3/2}}\right]$$

....By a triangular weir

where

A = cross-sectional area of the reservoir or tank ;
H_1 = initial height of the liquid surface above the crest or apex of the weir ; and
H_2 = final height of the liquid surface above the crest or apex of the weir.

14. Discharge over a broad crested weir is given by

$$Q = C_d LH\sqrt{2g(H-h)}$$

where

H = height of water surface above the crest of the weir on the upstream side ;
h = height of water surface above the crest of the weir at the middle of the weir ; and
L = length of the crest of the weir

The condition for maximum discharge over a broad crested weir is $h = \frac{2}{3}H$; and the maximum discharge is given by

$$Q = 1.70\ C_d LH^{3/2}$$

15. Discharge through a submerged or drowned weir is given by

$$Q = Q_1 + Q_2$$

where Q_1 and Q_2 are the discharges through the free and the drowned portions respectively.
The values of Q_1 and Q_2 are given by :
When velocity of approach is not taken into account

$$Q_1 = \frac{2}{3}C_{d_1}\sqrt{2g}\ L(H_1 - H_2)^{3/2}$$

$$Q_2 = C_{d_2}LH_2\sqrt{2g(H_1 - H_2)}$$

where H_1 and H_2 are respectively the heights of water surface above the crest (or heads) on the upstream and the downstream of the weir ; and C_{d_1} and C_{d_2} are the coefficient of discharge for the free and the drowned portion respectively.
When velocity of approach is taken into account

$$Q_1 = \frac{2}{3}C_{d_1}\sqrt{2g}L\left[(H_1 - H_2 + h_a)^{3/2} - h_a^{3/2}\right]$$

$$Q_2 = C_{d_2}(L \times H_2)\left[\sqrt{2g(H_1 - H_2) + V_a^2}\right]$$

16. Discharge over an Ogee spillway is given by

$$Q = C\,LH^{3/2}$$

where C is the coefficient of the spillway which may be determined by calibration.

17. In the case of siphon spillway the head H under which the water flows is equal to the difference between the water surfaces on the upstream and the downstream sides.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

18. The sides of a proportional weir (or Sutro weir) diverge downward in the form of hyperbolic curves having the equation

$$\frac{2x}{L} = \left[1 - \frac{2}{\pi}\tan^{-1}\sqrt{\frac{y}{a}}\right]$$

where a and L are respectively the height and width of the small rectangular shaped aperture which forms the base of the weir. Discharge through this weir is given by

$$Q = k\left(H - \frac{a}{3}\right)$$

where

$$K = C_d L\,(2ga)^{1/2}$$

PROBLEMS

10.1 What is a weir? How are the weirs classified? What is the difference between a sharp crested and a broad crested weir?

10.2 What is a notch? How are the notches classified?

10.3 Explain why ventilation of suppressed rectangular weir is necessary.

10.4 What is a Cipolletti weir? Show that its side slopes of 1 horizontal to 4 vertical are provided for a definite purpose.

10.5 What is suppressed weir and a weir with end contractions?

10.6 Define velocity of approach. How can you account for it while computing the discharge over weirs?

10.7 What is the difference between open spillway and siphon spillway? What are the advantages of a siphon spillway?

10.8 Discuss the various empirical formulae for discharge over weirs.

10.9 A rectangular weir is 2 m long and has a head of 0.675 m. Find the discharge taking into account two end contractions. [*Ans.* 1.90 m^3/s]

10.10 Find the discharge through a triangular notch under a constant head of 0.25 m if the angle of the notch is $120°$. Take $C_d = 0.62$.
[*Ans.* 0.079 m^3/s]

10.11 A rectangular notch has a crest length 1 m; the head over the notch is 0.2 m and the height of the sill above the bed level is 0.15 m. If the width of the channel is 1.2 m, calculate the discharge in cumec, taking into account the velocity of approach. [*Ans.* 0.166 cumec]

10.12 Determine the discharge over a sharp crested weir 4.5 m long with no end contractions, the measured head over the crest being 0.45 m. The width of the approach channel is 7.5 m and its depth below the crest of the weir is 1 m.
[*Ans.* 2.521 m^3/s]

10.13 During a test in a laboratory the water which has passed through a venturi meter flows over a right angled V-notch, the head at the V-notch being registered. The larger diameter of the venturi meter is 0.25 m and the diameter of the throat is 0.1 m. When a steady head over the V-notch of 0.181 m is maintained, the difference of pressure head at the venturi meter is found to be 0.322 5 m of water. Determine the coefficient of venturi meter on the assumption that the V-notch results are correct, the coefficient being 0.6. [*Ans.* 0.987]

10.14 In an experiment on a $90°$ V-notch the flow is collected in a vertical cylindrical tank 0.9 m diameter. It is found that the depth of water in the tank increases by 0.65 m in 16.8 s when the head over the notch is 0.2 m. Determine the coefficient of discharge of the notch. [*Ans.* 0.582]

10.15 A channel is conveying 0.6 m^3/s of water. Assuming that an error of 1.5 mm may be made in measuring the head, determine the percentage error resulting (a) from the use of a right angled triangular weir $C_d = 0.6$; (b) from the use of a suppressed rectangular weir 0.6 m long. [**Hint:** (a) Triangular notch:

$$Q = 0.6 = \frac{8}{15}C_d\sqrt{2g}H^{5/2}$$

$$\therefore \quad H = 0.709 \text{ m}$$

$$\frac{dQ}{Q} = \frac{5}{2}\frac{dH}{H}$$

$$\therefore \text{ Percentage error in } Q = \frac{5}{2}\frac{1.5\times10^{-3}}{0.709} \times 100 = 0.53\%$$

(b) Rectangular notch:
Percentage error in $Q = 0.33\%$]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

10.16 A stream approaching a water fall having a fall of 20 m is gaged by a weir. The measured head over the weir is 0.32 m and the length of the weir is 3 m. The velocity of approach V_a is 0.2 m/s and the head due to this may be supposed to be increased by 1.6 ($V_a^2/2g$). Determine the power available from the fall assuming that 60% of the energy can be used.

[*Ans.* 119.13 kW(162 h.p.)]

10.17 A rectangular notch is to be made to discharge 1.82×10^4 litres per minute with a head over the sill equal to 0.5 of the width of the notch. Neglecting the velocity of approach and allowing for two end contractions, determine the width of the notch and the head of water above the sill. [*Ans.* 0.768 m; 0.384 m]

10.18 A reservoir has an area of 8.5×10^4 m^2 and is provided with a weir 4.5 m long. Find how long will it take for the level at the sill to fall from 0.6 m to 0.3 m. [*Ans.* 3.112 h]

10.19 A maximum discharge of 56 m^3/s from a storage reservoir flows over an open spillway 36 m long. Find the head under which the water flows. If the open spillway is now replaced by a siphon type spillway having a section 0.6 m × 1.2 m with operating head of 6 m, find the number of siphons required. Take C_d = 0.6 in both cases, and average surface area of the reservoir as 6×10^5 m^2. If 0.15 m is the priming depth what will be the amount of the extra water stored.

[*Ans.* 0.917 m; 12 ; 46.02×10^4 m^3]

10.20 The following observations of head and the corresponding discharge were made for a *V*-notch.

Head (*m*)	Discharge cumec
0.05	8.1×10^{-4}
0.075	22.4 × 10–4
0.10	47.6×10^{-4}
0.125	80.3×10^{-4}
0.15	126.6×10^{-4}

Assuming $Q = KH^n$; find K and n.

[*Ans.* 1.585, 2.5]

10.21 A Cipolletti weir has a crest length of 0.25 m. If the head on the crest is 0.15 m, claculate the discharge flowing over it. Take C_d = 0.64.

[*Ans.* 0.027 m^3/s]

10.22 A broad crested weir with flat top is constructed across the entire 2.7 m width of a rectangular channel. If the head on the weir crest is 0.415 m, find the discharge over the weir. Take C_d = 0.97.

[*Ans.* 1.19 m^3/s]

10.23 If the ordinary formula for a rectangular notch without end contractions is used to give the flow across a broad crested weir, what should be the value of the coefficient of discharge in this formula? [*Ans.* 1.082]

10.24 A river with vertical banks is 60 m wide, the depth of flow is 1.5 m and the velocity of flow is 1.2 m/s. A broad crested weir 2.4 m high is constructed across the entire width of the river. Find the head on the weir crest.

[*Ans.* 1.06 m if velocity of approach is neglected; 1.05 m if velocity of approach is considered]

10.25 A rapid stream has a depth of flow 1 m and the velocity of flow of 3.7 m/s. Find the height of a suppressed diversion weir which should be constructed across the stream to raise the water level on the upstream side of the weir by 2 m. Take C_{d_1} = 0.58 and C_{d_2} = 0.80.

[*Ans.* 1.422 m (the weir is not submerged)]

10.26 (a) Derive an expression for discharge through a rectangular notch. Explain, how it is modified to take into account the effect of end contractions and velocity of approach.

(b) Water passing over a rectangular notch flows subsequently over a right angled triangular notch. The length of rectangular notch is 0.6 m and its coefficient is 1.84. If the coefficient for triangular notch is 1.42, what will be its working head, when the head on rectangular notch is 0.15 m? [*Ans.* 0.29 m]

10.27 The discharge in a channel varies from 0.16 m^3/s to 0.74 m^3/s. A contracted rectangular weir is to be constructed across this channel to measure the discharge. Determine the length of the weir such that the measured head will never be less than 0.16 m or greater than one-third of the length of the weir. [*Ans.* 1.38 m]

10.28 In one of the sides of a tank there is an isosceles triangular shaped opening with its apex at the top, height L and apex angle 2θ. If water flows through the opening under a head H above its base such that $H < L$, show that the discharge Q is given by

$$Q = \frac{4}{15} C_d \sqrt{2g} \tan\theta \quad H^{3/2}(5L - 2H)$$

where C_d is coefficient of discharge.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Flow Through Pipes

Chapter 11

11.1 INTRODUCTION

A pipe is a closed conduit which is used for carrying fluids under pressure. Pipes are commonly circular in section. As the pipes carry fluids under pressure, the pipes always run full. The fluid flowing in a pipe is always subjected to resistance due to shear forces between fluid particles and the boundary walls of the pipe and between the fluid particles themselves resulting from the viscosity of the fluid. The resistance to the flow of fluid is in general known as frictional resistance. Since certain amount of energy possessed by the flowing fluid will be consumed in overcoming this resistance to the flow, there will always be some loss of energy in the direction of flow, which however depends on the type of flow. The flow of fluid in a pipe may be either *laminar* or *turbulent*. Since different laws govern these two types of flows in pipes, the same are required to be dealt with separately. In this chapter, however, only some pipe flow problems have been dealt with and the detailed discussion of the two different types of flows has been made in the subsequent chapters.

11.2 TWO TYPES OF FLOW–REYNOLDS' EXPERIMENT

The existence of the two types of flow, viz., laminar and turbulent, was first demonstrated by Osborne Reynolds in 1883, with the help of a simple experiment as described below.

Reynolds' Experiment. Reynolds' experiment consisted essentially of a constant head tank filled with water, a small tank containing dye, a horizontal glass tube provided with a bell-mouthed entrance and a regulating valve as shown in Fig. 11.1. The water was made to flow from the tank through the glass tube into the atmosphere and the velocity of flow was varied by adjusting the regulating valve. A liquid dye having the same specific weight as that of water, was introduced into the flow at the bell-mouth through a small tube.

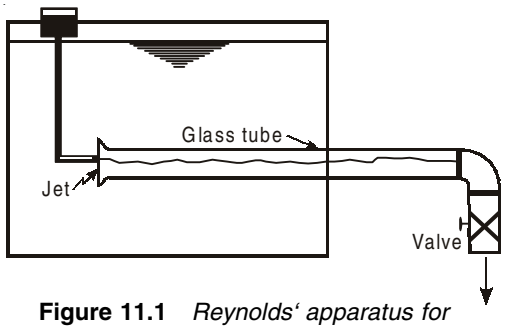


Figure 11.1 *Reynolds' apparatus for demonstrating the type of flow*

From the experiments it was disclosed that when the velocity of flow was low, the dye remained in the form of a straight and stable filament passing through

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the glass tube so steadily that it scarcely seemed to be in motion. With increase in the velocity of flow a critical state was reached at which the filament of dye showed irregularities and began of waver. With a further increase in the velocity of flow the fluctuations in the filament of dye became more intense and ultimately the dye diffused over the entire cross-section of the tube, due to the intermingling of the particles of the flowing fluid. Figure 11.2 shows the different states of the dye filament.

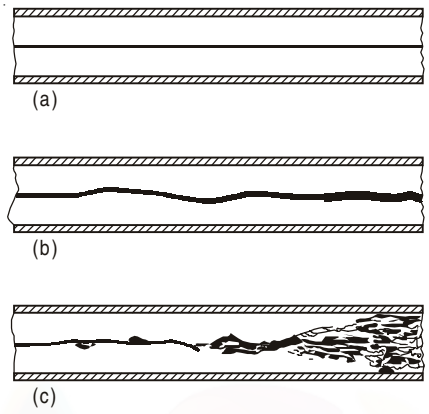


Figure 11.2 *Appearance of dye filament in :* (*a*) *laminar flow,* (*b*) *transition, and* (*c*) *turbulent flow*

Reynolds deduced from his experiments that at low velocities the intermingling of the fluid particles was altogether absent and that the fluid particles moved in parallel layers or laminae, sliding past adjacent laminae but not mixing with them. This is the regime of laminar flow. Since at higher velocities the dye filament diffused through the tube, it was apparent that the intermingling of the fluid particles was occurring, or in other words the flow was turbulent. The velocity at which the flow changes from the laminar to turbulent for the case of a given fluid at a given temperature and in a given pipe is known as *critical velocity.* The state of flow in between these two types of flow is known as 'transitional state' (or flow in transition).

On the basis of his experiments Reynolds discovered that the occurrence of a laminar and turbulent flow was governed by the relative magnitudes of the inertia and the viscous forces. It was indicated by Reynolds that at low velocities of flow, even for the fluids having very small viscosity, the viscous forces become predominant and therefore, the flow is largely viscous in character. However, at higher velocities of flow the inertial forces have predominance over the viscous forces. Reynolds related the inertia to viscous forces and arrived at a dimensionless parameter.

$$\text{Re or } N_R = \frac{\text{Inertia force}}{\text{Viscous force}} = \frac{F_i}{F_v}$$

According to Newton's second law of motion the inertia force F_i is given by

$$\begin{aligned} F_i &= \text{mass} \times \text{acceleration} \\ &= \rho \times \text{volume} \times \text{acceleration} \\ &= \rho \times L^3 \times (L/T^2) = (\rho L^2 V^2) \end{aligned}$$

Similarly viscous force F_v is given by Newton's law of viscosity as

$$\begin{aligned} F_v &= \tau \times \text{area} \\ &= \mu \frac{\partial v}{\partial y} \times L^2 = (\mu VL) \end{aligned}$$

$$\therefore \quad \text{Re or } N_R = \frac{(\rho L^2 V^2)}{\mu VL} = \frac{\rho VL}{\mu}$$

This dimensionless parameter is called *Reynolds number*, in which ρ and μ are respectively the mass density and viscosity of the flowing fluid, V is the characteristic (or representative) velocity of flow and L is the characteristic linear dimension. In the case of flow through pipes the characteristic linear

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

dimension L is taken as the diameter D of the pipe and the characteristic velocity is taken as the average velocity V of flow of fluid. Thus Reynolds number becomes $(\rho DV/\mu)$ or (VD/υ) where $(\mu/\rho) = \upsilon$, is kinematic viscosity of the flowing fluid. The Reynolds number is therefore a very useful parameter in predicting whether the flow is laminar or turbulent. The limiting values of Reynolds number corresponding to which the flow of fluid in a pipe is either laminar or turbulent are given below.

The existence of two flow regimes may also be indicated with the help of another simple experiment as shown in Fig. 11.3. The apparatus required consists of a uniform horizontal pipe of a known diameter, to which a manometer is connected for measuring the loss of head h_f, occurring in a length L, of the pipe. The head loss can be obtained from the manometer reading for a particular discharge and the mean velocity V of flow through pipe can be determined from the measured discharge. Several values of the head loss can thus be obtained for the corresponding values of the velocities of flow of fluid.

Now if a logarithmic plot of (h_f/L) as ordinate and the velocity V as abscissa is prepared, it will be as shown in Fig. 11.4. From this it will be found that for small values of V, the plot is a straight line with its slope equal to unity. This continues upto certain value of V, represented by point B on the figure, which thus indicates that as long as the velocity is less than the value corresponding to point B, the head loss due to friction will be directly proportional to the velocity of flow of fluid (i.e., $h_f/L \sim V$). Beyond the point B with increasing velocity it will be found that there exists certain transition region extending upto point C, during which there is an abrupt increase in the rate at which the loss of head varies. After the region of transition has passed, again the curve obtained is in the form of straight lines with slopes ranging from 1.72 to 2.00.

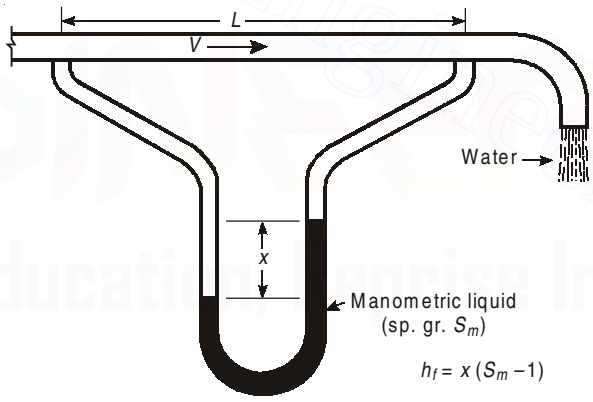


Figure 11.3 *Apparatus for measuring the loss of head in a pipe*

However, if the velocity is gradually reduced from a higher value, the line BC will not be retraced. Instead the points may be along curve CA, as indicated by arrows in Fig. 11.4. The point B is known as the *upper critical point* and the point A is known as the *lower critical point*, and the corresponding velocities are known as *upper critical velocity* and *lower critical velocity* respectively.

It is thus seen that upto point A, the drop in pressure head due to frictional resistance is directly proportional to the mean velocity of flow V, which is the range of laminar flow. Beyond point C, the drop in pressure head due to frictional resistance varies as V^n, where n ranges from 1.72 to 2.0, which is the zone of turbulent flow. Between the points A and C (i.e., in between the regimes of laminar and turbulent flow) lies the transition region as shown in Fig. 11.4.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The upper critical Reynolds number corresponding to point *B* (i.e., upper limit of laminar flow in pipes) was found by Reynolds to lie between 12000 to 14000. But the upper critical Reynolds number is indefinite, being dependent upon initial disturbance affecting the flow, shape of entry of pipe, roughness of pipe wall etc. Thus the practical value of upper critical Reynolds number may be considered to lie between 2700 to 4000.

The lower critical Reynolds number for flow of fluid in pipes corresponding to point *A* is of greater engineering importance as it indicates a condition below which all turbulence entering the flow from any source will be damped out by viscosity and thus sets a limit below which laminar flow will always occur. Experimentally the value of the lower critical Reynolds number has been found to be approximately 2000.

Between Reynolds numbers 2000 and 4000 the transition region exists.

The concept of critical Reynolds number which distinguishes the regimes of laminar and turbulent flow is indeed quite useful in the study of various fluid flow phenomena. Applying this concept to the flow of any fluid in circular pipes, one may predict that the flow will be laminar if Reynolds number is less than 2000 and turbulent if it is greater than 4000. It may however be pointed out that critical Reynolds number is very much a function of boundary geometry. Thus for example in the case of flow between parallel plates, the critical Reynolds number (computed by using mean velocity of flow and the spacing between the plates) is approximately 1000; in the case of flow in a wide open channel the critical Reynolds number (computed by using mean velocity of flow and depth of flow) is approximately 500; and for flow around a sphere the critical Reynolds number (computed by using the approach velocity and diameter of the sphere) is approximately 1.

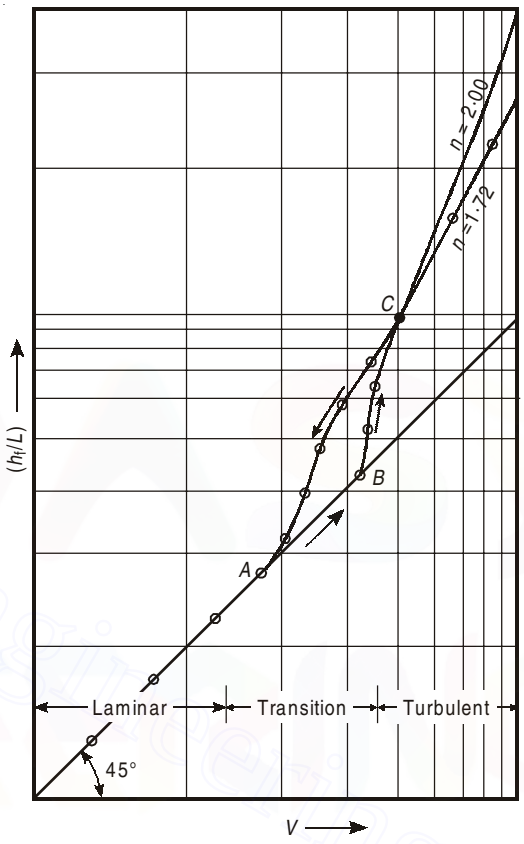


Figure 11.4 *Plot of (h_f/L) v/s V showing upper and lower critical points and velocities*

11.3 LAWS OF FLUID FRICTION

As stated earlier the frictional resistance offered to the flow depends on the type of flow. As such different laws are obeyed by the frictional resistance in the laminar and the turbulent flows. On the basis of the experimental observations the laws of fluid friction for the two types of flows may be narrated as follows.

1. Laws of Fluid Friction for Laminar Flow. The frictional resistance in the laminar flow is as follows

(i) proportional to the velocity of flow,

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(ii) independent of the pressure,
(iii) proportional to the area of surface in contact,
(iv) independent of the nature of the surface in contact,
(v) greatly affected by the variation of the temperature of the flowing fluid.

The first law enunciated above can also be proved analytically as indicated in the next chapter. Further the reason for the frictional resistance in the case of laminar flow being independent of the nature of the surface in contact, is that when a fluid flows past a surface with velocity less than critical velocity, a film of almost stationary fluid is formed over the surface, which prevents the flowing fluid to came in contact with the boundary surface. Similarly in the case of laminar flow the resistance is due to viscosity only and the viscosity of a fluid depends on its temperature.

2. Laws of Fluid Friction for Turbulent Flow. The frictional resistance in the case of turbulent flow is as follow

(i) proportional to (velocity)^n, where the index n varies form 1.72 to 2.0,
(ii) independent of the pressure,
(iii) proportional to the density of the flowing fluid,
(iv) slightly affected by the variation of the temperature of the flowing fluid,
(v) proportional to area of surface in contact,
(vi) dependent on the nature of the surface in contact.

Since mostly the flow of fluids in pipes is turbulent, in the various pipe flow problems dealt with in this chapter the flow is assumed to be turbulent.

11.4 FROUDE'S EXPERIMENTS

W. Froude conducted a series of experiments to investigate frictional resistance offered to the flowing water by different surfaces. The experiments were conducted in a tank about 100 m (300 ft) long, 11 m (36 ft) broad and 3 m (10 ft) deep and containing water. Thin wooden boards about 5 mm (3/16 in) thick, 0.475 m (19 in) wide and lengths varying from 0.6 m (2 ft) to 15 m (50 ft) were towed end wise in this tank by connecting them to a carriage running on rails provided on the sides of the tank. The carriage was hauled along at speeds varying from 30 m (100 ft) to 300 m (1000 ft) per minute, by means of a wire rope passing around a drum. The boards were towed in a completely submerged position such that the upper edge was about 0.45 m $\left(1\frac{1}{2}\text{ft}\right)$ below the water surface in the tank and the force required to tow the board being measured. In order to develop the surfaces of different types the surfaces of the boards were covered with varnish, tinfoil, calico and sand in turn.

From the results of these experiments Froude derived the following conclusions:

(i) The frictional resistance varies approximately with the square of the velocity.

(ii) The frictional resistance varies with the nature of the surface.

(iii) The frictional resistance per unit area of surface decreases as the length of the board increases but is constant for long lengths.

Thus if f' is the frictional resistance per unit area of given surface at unit velocity, A is the area of wetted surface and V is the velocity, then the total friction resistance F is given by

$$F = f'AV^n$$

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Assuming the index $n = 2$

$$F = f'AV^2$$

In the above expression except f' all the terms are known and hence the value of f' may be computed.

11.5 EQUATION FOR HEAD LOSS IN PIPES DUE TO FRICTION–DARCY-WEISBACH EQUATION

Consider a horizontal pipe of cross-sectional area A carrying a fluid with a mean velocity V. Let 1 and 2 be the two sections of the pipe L distance apart, where let the intensities of pressure be p_1 and p_2 respectively. By applying Bernoulli's equation between the sections 1 and 2, we obtain

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + Z_2 + h_f$$

Since $V_1 = V_2 = V$ and $Z_1 = Z_2$

$$\text{Loss of head} = h_f = \frac{p_1}{w} - \frac{p_2}{w}$$

i.e., the pressure intensity will be reduced by the frictional resistance in the direction of flow and the difference of pressure heads between any two sections is equal to the loss of head due to friction between these sections.

Further let f' be the frictional resistance per unit area at unit velocity, then frictional resistance

$$= f' \times \text{area} \times V^n$$
$$= f' \times PL \times V^n$$

where P is the wetted perimeter of the pipe.

The pressure forces at the sections 1 and 2 are (p_1A) and (p_2A) respectively. Thus resolving all the forces horizontally, we have

$$p_1A = p_2A + \text{frictional resistance}$$

or $$(p_1 - p_2)\,A = f' \times PL \times V^n$$

or $$(p_1 - p_2) = f' \times \frac{P}{A} \times LV^n$$

Dividing both sides by the specific weight w of the flowing fluid

$$\frac{p_1 - p_2}{w} = \frac{f'}{w} \times \frac{P}{A} LV^n$$

But $$h_f = \frac{p_1 - p_2}{w}, \text{ then}$$

$$h_f = \frac{f'}{w} \times \frac{P}{A} \times LV^n$$

The ratio of the cross-sectional area of the flow (wetted area) to the perimeter in contact with the fluid (wetted perimeter) i.e., $\left(\frac{A}{P}\right)$ is called *hydraulic mean depth* (H.M.D.) or *hydraulic radius* and it is represented by m or R.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Then $$h_f = \frac{f'}{w} \times \frac{LV^n}{m} \quad ...(11.1)$$

For pipes running full

$$m = \frac{A}{P} = \frac{\left(\frac{\pi D^2}{4}\right)}{(\pi D)} = \frac{D}{4}$$

Substituting this in the equation for h_f and assuming

$$n = 2$$

$$h_f = \frac{4f'}{w}\frac{LV^2}{D}$$

Putting $$\frac{4f'}{w} = \frac{f}{2g}$$

$$h_f = \frac{fLV^2}{2gD} \quad ...(11.2)$$

where f is known as *friction factor*, which is a dimensionless quantity.

Equation 11.2 is known as *Darcy–Weisbach equation* which is commonly used for computing the loss of head due to friction in pipes. It may be noted that the head loss due to friction is also expressed in terms of the velocity head ($V^2/2g$) corresponding to the mean velocity. Further the observations show that the coefficient f is not a constant but its value depends on the roughness condition of the pipe surface and the Reynolds number of the flow. As such in order to determine the loss of head due to friction correctly, it is essential to estimate the value of the factor f correctly. For this purpose on the basis of experimental observations certain relationships have been developed and the same have been represented diagrammatically as shown in Chapter 14, which indicate the manner in which f varies and also facilitate the correct estimation of the value of the friction factor f.

11.6 OTHER FORMULAE FOR HEAD LOSS DUE TO FRICTION IN PIPES

In addition to Darcy–Weisbach equation there are a few more formulae, as indicated below, which are also adopted in the analysis of the pipe flow problems.

1. Chezy's Formula. One of the formula which may be developed to represent the loss of head due to friction in pipe is the Chezy's formula.

The ratio $\left(\frac{h_f}{L}\right)$ represents the slope of the hydraulic grade line or energy grade line and it is usually represented by i or S, i.e., $\left(\frac{h_f}{L}\right) = i$ or S.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Then from Eq. 11.1

$$V^n = \frac{w}{f'} \times m \times \left(\frac{h_f}{L}\right) = \frac{w}{f'}(mi)$$

Assuming $n = 2$ and $C = \sqrt{\frac{w}{f'}}$

$$V = C\sqrt{mi} \text{ or } V = C\sqrt{RS} \quad ...(11.3)$$

Equation 11.3 is known as *Chezy's formula* in which C is known as Chezy's coefficient which is also determined experimentally. However, Chezy's formula is not commonly used in the case of flow through pipes.

In addition to the above derived formulae there are certain empirical formulae as indicated below which are also used for pipes.

2. Manning's Formula. It is one of the most common formula, which is mostly used for the analysis of the problems of flow through channels, but often used for the analysis of the pipe flow problems too. According to this formula the mean velocity of flow V is given by

$$V = \frac{1}{n} R^{2/3} S^{1/2} \quad ...(11.4)$$

where n is the Manning's roughness (or rugosity) coefficient, R is hydraulic radius and S is the slope of the hydraulic grade line or energy grade line. The value of n depends on the nature of the boundary and some of the typical values of n for pipe flow are as given below:

Type of pipe	*Manning's n*
(1) Brass and glass pipe	0.009 to 0.013
(2) Asbestos-cement pipe	0.010 to 0.012
(3) Wrought iron, Welded steel, Wooden stave	0.010 to 0.014
(4) Concrete pipe very smooth	0.011 to 0.012
(5) Concrete pipe with rough joints	0.016 to 0.017
(6) Vitrified sewer pipe	0.013 to 0.015
(7) Rivetted steel pipe	0.013 to 0.017
(8) Corrugated iron pipe	0.020 to 0.022

By comparing the Manning's formula with the Chezy's formula, we obtain

$$C = \frac{1}{n} R^{1/6} \quad ...(11.5)$$

Further for circular pipes, since $R = \left(\frac{D}{4}\right)$, where D is the diameter of the pipe the Manning's formula for pipes may also be written as

$$V = \frac{0.3968}{n} D^{2/3} S^{1/2} \quad ...(11.6)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

3. Hazen–Williams Formula. It is yet another formula which is widely used for designing water supply systems. According to this formula the mean velocity of flow is given by

$$V = 0.85\, C_1\, R^{0.63}\, S^{0.54} \qquad ...(11.7)$$

where C_1 is a coefficient, the value of which depends on the type of the boundary. Some of the values of C_1 are as given below:

Type of pipe	*Value of* C_1
(1) Extremely smooth and straight	140
(2) Very smooth	130
(3) Smooth wood stave, smooth masonry	120
(4) New rivetted steel, vitrified clay	110
(5) Old rivetted steel	95

11.7 OTHER ENERGY LOSSES IN PIPES

When a fluid flows through a pipe, certain resistance is offered to the flowing fluid, which results in causing a loss of energy. The various energy losses in pipes may be classified as:

(i) Major losses.

(ii) Minor losses.

The major loss of energy, as a fluid flows through a pipe, is caused by friction. It may be computed by Darcy–Weisbach equation as indicated earlier. The loss of energy due to friction is classified as a major loss because in the case of long pipelines it is usually much more than the loss of energy incurred by other causes.

The minor losses of energy are those which are caused on account of the change in the velocity of flowing fluid (either in magnitude or direction). In case of long pipes these losses are usually quite small as compared with the loss of energy due to friction and hence these are termed 'minor losses' which may even be neglected without serious error. However, in short pipes these losses may sometimes outweigh the friction loss. Some of the losses of energy which may be caused due to the change of velocity are indicated below:

(a) Loss of energy due to sudden enlargement,

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

(b) Loss of energy due to sudden contraction,

$$h_L = 0.5\frac{V^2}{2g}$$

(c) Loss of energy at the entrance to a pipe,

$$h_L = 0.5\,\frac{V^2}{2g}$$

(d) Loss of energy at the exit from a pipe,

$$h_L = \frac{V^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(e) Loss of energy due to gradual contraction or enlargement,

$$h_L = k\,\frac{(V_1 - V_2)^2}{2g}$$

(f) Loss of energy in bends,

$$h_L = k\,\frac{V^2}{2g}$$

(g) Loss of energy in various pipe fittings,

$$h_L = k\,\frac{V^2}{2g}$$

The expressions for the various minor losses of energy noted above have been derived in Chapter 9 (Section 9.8).

11.8 HYDRAULIC GRADE LINE AND ENERGY GRADE LINE

In the study of flow of fluid in pipes the concept of *hydraulic grade line (H.G.L.)* and *energy grade line* are quit useful. The *energy grade line* is also known as *total energy line (T.E.L.) or total head line.* The *hydraulic* and *energy grade lines* may be obtained as indicated below.

Consider a long pipeline carrying liquid from a reservoir *A* to a reservoir *B* as shown in Fig. 11.5. At several points along the pipeline let piezometers be installed. The liquid will rise in the piezometers to certain heights corresponding to the pressure intensity at each section. The height of the liquid surface above the axis of pipe in the piezometer at any section will be equal to the pressure head (p/w) at that section. On account of loss of energy due to friction, the pressure head will decrease gradually from section to section of the pipe in the direction of flow. If the pressure heads at the different sections of the pipe are plotted to scale as vertical ordinates above the axis of the pipe and all these points are joined by a straight line then as shown in Fig. 11.5, a straight slopping line will be obtained, which is known as *hydraulic grade line (H.G.L)*. Since at any section of the pipe the vertical distance between the pipe axis and the hydraulic grade line is equal to the pressure head at that section, it is also known as *pressure line.* Moreover if Z is the height of the pipe axis at any section above an arbitrary datum as shown in Fig.11.5, then the vertical height of the hydraulic grade line above the datum at that section of the pipe represents the piezometric head equal to $\left(\frac{p}{w} + Z\right)$. As such sometimes the hydraulic grade line is also known as *piezometric head line.*

As shown in Fig. 11.5, for some distance from the entrance section of the pipe the hydraulic grade line is not very well defined. This is so because as the liquid from the reservoir enters the pipe, vena-contracta is formed and a sudden drop in pressure head takes place in this portion of the pipe. The hydraulic grade line in this portion of the pipe is therefore shown by dotted curved line. Further the exit section of the pipe being submerged, the pressure head at this section is equal to the height of liquid surface in the reservoir *B* and hence the hydraulic grade line at the exit section of the pipe will meet the liquid surface in the reservoir *B*.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

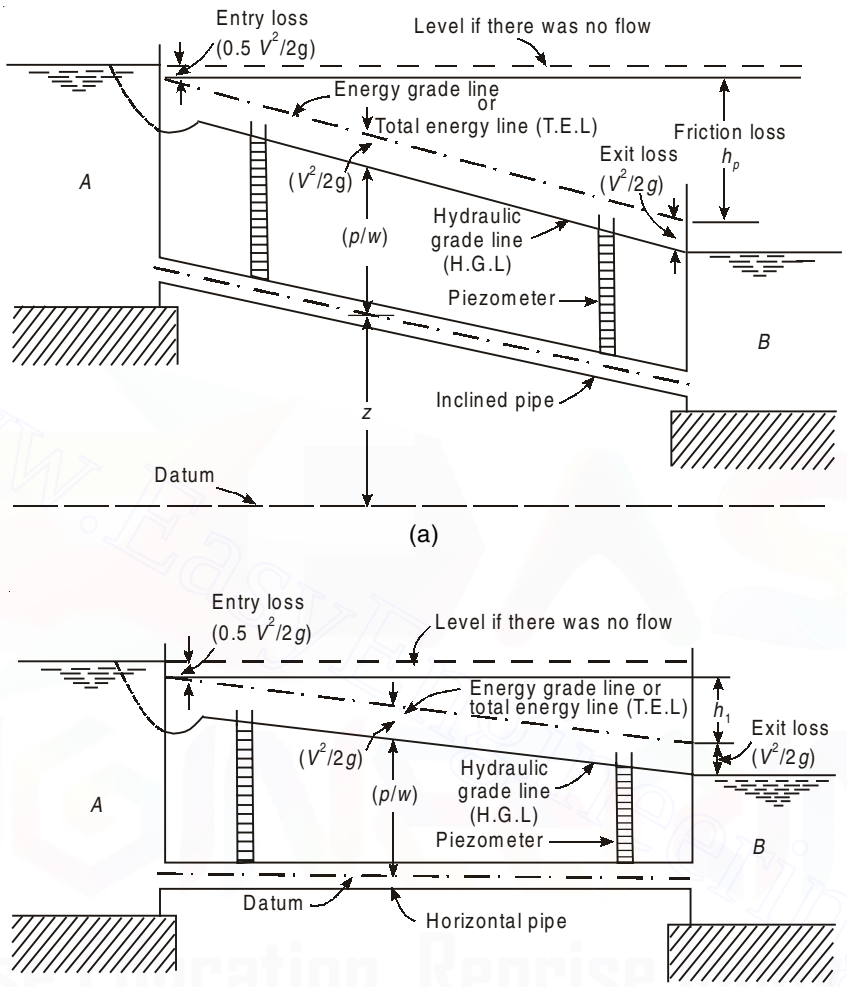


Figure 11.5 *Hydraulic grade line and energy grade line for* : (*a*) *an inclined pipe;* (*b*) *horizontal pipe, connecting two reservoirs*

If at different sections of the pipe the total energy (in terms of head) is plotted to scale as vertical ordinate above the assumed datum and all these points are joined then a straight slopping line will be obtained which is known as *energy grade line* or *total energy line (T.E.L.).* Since total energy at any section of the pipe is equal to the sum of the pressure head (p/w), the datum head Z, and the velocity head ($V^2/2g$), and the vertical distance between the datum and the hydraulic grade line is equal to the piezometric head $\left(\frac{p}{w}+Z\right)$, the energy grade line (or total energy line) will be parallel to the hydraulic grade line with a vertical distance between them equal to ($V^2/2g$). At the entrance section of the pipe there occurs some loss of energy called 'entrance loss' equal to $h_L = 0.5\ (V^2/2g)$ and hence the energy

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

grade line (or total energy line) at this section will lie at a vertical depth equal to 0.5 ($V^2/2g$) below the liquid surface in the reservoir *A*. Similarly at the exit section of the pipe since there occurs an exit loss equal to $h_L = (V^2/2g)$, the energy grade line (or total energy line) at this section will lie at a vertical distance equal to ($V^2/2g$) above the liquid surface in the reservoir *B*. Since at any section of the pipe the vertical distance between the datum and the energy grade line (or total energy line) represents the total energy possessed by the flowing liquid at the section, the vertical distance between the energy grade line (or total energy line) and the horizontal line drawn through the liquid surface in the reservoir *A* will represent the total loss of energy incurred up to that section.

If the pipe line connecting the two reservoirs is horizontal, as shown in Fig. 11.5 (b), then the datum may be assumed to be along the pipe axis only. The piezometric head and the pressure head will them become the same. The hydraulic and energy grade lines for this case may also be obtained in the same manner as shown Fig. 11. 5 (b).

If pipeline carrying liquid from the reservoir *A* discharges freely in the atmosphere at its exit end, then as shown in Fig. 11.6, the hydraulic grade line at the exit end of the pipe will pass through the

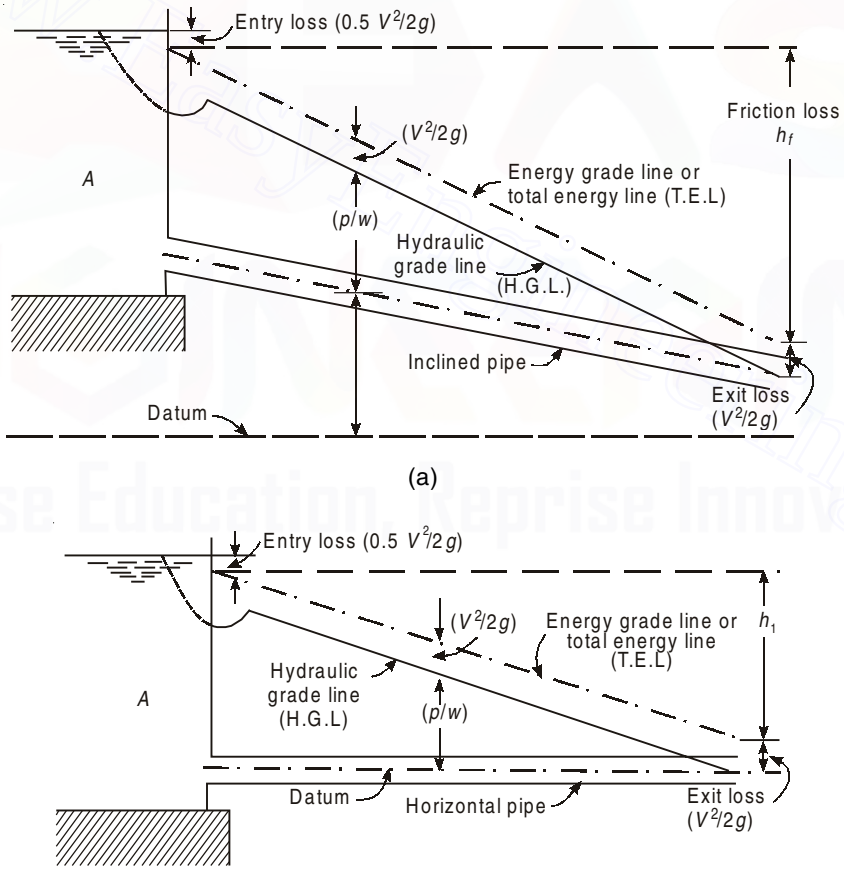


Figure 11.6 *Hydraulic grade line and energy grade line for* (*a*) *an inclined pipe;* (*b*) *horizontal pipe, discharging freely in atmosphere*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

centre line of the pipe since the pressure head at the exit end of the pipe will be zero (being atmospheric). The energy grade line (or total energy line) will again be parallel to the hydraulic grade line and it will be at a vertical distance of $(V^2/2g)$ above the hydraulic grade line.

As the hydraulic and energy grade lines are parallel to each other their slopes are equal. If the pipe is horizontal the slope i of the hydraulic and energy grade lines is equal to the loss of head due to friction h_f divided by the length of pipe L; that is $i = (h_f/L) = [d(p/w)/dL]$. If the pipe is inclined the slope i of the hydraulic and energy grade lines is equal to the loss of head due to friction h_f divided by the length of the horizontal projection of the pipe L'; that is $i = (h_f/L')$. However, if the inclination of the pipe is small the slope I of the hydraulic and energy grade lines may be approximately taken equal to (h_f/L) as in the case of a horizontal pipe.

Hydraulic Gradient and Energy Gradient

The change in piezometric head per unit length of the pipe is known as *hydraulic gradient*. Thus if the pipe is horizontal (for which $(dZ/dL) = 0$) the hydraulic gradient is equal to the slope of the hydraulic grade line [i.e., $d(p/w)/dL$]. However, if the pipe is inclined then the hydraulic gradient is equal to $[d(p/w + Z)/dL]$.

The change in total energy per unit length of the pipe is known as *energy gradient*. Thus if the pipe is horizontal the energy gradient is equal to the slope of the energy grade line [i.e., $(h_f/L) = d(p/w)/dL$]. However, if the pipe is inclined then the energy gradient is equal to $[d(p/w + V^2/2g + Z)/dL]$.

Figure 11.7 illustrates two reservoirs connected by three pipes of different diameters. Due to the introduction of a pipe of smaller diameter in the centre, there will be contraction at the entrance of the smaller pipe and an enlargement at its exit. As such in addition to the friction losses in each of the pipes, there will be entrance loss, contraction loss, enlargement loss and exit loss, and the corresponding

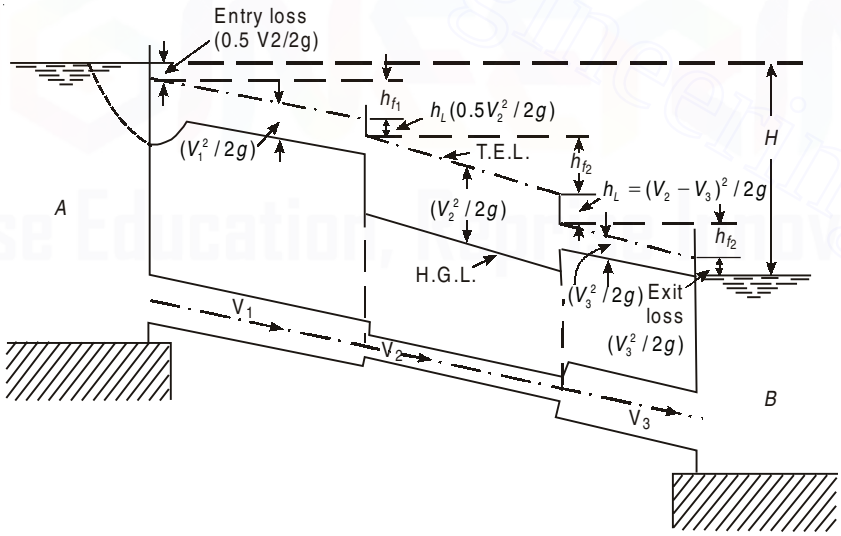


Figure 11.7 *Hydraulic grade line and energy grade line (or total energy line) for pipes of different diameters connected in series.*

hydraulic grade line and the energy grade line (or total energy line) will be as shown in Fig. 11.7. It may however be observed that the hydraulic grade line may even rise in the direction of flow when the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

flow passes from a narrower pipe to a larger pipe, since the velocity in the larger pipe is smaller than that in the smaller pipe and consequently the pressure in the larger pipe is more than that in the smaller pipe. It should, however, be noted that the energy grade line (or total energy line) will not rise in the direction of flow, since there is always a loss of energy in the direction of flow, unless there exists some device, such as pump, in the pipe system which will add energy to the flowing liquid, there by causing an abrupt rise in the hydraulic grade line as well as energy grade line (or total energy line).

11.9 FLOW THROUGH LONG PIPES

Consider a long pipeline of diameter D and length L carrying liquid from a reservoir A to another reservoir B, as shown in Fig. 11.8. Let H_A and H_B be the constant heights of the liquid surfaces in the reservoirs A and B respectively above the centre of the pipe. Further let Z_A and Z_B be the heights of the centres of the pipe ends connected to the reservoirs A and B respectively. Now if V is the mean velocity of flow through the pipe then the head loss due to friction

$$h_f = \frac{fLV^2}{2gD}$$

and head loss at the entrance of pipe

$$= 0.5\frac{V^2}{2g}$$

and head loss at the exit of pipe

$$= \frac{V^2}{2g}$$

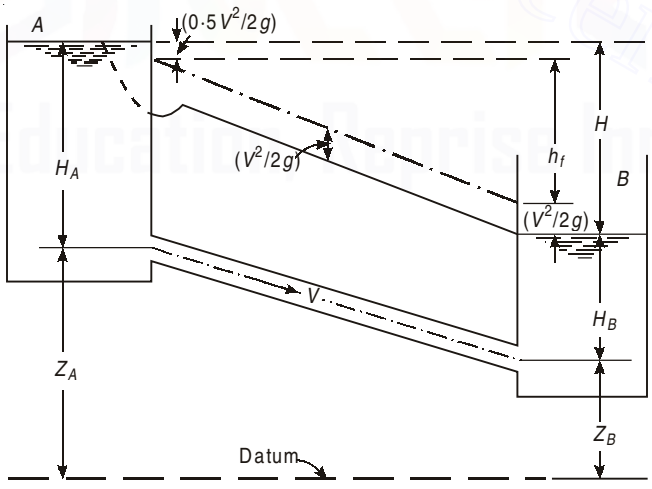


Figure 11.8 *Flow through a long pipe*

Applying Bernoulli's equation between points (1) and (2) in the reservoirs A and B respectively, we obtain

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_A + Z_A = H_B + Z_B + 0.5\frac{V^2}{2g} + \frac{fLV^2}{2gD} + \frac{V^2}{2g}$$

or $$(H_A + Z_A) - (H_B + Z_B) = \frac{V^2}{2g}\left(1.5 + \frac{fL}{D}\right)$$

But $(H_A + Z_A) - (H_B + Z_B) = H$

where H is the difference in the liquid surfaces in the reservoirs A and B. Thus

$$H = \frac{V^2}{2g}\left(1.5 + \frac{fL}{D}\right) \quad ...(11.8)$$

Equation 11.8 indicates that the difference in the liquid surfaces in the two reservoirs at the two ends of the pipe is equal to the sum of the various head losses. From this equation the unknown velocity may be computed.

If the pipe is long (say, more than 1000 times the diameter) the loss of head due to friction will be very large as compared with the minor losses which may then be neglected, thereby simplifying the expression as

$$H = \frac{fLV^2}{2gD}; \text{ or } V = \sqrt{\frac{2gHD}{fL}} \quad ...(11.9)$$

If the pipe in Fig. 11.8, instead of discharging into the reservoir B, discharges into the atmosphere the equation would then be

$$(H_A + Z_A) = Z_B + 0.5\frac{V^2}{2g} + \frac{fLV^2}{2gD} + \frac{V^2}{2g}$$

or $$H = \frac{V^2}{2g}\left(1.5 + \frac{fL}{D}\right)$$

where H is the height of the liquid surface in the reservoir A above the outlet end of the pipe.

11.10 PIPES IN SERIES OR COMPOUND PIPE

If a pipeline connecting two reservoirs is made up of several pipes of different diameters, D_1, D_2, D_3 etc., and lengths L_1, L_2, L_3 etc., all connected in series (i.e., end to end) as shown in Fig. 11.7, then the difference in liquid surface levels is equal to the sum of the head losses in all the sections. Further the discharge through each pipe will be same. Thus with reference to Fig. 11.7 following equations may be obtained

$$H = 0.5\frac{V_1^2}{2g} + \frac{f_1L_1V_1^2}{2gD_1} + \frac{0.5V_2^2}{2g} + \frac{f_2L_2V_2^2}{2gD_2} + \frac{(V_2 - V_3)^2}{2g} + \frac{f_3L_3V_3^2}{2gD_3} + \frac{V_3^2}{2g} \quad ...(11.10)$$

Also

$$Q = \frac{\pi D_1^2}{4}V_1 = \frac{\pi D_2^2}{4}V_2 = \frac{\pi D_3^2}{4}V_3 \quad ...(11.11)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, if the minor losses are neglected as compared with the loss of head due to friction in each pipe, then Eq. 11.10 becomes

$$H = \frac{f_1 L_1 V_1^2}{2gD_1} + \frac{f_2 L_2 V_2^2}{2gD_2} + \frac{f_3 L_3 V_3^2}{2gD_3} \quad ...(11.12)$$

The above noted equations may be used to solve the problems of pipelines in series. There are two types of problems which may arise for the pipelines in series, viz., (a) given discharge Q, to determine the head H, and (b) given H, to determine discharge Q.

In the case of (a) the solution is simple since the discharge and diameters and lengths of the pipes are known, Reynolds numbers can easily be computed and the values of the friction factors f_1, f_2 and f_3 readily found from Moody's diagram (shown in Chapter 14). These can then be substituted in the above equations to determine the value of H.

In the case of (b) first of all the values of f_1, f_2 and f_3 are assumed. For simplicity, these may even be assumed to be equal. Then by substituting in equation for H, and solving by trial the value of discharge Q may be obtained. Using this value of Q the value of Reynolds number for each pipe may be computed and from Moody's diagram the values of f_1, f_2 and f_3 are found. With these new values of f_1, f_2 and f_3 a new value of Q is computed by substitution in the equation for H. Since in actual practice rough turbulent flow* occurs for which the value of f varies very little with Reynolds number and therefore only one or two trials will be necessary to get the correct value of Q.

11.11 EQUIVALENT PIPE

Often a compound pipe consisting of several pipes of varying diameters and lengths is to be replaced by a pipe of uniform diameter, which is known as *equivalent pipe*. The uniform diameter of the equivalent pipe is known as the equivalent diameter of the compound pipe. The size of the equivalent pipe may be determined as follows.

If L_1, L_2, L_3 etc., are the lengths and D_1, D_2, D_3 etc., are the diameters respectively of the different pipes of a compound pipeline, then the total head loss in the compound pipe, neglecting the minor losses, is

$$h_L = \frac{f_1 L_1 V_1^2}{2gD_1} + \frac{f_2 L_2 V_2^2}{2gD_2} + \frac{f_3 L_3 V_3^2}{2gD_3} + ...$$

Again by continuity

$$Q = a_1 V_1 = a_2 V_2 = a_3 V_3 = ...$$

$$= \frac{\pi}{4} D_1^2 V_1 = \frac{\pi}{4} D_2^2 V_2 = \frac{\pi}{4} D_3^2 V_3 = ...$$

Assuming $f_1 = f_2 = f_3 = ... = f$

$$h_L = \frac{f}{2g} \frac{Q^2}{(\pi/4)^2} \left[\frac{L_1}{D_1^5} + \frac{L_2}{D_2^5} + \frac{L_3}{D_3^5} + ... \right]$$

If D is the diameter and L is the length of the equivalent pipe then it would carry the same discharge Q if the head loss due to friction in the equivalent pipe is same as that in the compound pipe. The loss of head due to friction in the equivalent pipe is

* See Chapter 14.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h_L = \frac{fLV^2}{2gD} = \frac{f}{2g}\frac{Q^2}{(\pi/4)^2}\frac{L}{D^5}$$

Thus equating the two head losses, we get

$$\frac{L}{D^5} = \left[\frac{L_1}{D_1^5} + \frac{L_2}{D_2^5} + \frac{L_3}{D_3^5} + ...\right] \quad ...(11.13)$$

Equation 11.13 is known as *Dupuit's equation*, which may be used to determine the size of the equivalent pipe. Thus if the length of the equivalent pipe is equal to the total length of the compound pipe i.e., $L = (L_1 + L_2 + L_3 + ...)$, then the diameter D of the equivalent pipe may be determined by using equation 11.13. Sometimes a pipe of given diameter D which is available may be required to be used as equivalent pipe to replace a compound pipe, in which case the length of the equivalent pipe may be required to be determined and the same may also be determined by using Eq. 11.13.

11.12 PIPES IN PARALLEL

When a main pipeline divides into two or more parallel pipes which again join together downstream and continue as a main line as shown in Fig. 11.9, the pipes are said to be in parallel. The pipes are connected in parallel in order to increase the discharge passing through the main. Such a system is analogous to a parallel electric circuit in which the drop in potential and flow of electric current can be compared to head loss and discharge (or flow rate) in a fluid flow respectively.

Referring Fig. 11.9, the discharge in the main line is equal to the sum of the discharges in each of the parallel pipes.

Thus $$Q = Q_1 + Q_2 \quad ...(11.14)$$

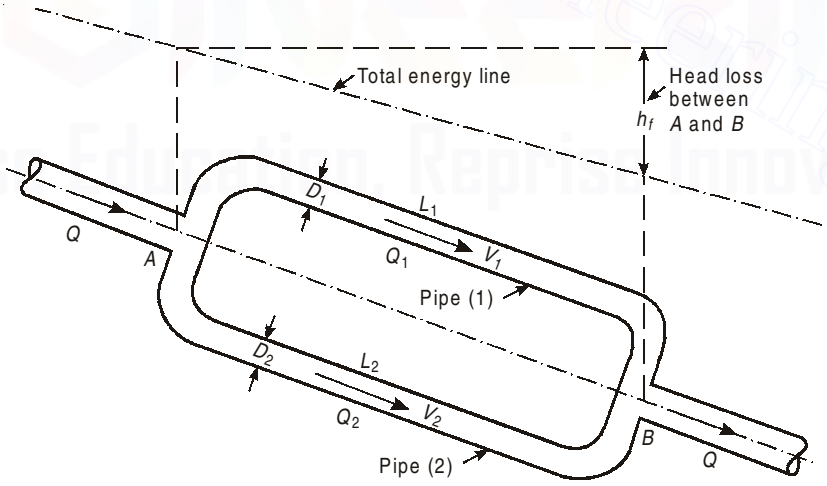


Figure 11.9 *Pipes in parallel*

The flow of liquid in pipes (1) and (2) takes place under the difference of head between the sections A and B and hence the loss of head between the sections A and B will be the same whether the liquid

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

flows through pipe (1) or pipe (2). Thus if D_1, D_2 and L_1, L_2 are the diameters and lengths of the pipes (1) and (2) respectively; then the velocities of flow V_1 and V_2 in the two pipes must be such as to give

$$h_f = \frac{fL_1V_1^2}{2gD_1} = \frac{fL_2V_2^2}{2gD_2} \qquad ...(11.15)$$

assuming the same value of f for each parallel pipe. However, if the values of f are different the same may be introduced in Eq. 11.15.

The various problems which may arise for the pipes in parallel may be solved using these equations.

11.13 FLOW THROUGH A BYE–PASS

A bye-pass is the name given to a diversion provided for a main pipeline in which the main line is tapped at some point by a smaller pipe which later returns to the main at another point as shown in Fig. 11.10. Such diversions are usually required to be provided in city water supply mains for purpose of distribution.

Referring Fig. 11.10, if sections 1 and 2 represent the inlet and the outlet of the diversion then the difference of head between sections 1 and 2 will be

$$H_1 - H_2 = \frac{fLV^2}{2gD} = \frac{flv^2}{2gd} + k'\frac{v^2}{2g}$$

where D, d and L, l are the diameters and the lengths of the main and the diversion respectively, V and v are the velocities of flow in the main and the diversion respectively, $k'\left(\frac{v^2}{2g}\right)$ represents the loss of head at the inlet and outlet sections of the diversion; and f is the friction factor which is assumed to be same for both the main and the diversion. Thus

$$\frac{LV^2}{D} = \frac{lv^2}{d} + kv^2 \left(\text{where } k = \frac{k'}{f}\right)$$

or

$$\frac{V^2}{v^2} = \frac{D}{L}\left(\frac{l}{d} + k\right)$$

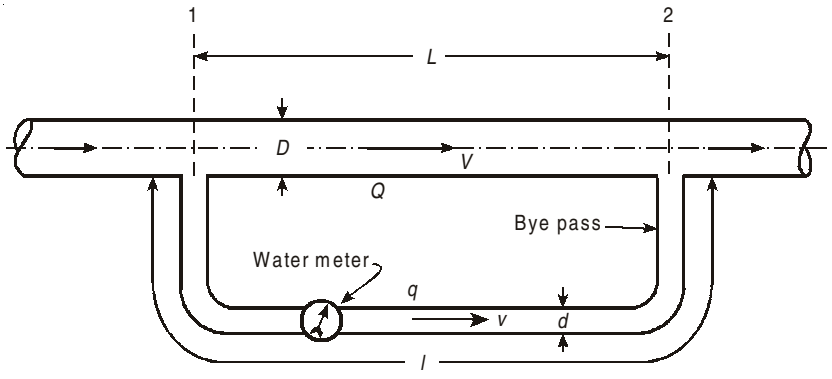


Figure 11.10 *Flow through a bye-pass*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Now if Q and q are the discharges through the main and the diversion respectively, then

$$Q = \frac{\pi D^2}{4} V \text{ and } q = \frac{\pi d^2}{4} v$$

$$\therefore \quad \frac{Q}{q} = \frac{D^2 V}{d^2 v} = \frac{D^2}{d^2}\sqrt{\frac{D}{L}\left(\frac{l}{d}+k\right)}$$

or
$$\frac{Q}{q} = \sqrt{\left(\frac{D}{d}\right)^5 \frac{l+kd}{L}} \qquad ...(11.16)$$

and
$$q = \frac{Q+q}{1+\sqrt{\left(\frac{D}{d}\right)^5 \frac{l+kd}{L}}} \qquad ...(11.17)$$

In the above analysis it is however assumed that no water is drawn from the main as well as diversion pipes between the sections 1 and 2. Then $(Q + q)$ represents the total discharge in the main before section 1 or after section 2.

A bye-pass may also be used for measuring the discharge passing through city water supply mains which are usually of very big size and for these other discharge measuring devices such as water meters etc., are not available. In such cases a bye-pass or diversion pipe containing a water meter is connected between suitable points on the main pipe, and the discharge passing through the bye-pass is determined from the water meter. The bye-pass is previously calibrated and its discharge coefficient determined earlier. The discharge coefficient K of a bye-pass is defined as the ratio between the total discharge Q_t passing through the main (before a part of it is diverted through the bye-pass) and the discharge q passing through the bye-pass. Thus

$$K = \frac{Q_t}{q} = \frac{Q+q}{q}$$

or
$$Q_t = Kq.$$

11.14 BRANCHED PIPES

In a water supply system often a number of reservoirs are required to be interconnected by means of a pipe system consisting of a number of pipes namely main and branches which meet at a junction. Figure 11.11 shows three reservoirs A, B and C interconnected by pipes 1, 2 and 3 which meet at a junction D. In the problems of this type the lengths, diameters, and friction factors of the different pipes are supposed to be known. Further it is assumed that the flow is steady, minor losses are small and hence neglected and the reservoirs are large enough so that their water surface levels are constant. The three basic equations that are used in order to solve these problems are continuity equation, Bernoulli's equation and Darcy–Weisbach equation. At any junction according to continuity equation the total rate of flow towards the junction is equal to the total rate of flow away from it.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Let D_1, D_2, D_3 be the diameters; L_1, L_2, L_3 be the lengths; Q_1, Q_2, Q_3 be the discharges; V_1, V_2, V_3 be the velocities of flow in the pipes 1, 2 and 3 respectively. Now if (p_d / w) is the pressure head at the junction D; Z_1, Z_2, Z_3 are respectively the heights of the water surfaces in the reservoirs A, B, C; Z_d is the height

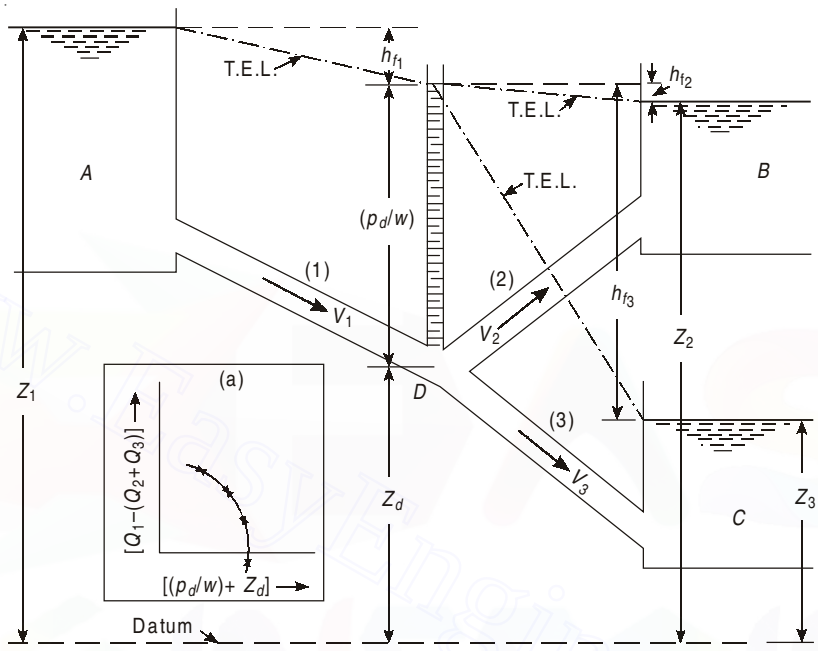


Figure 11.11 *Branched pipes—three interconnected reservoirs*

of the junction D above the assumed datum and h_{f1}, h_{f2}, h_{f3}, are the head losses due to friction in the pipes 1, 2 and 3 respectively, then applying Bernoulli's equation between the reservoir A and the junction D, we get

$$Z_1 = \left(\frac{p_d}{w} + Z_d\right) + h_{f1} \quad \text{...(i)}$$

applying Bernoulli's equation between the junction D and the reservoir B, we get

$$\left(\frac{p_d}{w} + Z_d\right) = Z_2 + h_{f2} \quad \text{...(ii)}$$

applying Bernoulli's equation between the junction D and the reservoir C, we get

$$\left(\frac{p_d}{w} + Z_d\right) = Z_3 + h_{f3} \quad \text{...(iii)}$$

In the above equations $h_{f1} = \dfrac{fL_1V_1^2}{2gD_1}$, $h_{f2} = \dfrac{fL_2V_2^2}{2gD_2}$, $h_{f3} = \dfrac{fL_3V_3^2}{2gD_3}$

assuming friction factor f to be same for all the three pipes, and minor losses being neglected.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further applying continuity equation, we get

$$Q_1 = Q_2 + Q_3$$

$$\frac{\pi}{4}D_1^2V_1 = \frac{\pi}{4}D_2^2V_2 + \frac{\pi}{4}D_3^2V_3 \quad ...(iv)$$

The above noted four equations involve four unknowns *viz.*, Q_1, Q_2, Q_3 and (p_d/w) which may be calculated by solving these four equations.

In deriving the above equations it has been considered that the water is flowing from the junction D to both the reservoirs B and C. This will happen only if the piezometric head at the junction D, $[(p_d/w + Z_d]$ is greater then Z_2 and Z_3. However, if $[(p_d/w) + Z_d] < Z_2$, then the water instead of flowing from the junction D to the reservoir B, it will flow from the reservoir B towards the junction D. In that case both the reservoirs A and B will be supplying water to the reservoir C and the corresponding equations will then be as noted below:

$$Z_1 = \left(\frac{p_d}{w} + Z_d\right) + h_{f1} \quad ...(i)$$

$$Z_2 = \left(\frac{p_d}{w} + Z_d\right) + h_{f2} \quad ...(ii)$$

$$\left(\frac{p_d}{w} + Z_d\right) = Z_3 + h_{f3} \quad ...(iii)$$

$$Q_1 + Q_2 = Q_3 \quad ...(iv)$$

On the other hand if $[(p_d/w) + Z_d] < Z_3$, then the water will flow from the reservoir C towards the junction D. In that case both the reservoirs A and C will be supplying water to the reservoir B and equations may then be modified accordingly. Moreover, in the above noted Eqs (i) to (iii), the minor losses have been neglected, but if these losses are not negligible then these may also be considered and the equations (i) to (iii) may be modified accordingly.

There are three types of problems which may arise in such systems of interconnected reservoirs as indicated below:

(a) Given diameters, friction factors and lengths of all the pipes, water surface elevations in the reservoirs A and B and the discharge Q_1 in pipe 1; to determine the elevation of water surface in reservoir C and the discharge Q_2 and Q_3.

This is a simple and direct problem. With the discharge Q_1 known the head loss h_{f1} can be computed and using Eq. (i) the piezometric head at the junction can be determined. Knowing the piezometric head at the junction, Q_2 can be determined by using Eq. (ii), Q_3 can then be determined by using equation (iv). Knowing Q_3 the elevation of water surface in reservoir C can be determined by using equation (iii).

(b) Given diameters, friction factors and lengths of all the pipes, water surface elevations in the reservoirs A and C and the discharge Q_2 in pipe 2; to determine the elevation of water surface in reservoir B and the discharge Q_1 and Q_3.

From Eqs (i) and (iii), $(h_{f1} + h_{f3}) = (Z_1 - Z_3)$. Thus in this case $(h_{f1} + h_{f3})$ is known and also $(Q_1 - Q_3)$ or $(Q_3 - Q_1)$ is known, but a trial solution is needed for this type of problem. There are two methods of solution, viz., (1) assume trial values of Q_1 and Q_3 and compute h_{f1} and h_{f3} and compare these with the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

known value of $(h_{f1} + h_{f3})$. Repeat the procedure until the computed value of the sum $(h_{f1} + h_{f3})$ is equal to the known value; (2) assume trial values of h_{f1} and h_{f3} and from these compute the discharge Q_1 and Q_3 and compare these with the known value of $(Q_1 - Q_3)$ or $(Q_3 - Q_1)$. Repeat the procedure until the two values are equal. From either of these two methods the value of the piezometric head at the junction D can be determined. From the given discharge Q_2, h_{f2} can be determined. From these two values the water surface elevation in reservoir B can be found by using Eq. (ii).

(c) Given diameters, friction factors and lengths of all the pipes and the water surface elevations of all the three reservoirs; to determine the discharge Q_1, Q_2 and Q_3 in each of the pipes.

For this case also a trial solution is necessary. Assume a trial value of the piezometric head at the junction and compute h_{f1}, h_{f2} and h_{f3} from the above noted equations and then determine Q_1, Q_2 and Q_3. Now depending on the assumed value of the piezometric head at the junction the direction of flow in pipe 2 may be considered. Thus if the flow in pipe 2 is from the junction to the reservoir B then compute the value of $[Q_1 - (Q_2 + Q_3)]$. By assuming some more trial values of the piezometric head and repeating the same procedure, more values of $[(Q_1 - (Q_2 + Q_3)]$ may be obtained. These values of $[Q_1 - (Q_2 + Q_3)]$ may be plotted against the piezometric head at the junction as shown in Fig. 11.11 (a) and the value of the piezometric head for which $[Q_1 - (Q_2 + Q_3)] = 0$ may be obtained which will be the actual value of the piezometric head at the junction. Knowing the piezometric head at the junction, the values of the discharge Q_1, Q_2 and Q_3 may be determined.

It may, however, be stated that if the direction of flow in pipe 2 is incorrectly assumed, then no solution will be obtained. As such in order to obtain the correct direction of flow in pipe 2 a preliminary trial value of the piezometric head at junction $[(p_d/w) + Z_d]$ equal to Z_2 may be assumed, in which case no flow will occur in pipe 2. The values of Q_1 and Q_3 may then be computed corresponding to this value of the piezometric head at the junction. Now if the computed value of Q_1 is greater than that of Q_3, that is, if the flow towards the junction exceeds that leaving the junction, then the actual piezometric head at the junction is greater than Z_2, and the flow in the pipe 2 will be from the junction to the reservoir B. On the other hand, if Q_1 is less than Q_3 then the actual piezometric head at the junction is less than Z_2, and the flow in pipe 2 will be from the reservoir B towards the junction. Obviously for the later case a plot of $[(Q_1 + Q_2) - Q_3]$ versus the piezometric head at the junction will have to be prepared, and the actual value of the piezometric head at the junction will correspond to $[(Q_1 + Q_2) - Q_3] = 0$, as indicated above.

For solving the problems of categories (b) and (c) indicated above there is another method, which eliminates the trial solution. In this method either discharge or velocity of flow in one of the pipes is assumed to be equal to m times the discharge or velocity of flow in the other pipe, where m is a proportionality factor. Then by substituting these values in the above noted (i) to (iv) equations and solving them simultaneously the value of the factor m is determined, from which the values of the other unknown quantities may be determined. This method has been used in the Illustrative Example 11.12.

11.15 SIPHON

A siphon is a long bent pipe which is used to carry water from a reservoir at a higher elevation to another reservoir at a lower elevation when the two reservoirs are separated by a hill or high level ground in between as shown in Fig.11.12. Since the siphon is laid over the hill or the high level ground, for some length from the entrance section it will rise above the water surface in the upper (or supply) reservoir, and then for the remaining length it will drop down to be connected to the lower reservoir.The rising portion of the siphon is known as the '*inlet leg* (or *inlet limb*), the highest point is known as *summit* and the portion between the summit and the lower reservoir is known as *outlet leg* (or

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

outlet limb). As may be seen in Fig. 11.12 the inlet leg (or inlet limb) of a siphon is usually smaller than the outlet leg (or outlet limb). As the siphon is also a long pipe, the loss of head due to friction will be very large and hence the other minor losses may be neglected. Further the length of the siphon may be taken as the length of its horizontal projection. Hence the hydraulic grade line and the energy grade line (or total energy line) for a siphon, as shown in Fig. 11.12, may also be obtained in the same manner

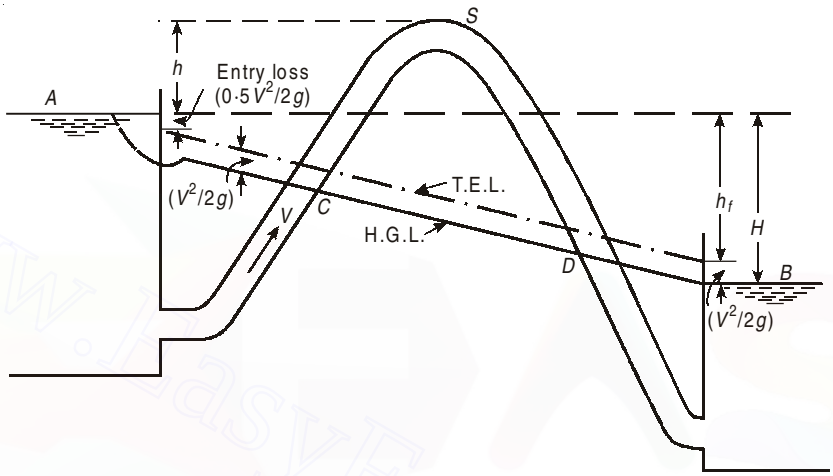


Figure 11.12 *Siphon*

as in the case of an ordinary long pipe. It will be seen from Fig. 11.12 that the hydraulic grade line cuts the siphon at points *C* and *D*, so that some portion of the siphon is above the hydraulic grade line. The vertical distance between the hydraulic grade line and the pipe centre line represents the pressure head at any section. If the hydraulic grade line is above the centre line of the pipe then the pressure is above atmospheric; and if the hydraulic grade line is below the centre line of the pipe, the pressure is negative or below atmospheric. Thus for the portion of the siphon below points *C* and *D* the pressure will be above atmospheric and at points *C* and *D* the pressure of the water flowing in the siphon is equal to atmospheric pressure. For the portion of the siphon between *C* and *D* the pressure will be below atmospheric. As the highest point of the siphon above the hydraulic grade line is the summit *S*, the water pressure at this point is the least. Further as the vertical distance between the summit of the siphon and the hydraulic grade line increases, the water pressure at this point reduces. Theoretically this pressure may be reduced to – 10.3 m of water (if the atmospheric pressure is 10.3 m of water) or absolute vacuum, because this limit would correspond to a perfect vacuum and the flow would stop. However, in practice if the pressure is reduced to about 2.5 m of water absolute or 7.8 m of water vacuum the dissolved air or other gases would come out of the solution and collect at the summit of the siphon in sufficient quantity to form an air-lock, which will obstruct the continuity of the flow, (or the flow will completely stop). A similar trouble may also be caused by the formation of the water vapour in the region of low pressure. Therefore the siphon should be laid so that no section of the pipe will be more than 7.8 m above the hydraulic grade line at the section. Moreover, in order to limit the reduction of the pressure at the summit the length of the inlet leg of the siphon is also required to be limited. This is so because as indicated below, if the inlet leg is very long a considerable loss of head due to friction is caused, resulting in further reduction of the pressure at the summit.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Assuming the siphon to run full, then applying Bernoulli's equation between point A and B, we get

$$H = 0.5\frac{V^2}{2g} + \frac{V^2}{2g} + \frac{fLV^2}{2gD}$$

or
$$H = \frac{V^2}{2g}\left[1.5 + \frac{fL}{D}\right] \quad ...(11.18)$$

where V is the velocity of flow, L and D are the total length and the diameter of the siphon.

Equation 11.18 may be used to determine H if the discharge Q is known; or it may be used to determine Q if H is given. In the first case the Reynolds number of the flow being known the value of f may be determined from Moody's diagram. But in the second case the value of f will be required to be assumed first and then checked for the computed discharge.

In order to determine the maximum value of h, the height of the summit above the water surface in the upper reservoir, Bernoulli's equation is applied between the points A and S, so that

$$\frac{p_a}{w} = \frac{p_s}{w} + \frac{V^2}{2g} + \frac{0.5V^2}{2g} + \frac{flV^2}{2gD} + h$$

where p_a is the atmospheric pressure, p_s is the absolute pressure at the summit, and l is the length of the inlet leg of the siphon.

or
$$h = \frac{p_a}{w} - \frac{p_s}{w} - \frac{V^2}{2g}\left[1.5 + \frac{fl}{D}\right] \quad ...(11.19)$$

As far as possible, the value of h lower than that given by Eq. 11.19 should be used. Further for a given value of h, the length of the inlet leg l should be so adjusted that pressure at the summit is not reduced below the permissible limit as indicated above.

11.16 LOSS OF HEAD DUE TO FRICTION IN TAPERING PIPE

A tapering pipe is the one which has a gradually varying cross-sectional area as shown in Fig. 11.13. Since the loss of head due to friction depends on the velocity of flow, which in the case of tapering pipe varies from section to section, the expression for the loss of head due to friction in such a pipe may be obtained by integration, as indicated below.

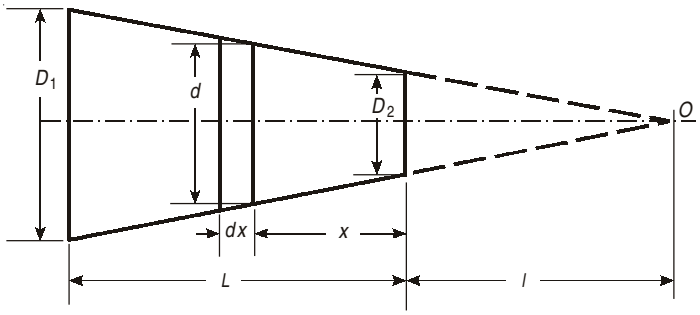


Figure 11.13 *Flow through a tapering pipe*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Let D_1 and D_2 be the diameters at the ends of a tapering pipe of length L, and Q be the discharge through the pipe. Now if O is the apex as shown in Fig. 11.13 and l is the distance of the apex from the end of the pipe with smaller diameter, then from similar triangles, we obtain

$$\frac{D_1}{D_2} = \frac{L+l}{l}, \text{ or } \frac{D_1}{D_2} = 1+\frac{L}{l}$$

$$\therefore \quad l = \frac{D_2 L}{D_1 - D_2} \qquad ...(11.20)$$

Consider a small portion of the pipe of length dx, at a distance x from the smaller end of the pipe. Let the diameter of the pipe at this section be d, then again from the similar triangles, we have

$$\frac{D_2}{d} = \frac{l}{l+x}; \text{ or } d = D_2\left(\frac{l+x}{l}\right)$$

Now the loss of head due to friction for the small length dx of the pipe is given by

$$dh_f = \frac{fdxv^2}{2gd}$$

where v is the velocity of flow at this section given by

$$v = \frac{Q}{(\pi d^2/4)}$$

$$\therefore \quad dh_f = \frac{16fQ^2dx}{2g\pi^2 d^5}$$

$$= \frac{16fQ^2}{2g\pi^2}\frac{l^5 dx}{D_2^5(l+x)^5}$$

By integrating both sides of the above equation, we get

$$h_f = \frac{16fQ^2l^5}{2g\pi^2 D_2^5}\int_0^L \frac{dx}{(l+x)^5}$$

$$= \frac{16fQ^2l^5}{2g\pi^2 D_2^5}\left[-\frac{1}{4}\frac{1}{(l+x)^4}\right]_0^L$$

$$= \frac{4fQ^2l^5}{2g\pi^2 D_2^5}\left[\frac{1}{l^4} - \frac{1}{(L+l)^4}\right] \qquad ...(11.21)$$

Equation 11.21 gives the loss of head due to friction in a tapering pipe in which the value of l is given by Eq. 11.20.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

11.17 LOSS OF HEAD DUE TO FRICTION IN A PIPE WITH SIDE TAPPINGS

If a pipe of uniform diameter is provided with side tappings along its length, through which the liquid may be withdrawn from the pipe, then the velocity of flow will vary from section to section along the length of the pipe. The loss of head due to friction, which depends on the velocity of flow, will also then vary and it may be obtained by integration as described below.

If the side tappings are uniformly and closely spaced and are assumed to remove the liquid at a uniform rate of q per unit length of the pipe then the rate of flow at a distance x from the inlet of the pipe is $(Q - qx)$, when Q is the initial rate of flow at the inlet end. Now for a uniform cross-sectional area A of the pipe

$$\frac{Q}{V} = A = \frac{(Q - qx)}{v}$$

or
$$v = V\left(1 - \frac{qx}{Q}\right)$$

where V and v are respectively the velocities of flow at the inlet and at section x distance from the inlet. The above equation indicates that in this case the velocity of flow varies linearly along the length of the pipe.

Considering a small portion of the pipe of length dx at a distance x from the inlet, the loss of head due to friction for the small length dx of the pipe is given by

$$dh_f = \frac{f dx v^2}{2gD}$$

where D is the diameter of the pipe. Substituting the value of v

$$dh_f = \frac{f}{D}\frac{V^2}{2g}\left(1 - \frac{qx}{Q}\right)^2 dx$$

By integrating the above expression from $x = 0$ to $x = L$, the loss of head due to friction over the entire length L of the pipe is obtained as

$$h_f = \frac{f}{D}\frac{V^2}{2g}\int_0^L \left(1 - \frac{qx}{Q}\right)^2 dx$$

or
$$h_f = \frac{f}{D}\frac{V^2}{2g}\left[-\frac{Q}{3q}\left(1 - \frac{qx}{Q}\right)^3\right]_0^L$$

$$\therefore \quad h_f = \frac{f}{D}\frac{V^2}{2g}\frac{Q}{3q}\left[1 - \left(1 - \frac{qL}{Q}\right)^3\right] \qquad ...(11.22)$$

where f is assumed constant.

If the end of the pipe is closed so that $(Q - qL) = 0$, then Eq. 11.22 becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h_f = \frac{f}{D}\frac{V^2}{2g}\frac{Q}{3q}$$

$$= \frac{f}{D}\frac{V^2}{2g}\frac{L}{3} \qquad ...(11.23)$$

Equation 11.23 shows that the total loss of head in this case is one-third of what it would be if the inlet discharge Q remains constant and no liquid is drawn off along the length of the pipe.

11.18 TIME OF EMPTYING A RESERVOIR THROUGH PIPE

Let a reservoir or tank be emptied by means of a long pipe of length L and diameter D, which is assumed to be discharging freely in the atmosphere. Let the area of water surface in the reservoir be constant equal to A, and the height of the water level above the outlet of the pipe be H_1. It is required to find the time taken to lower the water level in the reservoir from H_1 to H_2 above the pipe outlet.

Let at any instant the water level be h above the outlet of the pipe, when the velocity of flow in the pipe be v. Let the water level fall by a small amount dh in time dt. Then the quantity of water flowing from the reservoir equals the quantity passing along the pipe in time dt; thus

$$-Adh = \frac{\pi D^2}{4} v\, dt \qquad ...(i)$$

The negative sign is being introduced because as time increases the water level in the reservoir is decreasing.

Again $$h = \frac{0.5v^2}{2g} + \frac{v^2}{2g} + \frac{fLv^2}{2gD}$$

from which $$v = \sqrt{\frac{2gh}{1.5 + \frac{fL}{D}}}$$

Substituting this value of v in Eq. (i)

$$-Adh = \frac{\pi D^2}{4}\sqrt{\frac{2gh}{\left(1.5 + \frac{fL}{D}\right)}}\, dt$$

$$\therefore \quad dt = \frac{-4A\sqrt{1.5 + \frac{fL}{D}}}{\pi D^2\sqrt{2g}}\frac{dh}{\sqrt{h}}$$

By integrating the above equation, the total time

$$t = -\frac{4A\sqrt{1.5 + \frac{fL}{D}}}{\pi D^2\sqrt{2g}}\int_{H_1}^{H_2}\frac{dh}{\sqrt{h}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$t = \frac{8A\sqrt{1.5+\frac{fL}{D}}}{\pi D^2\sqrt{2g}}\left(H_1^{1/2}-H_2^{1/2}\right) \quad ...(11.24)$$

If the pipe, instead of discharging freely in the atmosphere, is connected to a reservoir, then the time required to reduce the difference between the water levels in the two reservoirs from H_1 to H_2 may be determined as follows:

Let A_1 and A_2 be the cross-sectional areas of the two tanks A and B, connected by a pipe of length L and diameter D. Let at any instant the difference in the water levels in the two reservoirs be h and in time dt the water level in the tank A fall by an amount dH, Fig. 11.14.

Then, the water level will rise in tank B by an amount $\left(dH\frac{A_1}{A_2}\right)$. Now if dh is the change in the difference in head causing the flow, then

$$dh = dH + dH\ \frac{A_1}{A_2}$$

or
$$dH = \frac{dh}{\left(1+\frac{A_1}{A_2}\right)} \quad ...(i)$$

Further the quantity of water flowing from tank A equals the quantity flowing along the pipe in time dt, thus

$$-A_1 dH = \frac{\pi D^2}{4}\,v\,dt \quad ...(ii)$$

Further
$$h = 0.5\ \frac{v^2}{2g} + \frac{v^2}{2g} + \frac{fLv^2}{2gD}$$

$$v = \sqrt{\frac{2gh}{1.5+\frac{fL}{D}}} \quad ...(iii)$$

Introducing Eqs. (i) and (iii) in (ii), we have

$$-A_1\frac{dh}{\left(1+\frac{A_1}{A_2}\right)} = \frac{\pi D^2}{4}\sqrt{\frac{2gh}{1.5+\frac{fL}{D}}}\,dt$$

or
$$dt = -\frac{4A_1A_2\left(\sqrt{1.5+\frac{fL}{D}}\right)dh}{\pi D^2(A_1+A_2)\sqrt{2g}\ \sqrt{h}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

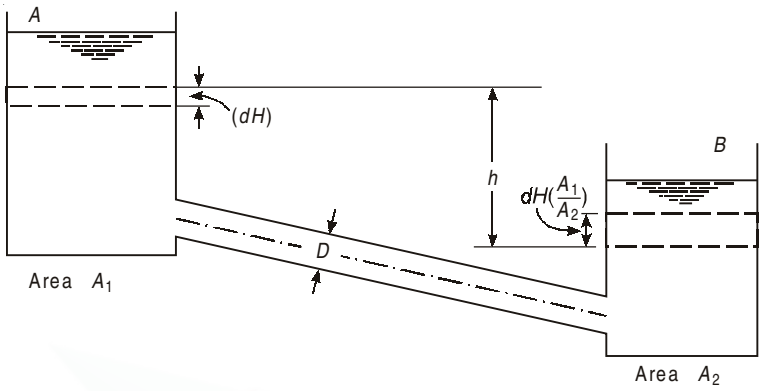


Figure 11.14 *Flow from one reservoir to other through a long pipe*

By integrating the above equation the total time

$$t = -\frac{4A_1A_2\sqrt{1.5+\frac{fL}{D}}}{\pi D^2(A_1+A_2)\sqrt{2g}}\int_{H_1}^{H_2}\frac{dh}{\sqrt{h}}$$

or
$$t = \frac{8A_1A_2\sqrt{1.5+\frac{fL}{D}}}{\pi D^2\left(A_1+A_2\right)\sqrt{2g}}\left(H_1^{1/2}-H_2^{1/2}\right) \quad ...(11.25)$$

11.19 TRANSMISSION OF POWER THROUGH PIPES

The pipes carrying water under pressure from one point to other may be utilised to transmit hydraulic power. Such transmission of power through pipes carrying water or other liquids is commonly used for the working of several hydraulic machines. The hydraulic power transmitted by a pipe however depends on the discharge passing through the pipe and the total head of the water. As the water flows along the pipe it will be subjected to frictional resistance causing a loss of head due to friction. Thus if H is the total head supplied at the entrance to the pipe and h_f is the loss of head due to friction then the head available at the outlet of the pipe (neglecting the minor losses)

$$= (H - h_f)$$

Further if Q is the discharge through the pipe, V is the velocity of flow and L and D are the length and the diameter of the pipe respectively, then

$$h_f = \frac{fLV^2}{2gD}$$

and
$$Q = \frac{\pi D^2}{4}V$$

The power (or energy per second) available at the outlet of the pipe is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$P = \text{weight of water per second} \times \text{head available}$$

or $$P = wQ\,(H - h_f)$$

or $$P = w\left(\frac{\pi D^2}{4} \times V\right)\left(H - \frac{fLV^2}{2gD}\right) \quad \text{....(11.26)}$$

For a given pipe supplied with a given total head H at the entrance, the condition for the maximum power transmitted through the pipe may be obtained by differentiating Eq. 11.26 with respect to V and equating it to zero; thus

$$\frac{dP}{dV} = w\left(\frac{\pi D^2}{4}\right)\left(H - \frac{3fLV^2}{2gD}\right) = 0$$

or $$H - \frac{3fLV^2}{2gD} = 0$$

or $$H - 3h_f = 0$$

$$\therefore \quad H = 3h_f \quad \text{...(11.27)}$$

That is, the power transmitted through a pipe is maximum when the loss of head due to friction is one-third of the total head supplied.

The efficiency of power transmission through pipe may be expressed as

$$\eta = \frac{H - h_f}{H} \quad \text{...(11.28)}$$

Thus corresponding to the maximum power transmitted the efficiency of power transmission is $\eta = (2/3)$ or 66.7%. It may however be observed that the efficiency corresponding to maximum power transmitted is not maximum. This is so because the efficiency of power transmission would be maximum (equal to 100%), if $h_f = 0$ and as h_f increases the efficiency of power transmission decreases.

11.20 FLOW THROUGH NOZZLE AT THE END OF PIPE

A nozzle is a gradually converging short tube which is fitted at the outlet end of a pipe for the purpose of converting the total energy of the flowing water into velocity energy. As such nozzles are used where higher velocities of flow are required to be developed. For example in practice higher velocity of flow is required for extinguishing fire, which is obtained by fitting a nozzle at the end of hose. Further for the impulse type of turbines such as Pelton wheel turbines, it is required to convert whole of the hydraulic energy into kinetic (or velocity) energy and the same is obtained by fitting a nozzle at the end of the pipe (or penstock as is generally called).

Let a nozzle be fitted at the end of a pipe connected to a reservoir with its water level at a height H above the centre line of the nozzle. Let D and L be the diameter and the length of the pipe (upto the base of the nozzle) respectively, and V be the velocity of flow in the pipe. Let d be the diameter of the nozzle at the outlet end and v be the velocity of the issuing jet from nozzle. Thus if Q is the discharge passing through the pipe, then by continuity equation

$$Q = AV = av$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{V}{v} = \frac{a}{A} \qquad ...(i)$$

or $$Q = \frac{\pi D^2}{4} V = \frac{\pi d^2}{4} v$$

Further neglecting the minor losses and the loss of energy in the nozzle, we have

$$H = \frac{fLV^2}{2gD} + \frac{v^2}{2g} \qquad ...(11.29)$$

Substituting the value of V from (i)

$$H = \frac{fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g} + \frac{v^2}{2g}$$

or $$v = \sqrt{\frac{2gH}{1+\frac{fL}{D}\frac{a^2}{A^2}}} \qquad ...(11.30)$$

The kinetic energy of the jet issuing from the nozzle

$$= \frac{1}{2}\frac{wQ}{g} v^2 = \frac{wav^3}{2g}$$

Efficiency of power transmission

$$\eta = \frac{\text{head transmitted}}{\text{head supplied}}$$

$$= \frac{\frac{v^2}{2g}}{H} = \frac{v^2}{2gH} = \left[\frac{1}{1+\frac{fL}{D}\frac{a^2}{A^2}}\right]$$

(a) Maximum Power Available from a Nozzle. Power transmitted by the jet issuing from a nozzle is

$$P = w(av)\frac{v^2}{2g}$$

Substituting the value of $\left(\frac{v^2}{2g}\right)$ from Eq. 11.29, we get

$$P = w(av)\left[H - \frac{fLV^2}{2gD}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the value of V from (i), we get

$$P = w(av)\left[H - \frac{fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g}\right]$$

or
$$P = wa\left[Hv - \frac{fL}{D}\frac{a^2}{A^2}\frac{v^3}{2g}\right]$$

Again the condition for the maximum power transmitted by the nozzle may be obtained by differentiating the above expression with respect to v and equating it to zero; thus

$$\frac{d(P)}{dv} = wa\left[H - \frac{3fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g}\right] = 0$$

or
$$H - \frac{3fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g} = 0$$

or
$$H = \frac{3fLV^2}{2gD} = 3h_f \qquad ...(11.31)$$

That is, the power transmitted by a nozzle is maximum when the head supplied is equal to three times the loss of head due to friction in the pipe.

(b) Diameter of the Nozzle for Transmitting Maximum Power. For maximum transmission of power, from Eq. 11.31

$$H = 3h_f = \frac{3fLV^2}{2gD}$$

Substituting for H from Eq. 11.29, we get

$$\frac{fLV^2}{2gD} + \frac{v^2}{2g} = \frac{3fLV^2}{2gD}$$

or
$$\frac{fLV^2}{2gD} = \frac{1}{2} \times \frac{v^2}{2g}$$

Substituting the value of V from (i), we get

$$\frac{fL}{2gD}\frac{a^2}{A^2}v^2 = \frac{1}{2} \times \frac{v^2}{2g}$$

or
$$\frac{2fL}{D} = \frac{A^2}{a^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad \frac{A}{a} = \sqrt{\frac{2fL}{D}} \qquad \text{...(11.32)}$$

Equation 11.32 gives the ratio between the areas of the supply pipe and the nozzle for maximum power transmission.

11.21 WATER HAMMER IN PIPES

When the water flowing in a long pipe is suddenly brought to rest by closing the valve or by any similar cause, there will be a sudden rise in pressure due to the momentum of the moving water being destroyed. This causes a wave of high pressure to be transmitted along the pipe which creates noise known as *knocking*. This phenomenon of sudden rise in pressure in the pipe is known as *water hammer* or *hammer blow*. The rise in pressure in some cases may be so large that the pipe may even burst and therefore it is essential to take into account this pressure rise in the design of the pipes. The magnitude of the pressure rise depends on the speed at which the valve is closed, the velocity of flow, the length of the pipe, and the elastic properties of the pipe material as well as that of the flowing fluid.

The valve may be closed either gradually or instantaneously and according to the nature of the closure of the valve the expressions for calculating the pressure head due to water hammer may be developed as indicated below.

1. Gradual Closure of Valve. Consider a long pipe of length L and of cross-sectional area a, carrying liquid flowing with a velocity V. Due to the closing of the valve, let the liquid be brought to rest in time t seconds. The total mass of liquid contained in the pipe is $\frac{w}{g}(a \times L)$.

It is however assumed that the rate of closure of the valve is so adjusted that the liquid column in the pipe is brought to rest with a uniform retardation; from an initial velocity V to O in time t seconds.

Thus rate of retardation $= \frac{V}{t}$.

The axial force available for producing retardation

$$= \text{inertia pressure} \times \text{cross-sectional area}$$
$$= (p_i \times a)$$

But

$$\text{force} = \text{mass} \times \text{acceleration (or retardation)}$$

$$\therefore \quad p_i \times a = \frac{waL}{g} \times \frac{V}{t}$$

$$p_i = \frac{wLV}{gt} \qquad \text{...(11.33)}$$

$$\text{Inertia head} = \frac{p_i}{w} = \frac{LV}{gt} \qquad \text{...(11.34)}$$

2. Instantaneous Closure of Valve. Equation 11.34 shows that if the time of closure t is zero, the inertia head should rise to infinity. But the experimental observations indicate that even for a very rapid closure of the valve, the pressure rise is quite finite and measurable. It may therefore be concluded that in the derivation of Eq. 11.34 certain essential factors have been omitted. These factors are (i) the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

compressibility of the liquid; and (ii) the elasticity of the pipe material; and both these factors make it impossible for the liquid column to be stopped instantaneously even if it was possible to close the

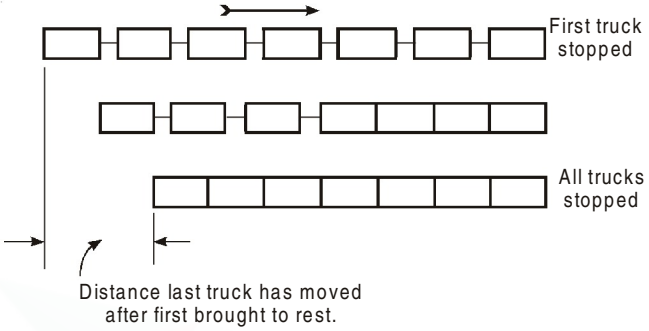


Figure 11.15 *Stopping of a loosely coupled train*

valve instantaneously. This may however be explained by means of a simple example. As shown in Fig. 11.15, if the head of a loosely coupled trucks is suddenly checked the following trucks still keep on moving until all the gap between them has been covered and only then the whole train will come to rest. That means the last truck will move through a certain distance even after the first truck has stopped, before it also finally comes to rest.

With this analogy, the actual sequence of what happens in a pipe, when the valve at its outlet end is instantaneously closed, may be studied. Fig. 11.16 (a) shows the normal flow condition of the liquid in the pipe with normal velocity V, under normal pressure p, at the moment just before the valve at end B is instantaneously shut. Since the friction head will be quite small in comparison with the inertia head, it has been neglected. Fig. 11.16 (b) shows the condition immediately after the valve at B is

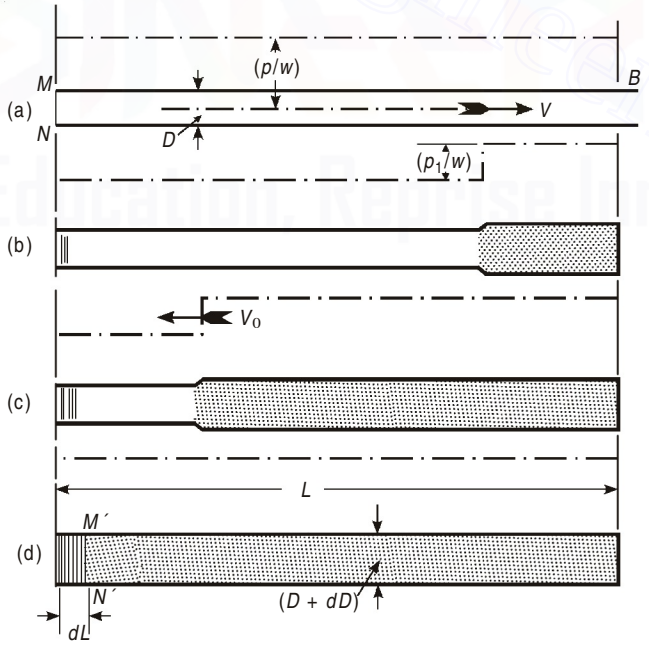


Figure 11.16 *Transmission of pressure wave along a pipe due to instantaneous closure of valve*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

closed. Thus a wave of inertia pressure, as shown by the step in the hydraulic grade line, has begun to travel with velocity V_0 in the upstream direction, which results in compressing the water and also expanding the pipe. Meanwhile the liquid on the left of the advancing wave continues to move on as though nothing has happened to it. Figure 11.16 (c) shows that the wave has advanced further; and Fig. 11.16 (d) shows that the liquid in the entire pipe is at rest under a pressure ($p + p_i$); the whole of the pipe having been distended under this pressure.

Thus if dt is the time required to stop the liquid column, then in this interval of time a transverse plane MN at the end of the column has been able to move through a distance dL to a new position $M'N'$; in other words, the compression of the liquid column together with the increase in the pipe diameter have made space for an amount of liquid dQ (shown by hatching) to enter the pipe after the valve has been shut.

Let
dq_c = the volume by which the liquid is compressed due to pressure p_i
dq_e = the additional volume provided by the stretching of the pipe walls under pressure p_i
dD = the resulting increase in pipe diameter
dt = time for pressure wave to traverse the pipe
T = the wall thickness of pipe
E = Young's modulus for the material of the pipe
K = Bulk modulus for the liquid
f_t = tensile hoop stress in pipe walls due to pressure P_i
$\frac{1}{m}$ = Poisson's ratio

Now
$$dQ = \frac{\pi D^2}{4} \times dL$$

$$dq_c = \frac{p_i}{K} \frac{\pi}{4} D^2 L$$

and
$$dq_e = \left[\frac{\pi}{4}(D + dD)^2 - \frac{\pi}{4} D^2\right] L \, \underline{\Omega} \, \pi \, D \frac{dD}{2} L$$

The strain in the direction of hoop stress or circumferential strain is given by

$$\frac{dD}{D} = \frac{f_t}{E}\left(1 - \frac{1}{2m}\right)$$

as hoop stress
$$f_t = \frac{p_i D}{2T}$$

$$\frac{dD}{D} = \frac{p_i D}{2TE}\left(1 - \frac{1}{2m}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Therefore $$dD = \frac{p_i D^2}{2TE}\left(1-\frac{1}{2m}\right)$$

and $$dq_e = \frac{p_i D^2}{2TE}\left(1-\frac{1}{2m}\right)\times\frac{\pi DL}{2}$$

But $$dQ = dq_c + dq_e$$

Thus by substituting the values noted above, we get

$$\frac{\pi D^2}{4}\times dL = \frac{p_i}{K}\frac{\pi}{4}D^2L + \frac{p_i D^2}{2TE}\left(1-\frac{1}{2m}\right)\times\frac{\pi DL}{2}$$

or $$dL = p_i L\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]$$

But $$dL = V\,dt$$

$\therefore$ $$p_i = \frac{Vdt}{L\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]} \quad ...(i)$$

This expression still contains one unknown, *dt*, but it can be eliminated by using Eq. 11.33 which is now applicable with time *dt* representing the time to bring the liquid column to rest. Thus

$$p_i = \frac{wLV}{gdt}$$

or $$dt = \frac{wLV}{gp_i} \quad ...(ii)$$

Substituting this value in Eq. (i), we get

$$p_i = \frac{wV^2}{gp_i\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}$$

or inertia pressure, $$p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}} \quad ...(11.35)$$

If elasticity of the pipe material is not considered, then the term $\frac{D}{TE}\left(1-\frac{1}{2m}\right)$ may be neglected. The equation then becomes

$$p_i = V\sqrt{\frac{wK}{g}} \quad ...(11.36)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 11.36 therefore represents the maximum pressure rise for instantaneous closure of valve. It may, however, be noted that maximum pressure rise is independent of the dimensions of the pipe, but it depends only on the initial velocity of flow of liquid and its physical characteristics.

The velocity of the compression wave V_0 is represented by

$$V_0 = \frac{L}{dt} = \sqrt{\frac{g}{w\left[\frac{1}{K} + \frac{D}{TE}\left(1 - \frac{1}{2m}\right)\right]}} \quad ...(iii)$$

Again if the term $\frac{D}{TE}\left(1 - \frac{1}{2m}\right)$ is neglected then $V_0 = \sqrt{(gK/w)} = \sqrt{K/\rho}$, which is identical with the velocity of the propagation of sound wave in the liquid.

By introducing Eq. (iii) in Eq. 11.35 the following simple expression for the inertia head may be obtained

$$h_i = \frac{p_i}{w} = \frac{VV_0}{g} \quad ...(11.37)$$

This is often known as the 'Allievi formula'.

Further as an approximation if the Poisson's ratio is not considered then the circumferential strain is given by

$$\frac{dD}{D} = \frac{f_t}{E}$$

or
$$\frac{dD}{D} = \frac{p_i D}{2TE}$$

or
$$dD = \frac{p_i D^2}{2TE}$$

and
$$dq_e = \frac{p_i D^2}{2TE} \times \frac{\pi DL}{2}$$

Thus by substituting this value of dq_e in the above equation, we get

$$\frac{\pi D^2}{4} \times dL = \frac{p_i}{K}\frac{\pi}{4} D^2 L + \frac{p_i D^2}{2TE} \times \frac{\pi DL}{2}$$

or
$$dL = p_i L\left[\frac{1}{K} + \frac{D}{TE}\right]$$

By adopting the same procedure as indicated earlier the expression for the inertia pressure is obtained as

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K} + \frac{D}{TE}\right)}} \quad ...(11.38)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again if the elasticity of the pipe material is not considered, then the term (D/TE) may be neglected. Equation 11.38 then becomes same as Eq. 11.36.

11.22 PIPE NETWORKS

A group of interconnected pipes forming several loops or circuits as shown in Fig. 11.17 is called a network of pipes. Such networks of pipes are commonly used for municipal water distribution systems

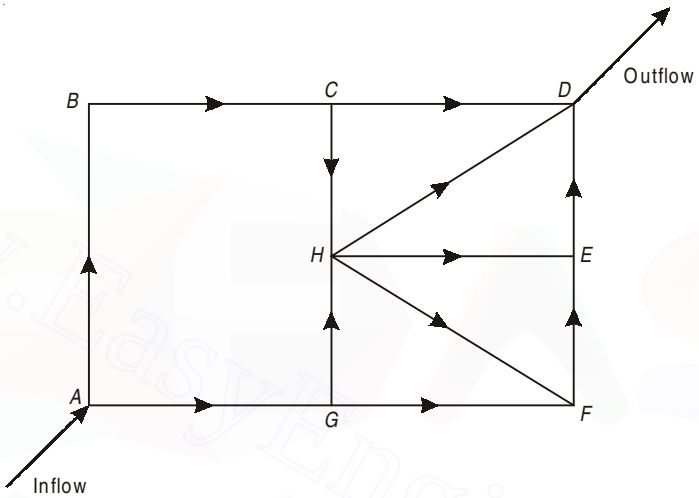


Figure 11.17 *Pipe network*

in cities. The main problem in a pipe network is to determine the distribution of flow through the various pipes of the network such that all the conditions of flow are satisfied and all the circuits are then balanced. The conditions to be satisfied in any network of pipes are as follows:

(1) According to the principle of continuity the flow into each junction must be equal to the flow out of the junction. For example at junction A, the inflow must be equal to the flow through AB and AG.

(2) In each loop, the loss of head due to flow in clockwise direction must be equal to the loss of head due to flow in anticlockwise direction. For example in the loop $ABCHG$ the sum of the head losses due to flow in AB, BC and CH (clockwise flow) must be equal to the sum of the head losses due to flow in AG and GH (anticlockwise flow).

(3) The Darcy–Weisbach equation must be satisfied for flow in each pipe. Minor losses may be neglected if the pipe lengths are large. However if the minor losses are large, they may be taken into account by considering them in terms of the head loss due to friction in equivalent pipe lengths. According to Darcy–Weisbach equation the loss of head h_f through any pipe discharging at the rate of Q can be expressed as

$$h_f = r Q^n \qquad \text{...(11.39)}$$

where r is a proportionality factor which can be determined for each pipe knowing the friction factors f, the length L and the diameter D of the pipe $\left(r = \frac{fL}{2g(\pi/4)^2 D^5} = \frac{fL}{12.10 D^5} \right)$; and n is an exponent having a numerical value ranging from 1.72 to 2.00.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The pipe network problems are in general complicated and cannot be solved analytically. As such methods of successive approximations are utilised. One such method which is commonly used is 'Hardy Cross Method', named after its original investigator. The procedure for the solution of pipe network problems by the Hardy Cross Method is as follows:

(1) Assume a most suitable distribution of flow that satisfies continuity at each junction.

(2) With the assumed values of Q, compute the head losses for each pipe using Eq. 11.39.

(3) Consider different loops or circuits and compute the net head loss around each circuit considering the head loss in clockwise flows as positive and in anti-clockwise flows as negative. For a correct distribution of flow the net head loss around each circuit should be equal to zero, so that the circuit will be balanced. However, in most of the cases, for the assumed distribution of flow the head loss around the circuit will not be equal to zero. The assumed flows are then corrected by introducing a correction ΔQ for the flows, till the circuit is balanced. The value of the correction ΔQ to be applied to the assumed flows of the circuit may be obtained as follows:

For any pipe if Q_0 is the assumed discharge and Q is the correct discharge, then

$$Q = Q_o + \Delta Q$$

and the head loss for the pipe is

$$h_f = rQ^n = r\,(Q_o + \Delta Q)^n$$

Thus for the complete circuit

$$\Sigma h_f = \Sigma rQ^n = \Sigma r\,(Q_o + \Delta Q)^n$$

By expanding the terms in the brackets by binomial theorem

$$\Sigma rQ^n = \Sigma\, r[Q_0^n + nQ_0^{n-1}\Delta Q + ...]$$

If ΔQ is small compared with Q_0 all terms of the series after the second one may be dropped. Thus

$$\Sigma rQ^n = \Sigma\, rQ_0^n + \Sigma\, rnQ_0^{n-1}\Delta Q$$

For the correct distribution the circuit is balanced and hence $\Sigma rQ^n = 0$.

Therefore, $\Sigma\, rQ_0^n + \Delta Q \Sigma\, rnQ_0^{n-1} = 0$

In the above expression ΔQ has been taken out of the summations as it is same for all the pipes in the circuit. Solving for ΔQ

$$\Delta Q = -\frac{\Sigma rQ_0^n}{\Sigma rn\, Q_0^{n-1}} \qquad ...(11.40)$$

In the above expression for the correction the denominator is the sum of absolute terms and hence it has no sign. Further if the head losses due to flow in the clockwise direction are more than losses due to flow in the anti-clockwise direction, then according to the sign convention adopted, ΔQ will be negative and hence it should be added to the flow in the anti-clockwise direction and subtracted from the flow in the clockwise direction. On the other hand if the head losses due to flow in the clockwise direction are less than the head losses due to flow in the anti-clockwise direction, then ΔQ will be positive and hence it should be added to the flow in the clockwise direction and subtracted from the flow in the anti-clockwise direction. Moreover, for the pipes common to two circuits or loops (such as *CH*, *GH*, *HF* etc.) a correction from both the loops will be required to be applied.

(4) With the corrected flows in all the pipes, a second trial calculation is made for all the loops and the process is repeated till the corrections become negligible.

The method described above for the solution of pipe network problems is indicated in Illustrative Example 11.20.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ILLISTRATIVE EXAMPLES

Example 11.1. *A pipeline ABC 180 m long, is laid on an upward slope of 1 in 60. The length of the portion AB is 90 m and its diameter is 0.15 m. At B the pipe section suddenly enlarges to 0.30 m diameter and remains so for the remainder of its length BC, 90 m. A flow of 50 litres per second is pumped into the pipe at its lower end A and is discharged at the upper end C into a closed tank. The pressure at the supply end A is 137.34 kN/m². Sketch (a) the total energy line, (b) the hydraulic grade line and also find the pressure at the discharge end C. Take f = 0.02.*

Solution

$$Q = 50 \text{ l/s} = 50 \times 10^{-3} \text{ m}^3/\text{s}$$

$$D_1 = 0.15 \text{ m}; D_2 = 0.30 \text{ m}$$

$$V_1 = \frac{Q}{\frac{\pi D_1^2}{4}} = \frac{50 \times 10^{-3}}{\frac{\pi}{4} \times (0.15)^2} = 2.83 \text{ m/s}$$

$$V_2 = \frac{Q}{\frac{\pi D_2^2}{4}} = \frac{50 \times 10^{-3}}{\frac{\pi}{4} \times (0.30)^2} = 0.71 \text{ m/s}$$

Loss of head in section *AB*

$$hf_1 = \frac{fL_1}{D_1} \frac{V_1^2}{2g}$$

$$= \frac{0.02 \times 90}{0.15} \times \frac{(2.83)^2}{2 \times 9.81} = 4.90 \text{ m}$$

Loss of head in section *BC*

$$hf_2 = \frac{fL_2}{D_2} \frac{V_2^2}{2g}$$

$$= \frac{0.02 \times 90}{0.30} \times \frac{(0.71)^2}{2 \times 9.81} = 0.153 \text{ m}$$

Loss of head at *B* due to sudden enlargement

$$= \frac{(V_1 - V_2)^2}{2g} = \frac{(2.83 - 0.71)^2}{2 \times 9.81} = 0.229 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

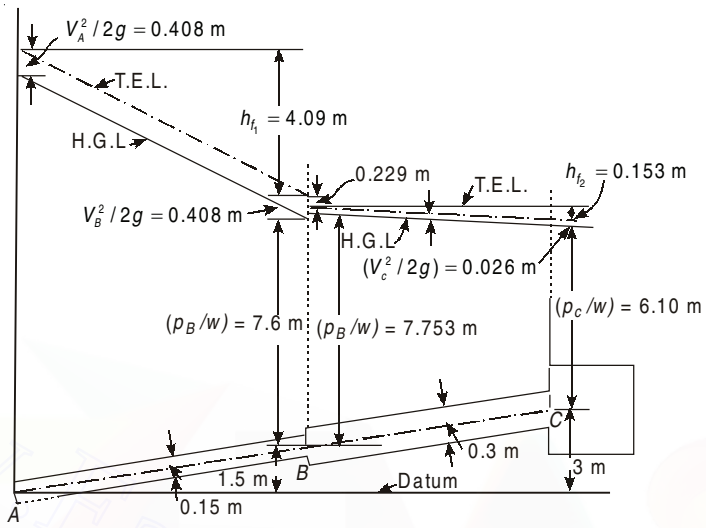


Figure Ex. 11.1

Section AB

Assuming datum to be passing through A, different heads at A are

$$Z_A = 0;\ \frac{p_A}{w} = \frac{137.34 \times 10^3}{9810} = 14 \text{ m}$$

$$\frac{V_A^2}{2g} = \frac{(2.83)^2}{2 \times 9.81} = 0.408 \text{ m}$$

Different heads at B

$$Z_B = 90 \times \frac{1}{60} = 1.5 \text{ m};\ \frac{V_B^2}{2g} = \frac{(2.83)^2}{2 \times 9.81} = 0.408 \text{ m}$$

By applying Bernoulli's equation between points A and B

$$\frac{p_A}{w} + \frac{V_A^2}{2g} + Z_A = \frac{p_B}{w} + \frac{V_B^2}{2g} + Z_B + h_{f1}$$

or $$14 + 0.408 + 0 = \frac{p_B}{w} + 0.408 + 1.5 + 4.90$$

$\therefore$ $$\frac{p_B}{w} = 7.6 \text{ m}$$

Thus hydraulic grade line of section AB will be a line joining $\frac{p_A}{w} = 14$ m and $\frac{p_B}{w} = 7.6$ m as shown

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in Fig. Ex. 11.1. Total energy line is obtained by adding $\left(\frac{V_A^2}{2g}\right) = \left(\frac{V_B^2}{2g}\right) = 0.408$ m to the hydraulic grade line. Thus both lines will be running parallel to each other.

Section BC

Different heads at *B*

$$\frac{p'_B}{w} = \frac{p_B}{w} + \frac{V_B^2}{2g} - \frac{(V'_B)^2}{2g} - \text{loss due to sudden enlargement}$$

$$\frac{(V'_B)^2}{2g} = \frac{(0.71)^2}{2\times 9.81} = 0.026 \text{ m (as } V'_B = V_2)$$

$$\therefore \quad \frac{p'_B}{w} = 7.6 + 0.408 - 0.026 - 0.229 = 7.753 \text{ m}$$

$$Z_B = 1.5 \text{ m}$$

Different heads at *C*

$$Z_C = 180 \times \frac{1}{60} = 3 \text{ m}$$

$$\frac{V_C^2}{2g} = \frac{(0.71)^2}{2\times 9.81} = 0.026 \text{ m}$$

$$\frac{p_C}{w} = \frac{p_B}{w} - (Z_C - Z_B) - h_{f2}$$

$$= 7.753 - (3.0 - 1.5) - 0.153 = 6.10 \text{ m}$$

Thus hydraulic grade line of section *BC* will be a line joining $\frac{p'_B}{w} = 7.753$ m and $\frac{p_C}{w} = 6.10$ m as shown in the figure. Total energy line will be at a distance $\frac{(V'_B)^2}{2g} = \frac{V_C^2}{2g} = 0.026$ m above the hydraulic grade line

The pressure at the discharge end = $\frac{p_C}{w} = 6.10$ m of water

or $$p_C = \frac{6.10\times 9810}{10^3}$$

$$= 59.841 \text{ kN/m}^2 = 59.841 \text{ kP}_a$$

Example 11.2. *A pipeline 0.225 m in diameter and 1580 m long has a slope of 1 in 200 for the first 790 m and 1 in 100 for the next 790 m. The pressure at the upper end of the pipeline is 107.91 kP_a and at the lower end is 53.955 kP_a.. Taking f = 0.032 determine the discharge through the pipe.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Assuming the datum to be passing through the lower end of the pipeline, the datum head for the upper end of the pipeline is

$$Z_1 = \frac{790}{200} + \frac{790}{100} = (3.95 + 7.90) = 11.85 \text{ m}$$

Applying Bernoulli's equation between the upper and the lower ends of the pipeline

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + Z_2 + h_f$$

But $V_1 = V_2 = V$

Then the loss of head due to friction

$$h_f = \frac{fLV^2}{2gD} = \frac{0.032 \times 1580 \times V^2}{2 \times 9.81 \times 0.225} = 11.45\ V^2$$

Thus by substitution, we get

$$\frac{107.91 \times 10^3}{9810} + \frac{V^2}{2g} + 11.85 = \frac{53.955 \times 10^3}{9810} + \frac{V^2}{2g} + 0 + 11.45\ V^2$$

or $11.45V^2 = 17.35$

or $V = \sqrt{\frac{17.35}{11.45}} = 1.23 \text{ m/s}$

$\therefore$ Discharge $Q = \frac{\pi D^2}{4} V$

$$= \frac{\pi(0.225)^2}{4} \times 1.23 = 0.048\ 9 \text{ m}^3/\text{s}$$

Example 11.3. *The population of a city is 800 000 and it is to be supplied with water from a reservoir 6.4 km away. Water is to be supplied at the rate of 140 litres per head per day and half the supply is to be delivered in 8 hours. The full supply level of the reservoir is R.L 180.00 and its lowest water level is R.L. 105.00. The delivery end of the main is at R.L. 22.50 and the head required there is12 m. Find the diameter of the pipe. Take f = 0.04.*

Solution

Total daily supply = $(800\ 000 \times 140 \times 10^{-3}) = 11.2 \times 10^4 \text{ m}^3$. Supply to be delivered in 8 hours

$$= \frac{11.2 \times 10^4}{2} = 5.6 \times 10^4 \text{ m}^3$$

$\therefore$ Discharge $= \frac{5.6 \times 10^4}{8 \times 60 \times 60} = 1.94 \text{ m}^3/\text{s}$

The two extreme cases are considered as follows.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) When the reservoir is full, the head available

$$= (180.00 - 22.50 - 12.00)$$
$$= 145.50 \text{ m}$$

Neglecting the minor losses, we have

$$145.50 = \frac{fLV^2}{2gD} = \frac{fLQ^2}{2g(\pi/4)^2 D^5}$$

or $$D^5 = \frac{0.04 \times 6.4 \times 10^3 \times (1.94)^2}{2 \times 9.81 \times (\pi/4)^2 \times 145.50}$$

$$\therefore \quad D = 0.886 \text{ m}$$

(b) For the lowest water level in the reservoir, the head available

$$= (105.00 - 22.50 - 12.00) = 70.50 \text{ m}$$

Again neglecting the minor losses, we have

$$70.50 = \frac{fLV^2}{2gD} = \frac{fLQ^2}{2g(\pi/4)^2 D^5}$$

or $$D^5 = \frac{0.04 \times 6.4 \times 10^3 \times (1.94)^2}{2 \times 9.81 \times (\pi/4)^2 \times 70.50}$$

$$\therefore \quad D = 1.025 \text{ m}$$

Thus the pipe diameter equal to 1.025 m may be adopted. This is so because a pipe of larger diameter than required when the reservoir has water level at full supply level, will result in less loss of head. Hence the required quantity of water at the required head may be made available by regulating the flow by means of a valve. On the other hand if a pipe of smaller diameter than required when the reservoir has the lowest water level, is provided then it would result in a considerable loss of head and the required discharge at the requisite residual head will not be obtained.

Example 11.4. *A pipe 50 mm diameter is 6 m long and the velocity of flow of water in the pipe is 2.4 m/s. What loss of head and the corresponding power would be saved if the central 2 m length of pipe was replaced by 75 mm diameter pipe, the change of section being sudden? Take $f = 0.04$ for the pipes of both diameters.*

Solution

Velocity of flow in the larger diameter pipe

$$= \frac{2.4 \times (\pi/4) \times (0.05)^2}{(\pi/4) \times (0.075)^2} = 1.07 \text{ m/s}$$

Head loss due to friction in the central 2 m length of the pipe is

$$h_{f1} = \frac{fLV^2}{2gD} = \frac{0.04 \times 2 \times (2.4)^2}{2 \times 9.81 \times 0.05} = 0.47 \text{ m}$$

When the central 2 m length of the 50 mm diameter pipe is replaced by a 75 mm diameter pipe, then the various head losses are as follows:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Loss of head due to sudden enlargement

$$= \frac{(V_1 - V_2)^2}{2g} = \frac{(2.4-1.07)^2}{2\times 9.81} = 0.090 \text{ m}$$

Loss of head due to friction

$$h_f = \frac{fLV^2}{2gD} = \frac{0.04\times 2\times (1.07)^2}{2\times 9.81\times 0.075} = 0.062 \text{ m}$$

Loss of head due to sudden contraction

$$= \frac{0.5V^2}{2g} = \frac{0.5\times (2.4)^2}{2\times 9.81} = 0.147 \text{ m}$$

$\therefore$ Total head loss = (0.090 + 0.062 + 0.147)
= 0.299 m

$\therefore$ Head saved h = (0.47 – 0.299)= 0.171 m

Discharge $Q = \frac{\pi}{4} \times (0.05)^2 \times 2.4 = 4.71 \times 10^{-3}$ m^3/s

$\therefore$ Power saved = (wQh)
= $(9810 \times 4.71 \times 10^{-3} \times 0.171)$
= 7.901 W

Example 11.5. *Two pipes each 300 m long are available for connecting to a reservoir from which a flow of 0.085 m^3/s is required. If the diameters of the two pipes are 0.30 m and 0.15 m respectively, determine the ratio of the head lost when the pipes are connected in series to the head lost when they are connected in parallel. Neglect minor losses.*

Solution

When the pipes are connected in series, the loss of head due to friction is

$$(h_f)_s = h_{f1} + h_{f2}$$

$$= \frac{fLV_1^2}{2gD_1} + \frac{fLV_2^2}{2gD_2}$$

$$= \frac{fLQ^2}{2g(\pi/4)^2 D_1^5} + \frac{fLQ^2}{2g(\pi/4)^2 D_2^5}$$

$$= \frac{f\times 300\times (0.085)^2}{2g(\pi/4)^2}\left[\frac{1}{(0.30)^5} + \frac{1}{(0.15)^5}\right]$$

When the pipes are connected in parallel, then let Q_1 and Q_2 be the discharge in the bigger and the smaller pipes respectively.

Thus

$$0.085 = Q_1 + Q_2 \qquad \text{...(i)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Also $$(h_f)_p = \frac{fLQ_1^2}{2g(\pi/4)^2 D_1^5} = \frac{fLQ_2^2}{2g(\pi/4)^2 D_2^5} \quad \ldots(ii)$$

From Eq. (ii), we have

$$\frac{Q_1^2}{(0.30)^5} = \frac{Q_2^2}{(0.15)^5}$$

or $$Q_1 = 2^{5/2} Q_2 = 5.657\, Q_2 \quad \ldots(iii)$$

Solving Eqs (i) and (iii), we get

$$6.657\, Q_2 = 0.085$$

or $$Q_2 = \frac{0.085}{6.657}$$

$$\therefore \quad (h_f)_p = \frac{f \times 300 \times (0.085)^2}{2g(\pi/4)^2 \times (0.15)^5 \times (6.657)^2}$$

Thus $$\frac{(h_f)_s}{(h_f)_p} = \frac{\dfrac{f \times 300 \times (0.085)^2}{2g(\pi/4)^2}\left[\dfrac{1}{(0.30)^5} + \dfrac{1}{(0.15)^5}\right]}{\dfrac{f \times 300 \times (0.085)^2}{2g(\pi/4)^2 \times (0.15)^5 \times (6.657)^2}}$$

or $$\frac{(h_f)_s}{(h_f)_p} = \frac{[(0.30)^5 + (0.15)^5] \times (6.657)^2}{(0.30)^5}$$

$$= 45.7$$

Example 11.6. *A pipe having a length of 6 km and diameter 0.70 m connects two reservoirs A and B, the difference between their water levels is 30 m. Halfway along the pipe there is a branch through which water can be supplied to a third reservoir C. Taking f = 0.024 determine the rate of flow to reservoir B when (a) no water is discharged to reservoir C; (b) the quantity of water discharged to reservoir C is 0.15 m³/s. Neglect minor losses.*

Solution

(a) When no water is discharged to reservoir C, then we have

$$30 = h_f = \frac{fLV^2}{2gD} = \frac{0.024 \times 6 \times 10^3 \times V^2}{2 \times 9.81 \times 0.70}$$

or $$V = 1.692 \text{ m/s}$$

$$\therefore \quad Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.70)^2}{4} \times 1.692$$

$$= 0.65 \text{ m}^3/\text{s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) When 0.15 m^3/s is discharged to reservoir C, then let Q be the total discharge from reservoir A. The discharge flowing into reservoir $B = (Q - 0.15)$.

Now applying Bernoulli's equation between the reservoirs A and B, we have (neglecting the minor losses)

$$30 = \frac{0.024 \times 3 \times 10^3 \times Q^2}{2 \times 9.81 \times (\pi/4)^2 \times (0.70)^5} + \frac{0.024 \times 3 \times 10^3 \times (Q - 0.15)^2}{2 \times 9.81 \times (\pi/4)^2 \times (0.70)^5}$$

or

$$\text{or} \qquad 30 = \frac{0.024 \times 3 \times 10^3}{2 \times 9.81 \times (\pi/4)^2 \times 0.70)^5} \left[Q^2 + (Q - 0.15)^2\right]$$

$$\text{or} \qquad 30 = 35.4\left[2Q^2 - 0.3Q + 0.0225\right]$$

$$\text{or} \qquad Q^2 - 0.15\,Q - 0.401 = 0$$

$$\therefore \qquad Q = 0.713 \text{ m}^3/\text{s}$$

Hence flow to reservoir $B = (0.713 - 0.15)$

$$= 0.563 \text{ m}^3/\text{s}$$

Example 11.7. *A compound piping system consists of 1800 m of 0.50 m, 1200 m of 0.40 m and 600 m of 0.30 m new cast iron pipes connected in series. Convert the system to (a) an equivalent length of 0.40 m pipe, and (b) and equivalent size pipe 3600 m long.*

Solution

From Eq. 11.13, we have

$$\frac{L}{D^5} = \left[\frac{L_1}{D_1^5} + \frac{L_2}{D_2^5} + \frac{L_3}{D_3^5}\right]$$

$L_1 = 1800$ m; $D_1 = 0.50$ m

$L_2 = 1200$ m; $D_2 = 0.40$ m

$L_3 = 600$ m; $D_3 = 0.30$ m

(a) To find L for $D = 0.40$ m

Thus by substitution, we get

$$L = (0.40)^5 \left[\frac{1800}{(0.50)^5} + \frac{1200}{(0.40)^5} + \frac{600}{(0.30)^5}\right]$$

$$= 4\,318 \text{ m}$$

(b) To find D for $L = 3600$ m

Thus by substitution, we get

$$\frac{3600}{D^5} = \left[\frac{1800}{(0.50)^5} + \frac{1200}{(0.40)^5} + \frac{600}{(0.30)^5}\right]$$

$$\therefore \qquad D = 0.386 \text{ m} \simeq 0.39 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 11.8. *A 0.3 m diameter pipe 2340 m long is connected with a reservoir whose surface is 72 m above the discharging end of the pipe. If for the last 1170 m, a second pipe of the same diameter be laid beside the first and connected to it, what would be the increase in the discharge? Take $f = 0.02$.*

Solution

In the first case, from Eq. 11.8, we have

$$H = \left(1.5 + \frac{fL}{D}\right)\frac{V^2}{2g}$$

or $$72 = \left(1.5 + \frac{0.02 \times 2340}{0.30}\right)\frac{V^2}{2 \times 9.81}$$

or $$V = \sqrt{\frac{72 \times 2 \times 9.81}{157.5}} = 3 \text{ m/s}$$

$\therefore$ Discharge $$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.30)^2}{4} \times 3 = 0.212 \text{ m}^3/\text{s}$$

In the second case let Q be the total discharge. Since for the second half of the length there are two parallel pipes of the same diameter, each pipe will carry discharge equal to $(Q/2)$ and the velocity of flow in each pipe will be equal to half the velocity of flow V in the first half of the length.

Thus applying Bernoulli's equation between the water surface in the reservoir and the outlet of pipe, we get

$$72 = 0.5\frac{V^2}{2g} + \frac{0.02 \times 1170 \times V^2}{2g \times 0.30} + \frac{0.02 \times 1170 \times (V/2)^2}{2g \times 0.30} + \frac{(V/2)^2}{2g}$$

or $$72 = \frac{V^2}{2g}[0.5 + 78.0 + 0.25]$$

or $$V = 3.79 \text{ m/s}$$

$\therefore$ $$Q = \frac{\pi}{4}(0.30)^2 \times 3.79 = 0.268 \text{ m}^3/\text{s}$$

$\therefore$ Increase in discharge

$$= (0.268 - 0.212) = 0.056 \text{ m}^3/\text{s}$$

or percentage increase in discharge

$$= \frac{0.056}{0.212} \times 100 = 26.42 \%$$

Example 11.9. *For the distribution main of a city water supply a 0.30 m main is required. As pipes above 0.25 m diameter are not available, it is decided to lay two parallel mains of same diameter. Find the diameter of the parallel main.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Let D be the diameter of the single large pipe which is to be replaced by two parallel pipes of equal diameter d. Let all the three pipes be of equal length say L.

Then

$$h_f = \frac{fL}{D}\frac{V^2}{2g}$$

$\therefore$ Velocity of flow in large pipe

$$V = \sqrt{\frac{h_f \times 2g}{f}}\sqrt{\frac{D}{L}}$$

Discharge from large pipe

$$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi D^2}{4} \times \sqrt{\frac{h_f \times 2g}{f}}\sqrt{\frac{d}{L}}$$

$$= \frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{d^{5/2}}{L^{1/2}}\right)$$

Frictional head loss in each of the parallel pipes = frictional head loss in large pipe = h_f

$\therefore$ Velocity of water in each of the parallel pipes

$$= \sqrt{\frac{h_f \times 2g}{f}}\sqrt{\frac{d}{L}}$$

Discharge from each of the parallel pipes

$$= \frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{d^{5/2}}{L^{1/2}}\right)$$

From Eq. 11.14, we have

$$Q = Q_1 + Q_2$$

but $Q_1 = Q_2$ and hence

$$\frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{D^{5/2}}{L^{1/2}}\right) = 2 \times \frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{d^{5/2}}{L^{1/2}}\right)$$

or $$D^{5/2} = 2d^{5/2}$$

or $$d = \frac{D}{2^{2/5}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{0.30}{2^{2/5}} = 0.23 \text{ m}$$

Since pipe of 0.23 m diameter is not a standard one, it may not be available. As such each pipe of diameter 0.25 m may be used.

Note: If in place of a single pipe of diameter D, n pipes each of diameter d are laid in parallel, then

$$d = \frac{D}{n^{2/5}}$$

Example 11.10. *Two reservoirs are connected by a pipeline which rises above the level of the highest reservoir. What will be the highest point of the siphon above this level if the length of the pipe leading upto this point is 540 m, the diameter of the pipe being 0.35 m? The difference in level of the two reservoirs is 12.6 m and the total length of the pipe is 960 m. The siphon must run full, the separation of dissolved gases may be assumed to occur at an absolute pressure of 2.4 m of water. What will be the discharge under the above conditions in litres per minute? Assume $f = 0.04$ and allow for all losses.*

Solution

Applying Bernoulli's equation between the reservoirs A and B, we get

$$12.6 = 0.5 \frac{V^2}{2g} + \frac{fLV^2}{2gD} + \frac{V^2}{2g}$$

or
$$12.6 = \frac{V^2}{2g}\left[1.5 + \frac{fL}{D}\right]$$

$$= \frac{V^2}{2g}\left[1.5 + \frac{0.04 \times 960}{0.35}\right]$$

or
$$V = 1.49 \text{ m/s}$$

Now applying Bernoulli's equation between the reservoir A and the summit S, we have, taking water surface in reservoir A as datum

$$\frac{p_a}{w} = h + 2.4 + 0.5 \frac{V^2}{2g} + \frac{V^2}{2g} + \frac{fL_1V^2}{2gD}$$

where h is the height of the summit above the water surface in the reservoir A.

Also atmospheric pressure head

$$\frac{p_a}{w} = 10.3 \text{ m of water}$$

$$L_1 = 540 \text{ m}; V = 1.49 \text{ m/s}$$

Thus by substitution, we get

$$10.3 = h + 2.4 + \frac{(1.49)^2}{2 \times 9.81}\left[1.5 + \frac{0.04 \times 540}{0.35}\right]$$

$$\therefore \quad h = 0.75 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

i.e., the height of the summit of the siphon above the water surface in the reservoir $A = 0.75$ m.

Discharge $$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.35)^2}{4} \times 1.49$$

$$= 0.143\ 4\ \text{m}^3/\text{s}$$

$$= 0.143\ 4 \times 10^3 \times 60\ l/\text{min}$$

$$= 8.6 \times 10^3\ l/\text{min}.$$

Example 11.11. *The difference in the water surface levels of two reservoirs which are connected by a siphon is 8 m. The length of the siphon is 600 m and its diameter 0.3 m. Assuming $f = 0.02$, determine the discharge when the siphon is running full.*

If the summit of the pipeline is 5 m above the surface level of the upper reservoir, determine the maximum length of the inlet leg for the pipe to run full. Allow for all the losses and assume the permissible minimum pressure at the summit of the siphon to be absolute zero.

Solution

Applying Bernoulli's equation between the two reservoirs, we get

$$8 = 0.5\frac{V^2}{2g} + \frac{fLV^2}{2gD} + \frac{V^2}{2g}$$

or $$8 = \frac{V^2}{2g}\left[1.5 + \frac{fL}{D}\right]$$

$$= \frac{V^2}{2g}\left[1.5 + \frac{0.02 \times 600}{0.30}\right]$$

or $$V = 1.945\ \text{m/s}$$

$\therefore$ Discharge $$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.30)^2}{4} \times 1.945$$

$$= 0.1375\ \text{m}^3/\text{s}$$

Applying Bernoulli's equation between the water surface in the upper reservoir and the summit of the siphon, we have

$$10.3 = 0 + 5 + 0.5\ \frac{V^2}{2g} + \frac{fL_1V^2}{2gD} + \frac{V^2}{2g}$$

or $$10.3 = 0 + 5 + \frac{V^2}{2g}\left[1.5 + \frac{fL_1}{D}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$10.3 = 0 + 5 + \frac{(1.945)^2}{2 \times 9.81}\left[1.5 + \frac{0.02 \times L_1}{0.30}\right]$$

$$\therefore \quad L_1 = 390 \text{ m}$$

Example 11.12. *Water flows from a reservoir through a pipe of 0.15 m diameter and 180 m long to a point 13.5 m below the open surface of the reservoir. Here it branches into two pipes, each of 0.1 m diameter, one of which is 48 m long discharging to atmosphere at a point 18 m below reservoir level, and the other 60 m long discharging to atmosphere 24 m below reservoir level. Assuming a constant friction coefficient of 0.032, calculate the discharge from each pipe. Neglect any losses at the junction.*

Solution

Let the pressure at the junction (2) be p and let atmospheric pressure be 10.3 m of water.

Applying Bernoulli's equation between (1) and (2), we get

$$10.3 + 13.5 = \frac{p}{w} + \frac{V_1^2}{2g} + \frac{0.032 \times 180 \times V_1^2}{2g \times 0.15}$$

$$\therefore \quad \frac{p}{w} = 23.8 - 2.008\, V_1^2 \qquad \text{...(i)}$$

Applying Bernoulli's equation between (2) and (3), we get

$$\frac{p}{w} + \frac{V_2^2}{2g} + 4.5 = 10.3 + \frac{V_2^2}{2g} + \frac{0.032 \times 48 \times V_2^2}{2g \times 0.10}$$

$$\therefore \quad \frac{p}{w} = 5.8 + 0.783\, V_2^2 \qquad \text{...(ii)}$$

Applying Bernoulli's equation between (2) and (4), we get

$$\frac{p}{w} + \frac{V_3^2}{2g} + 10.5 = 10.3 + \frac{V_3^2}{2g} + \frac{0.032 \times 60 \times V_3^2}{2g \times 0.10}$$

$$\therefore \quad \frac{p}{w} = 0.979 V_3^2 - 0.2 \qquad \text{...(iii)}$$

By continuity, we have

$$9V_1 = (V_2 + V_3) \times 4 \qquad \text{..(iv)}$$

These four equations are to be solved, which may be accomplished as follows:

Let $V_3 = mV_2$

Thus from Eq. (iv), we have

$$V_1 = \frac{4}{9}(m+1)\, V_2$$

Equating (i) and (ii), we get

$$23.8 - 2.008 V_1^2 = 5.8 + 0.783 V_2^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$18 = \left[2.008 \times \frac{16}{81}(m+1)^2 + 0.783\right] V_2^2 \quad \text{...(v)}$$

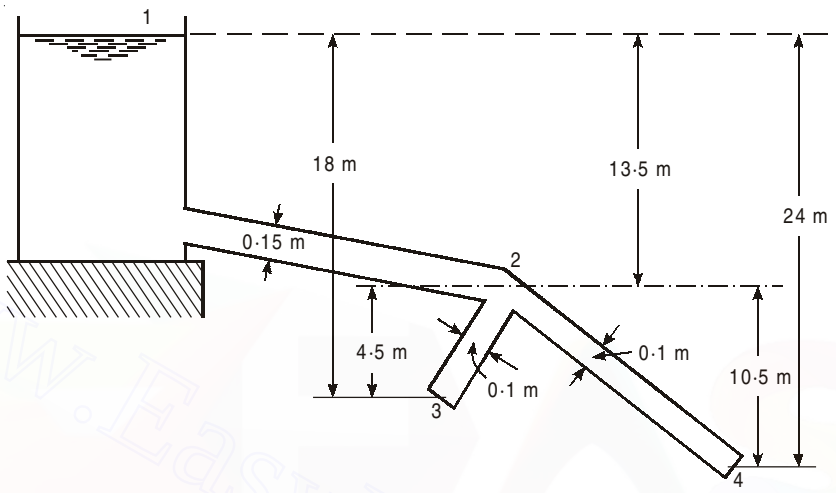


Figure Ex. 11.12

Again equating (ii) and (iii), we get

$$5.8 + 0.783 V_2^2 = 0.979 V_2^2 - 0.2$$

or $$6 = [0.979 \text{ m} - 0.783] V_2^2 \quad \text{...(vi)}$$

Dividing (v) by (vi), we get

$$3 = \frac{0.397 \text{m}^2 + 0.794 \text{ m} + 1.180}{0.979 \text{ m}^2 - 0.783}$$

or $2.540 \text{ m}^2 - 0.794 \text{ m} - 3.529 = 0$

or $\text{m}^2 - 0.313 \text{ m} - 1.39 = 0$

or $m = 1.346$

Substituting this value of m in Eq. (vi), we get

$$V_2 = 2.461 \text{ m/s}$$

$$V_3 = (1.346 \times 2.461) = 3.313 \text{ m/s}$$

$$\therefore \quad Q_2 = \left[\frac{\pi}{4} \times (0.10)^2 \times 2.461\right] = 0.019\,3 \text{ m}^3/\text{s}$$

$$Q_3 = \left[\frac{\pi}{4} \times (0.10)^2 \times 3.313\right] = 0.026\,0 \text{ m}^3/\text{s}$$

$\therefore$ Total discharge

$$Q_1 = (Q_2 + Q_3)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= (0.0193 + 0.0260)$

$= 0.045\ 3\ m^3/s$

Example 11.13. *Water is pumped through a 600 mm diameter pipe with a head loss of 20 m. It is intended to reduce the power consumption by laying another parallel pipe of appropriate diameter laid by the side of the existing pipeline with the same overall length and the same friction factor—by reducing the head loss to 10m, but still delivering the same discharge jointly through both pipes.What should be the diameter of this additional pipe?*

Solution

Initial head loss $h_f = 20$ m.

Thus if Q is the discharge through the pipe of diameter 600 mm, then from Darcy-Weisbach equation, we have

$$h_f = \frac{fLV^2}{2gd}$$

or $$h_f = \frac{fLQ^2}{2g \times (\pi/4)^2 d^5}$$

or $$h_f = \frac{fLQ^2}{12.1 \times d^5}$$

$$d = 600 \text{ mm} = 0.6 \text{ m}$$

$$\therefore \quad 20 = \frac{fLQ^2}{12.1 \times (0.6)^5} \qquad ...(i)$$

Let d_1 be the diameter of the pipe required to be connected in parallel to the existing pipe, and Q_1 and Q_2 be the discharge through the pipes of diameters 600 m and d_1 respectively.

Since total discharge remains same,

$$Q = Q_1 + Q_2 \qquad ...(ii)$$

The head loss is reduced to 10 m and both the pipes have same length L and same friction f. Thus from Darcy–Weisbach Equation, we have

$$10 = \frac{fLQ_1^2}{12.1 \times (0.6)^5} \qquad ...(iii)$$

$$10 = \frac{fLQ_2^2}{12.1 \times (d_1)^5} \qquad ...(iv)$$

From Eqs (i) and (iii), we obtain

$$Q_1 = \frac{Q}{\sqrt{2}} = 0.0707\ Q$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the value of Q_1 in Eq. (ii), we get

$$\begin{aligned} Q_2 &= Q - Q_1 \\ &= Q - 0.707Q \\ &= 0.293\ Q \end{aligned}$$

From Eqs. (iii) and (iv), we obtain

$$\frac{Q_1^2}{(0.6)^5} = \frac{Q_2^2}{(d_1)^5}$$

or $$\frac{Q_1}{Q_2} = \left(\frac{0.6}{d_1}\right)^{5/2}$$

or $$\left(\frac{0.707Q}{0.293Q}\right) = \left(\frac{0.6}{d_1}\right)^{5/2}$$

$\therefore$ $$d_1 = 0.6\left(\frac{0.293}{0.707}\right)^{2/5}$$

$$= 0.422 \text{ m}$$

$$= 422 \text{ mm}$$

Thus a pipe of nearest available size of diameter 425 mm may be laid by the side of the existing pipe in parallel.

Example 11.14. *A fire engine supplies water to a hose pipe, 75 m long and 75 mm in diameter at a pressure of 294.3 kN/m^2 [3 kg (f)/cm^2] (gage). The discharge end of the hose pipe has a nozzle of diameter d fixed to it. Taking friction factor f as 0.032, determine the diameter d of the nozzle so that the momentum of the issuing jet may be maximum.*

Solution

Let D be the diameter of the hose pipe; L be the length of the hose pipe; V_1 be the velocity of flow in the hose pipe; V_2 be the velocity of the issuing jet; and Q be the discharge.

Thus momentum of the issuing jet is

$$M = \frac{w}{g} Q V_2$$

or $$M = \frac{w}{g}\frac{\pi d^2}{4} V_2^2 \qquad ...(i)$$

From continuity equation, we have

$$Q = \frac{\pi}{4} D^2 V_1 = \frac{\pi}{4} d^2 V_2 \qquad ...(ii)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If H is the head causing the flow, then

$$H = \frac{V_2^2}{2g} + \frac{fLV_1^2}{2gD} \quad \text{...(iii)}$$

From Eqs (ii) and (iii), we have

$$2gH = V_2^2 \left[1 + \frac{fLd^4}{D^5}\right] \quad \text{...(iv)}$$

By substituting the value of V_2^2 from Eq. (iv) in Eq. (i), we get

$$M = \frac{w}{g}\frac{\pi}{4} d^2 \frac{2gH}{1 + \frac{fLd^4}{D^5}}$$

or
$$M = w \frac{\pi}{4} \times 2H \left[\frac{D^5 d^2}{D^5 + fLd^4}\right] \quad \text{...(v)}$$

For maximum value of M,

$$\frac{dM}{dd} = 0$$

From Eq. (v), we have

$$\frac{dM}{dd} = w \frac{\pi}{4} \times 2H \times D^5 \left[\frac{2d(D^5 + fLd^4) - d^2(4fLd^3)}{(D^5 + fLd^4)^2}\right]$$

Thus

$$2d\,(D^5 + fLd^4) = d^2\,(4fLd^3)$$

or
$$fLd^4 = D^5$$

or
$$d = \left(\frac{D^5}{fL}\right)^{1/4}$$

In this case

$$D = 75 \text{ mm} = 0.075 \text{ m}$$
$$L = 75 \text{ m}; f = 0.032$$

Thus by substitution, we get

$$d = \left[\frac{(0.075)^5}{0.032 \times 75}\right]^{1/4}$$

or
$$d = 0.031\,5 \text{ m} = 31.5 \text{ mm}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 11.15. *The turbine as shown in Fig. Ex. 11.15 extrats 45 mhp (metric horse power) from the water flowing through it.The 30 cm diameter 100 m long pipe has a friction factor of 0.02. Neglecting minor losses determint the flow rate through the pipe and the turbine.*

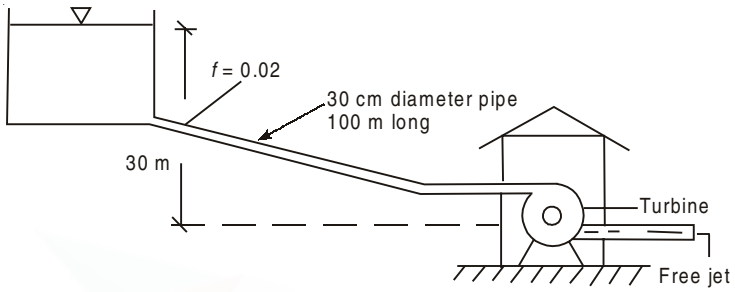


Figure Ex. 11.15

Solution

Head loss due to friction is given by

$$h_f = \frac{fLV^2}{2gd}$$

$f = 0.02$, $L = 100$ m and $d = 30$ cm $= 0.3$ m

Let Q be the discharge through the turbine. Thus

$$V = \frac{Q}{A} = \frac{Q}{(\pi/4)(0.3)^2}$$

Thus by substitution, we get,

$$h_f = \frac{0.02 \times 100 \times Q^2}{2 \times 9.81 \times (\pi/4)^2 \times (0.3)^5}$$

$$= 68.006\ Q^2$$

$\therefore$ Net head on the turbine

$$= (H - h_f)$$

$$= (30 - 68.006\ Q^2)$$

Power extracted by the turbine

$$= 45 \text{ mhp}$$

Thus

$$45 = \frac{wQ\left(H - h_f\right)}{75}$$

or

$$45 = \frac{1000 \times Q \times \left(30 - 68.006\ Q^2\right)}{75}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $68.006\ Q^3 - 30Q + 3.375 = 0$

Solving by trial and error, we get

$$Q = 0.11604 \text{ cumec}$$

Example 11.16. *Power is to be transmitted hydraulically to an accumulator at a distance of 8 km by means of a number of 100 mm pipes laid horizontally, for which the coefficient of friction may be taken as 0.03. The pressure at the accumulator is maintained constant at 6524 kN/m^2. Determine the minimum number of pipes required to ensure an efficiency of at least 92 per cent when the power delivered is 162 kW. Also determine the maximum power that can be transmitted in this case.*

Solution

Let the pressure head at the reservoir be H.

The pressure head at the accumulator

$$= 6524 \text{ kN/m}^2 = 665.04 \text{ m of water}$$

Efficiency of transmission

$$= \frac{H - h_f}{H} \times 100$$

where h_f is head loss due to friction

$$\therefore \quad 0.92 = \frac{H - h_f}{H} = \frac{665.04}{H}$$

or $$H = \frac{665.04}{0.92} = 722.87 \text{ m}$$

$\therefore$ Head lost due to friction

$$h_f = (722.87 - 665.04) = 57.83 \text{ m}$$

But $$h_f = \frac{fLV^2}{2gD}$$

$$\therefore \quad 57.83 = \frac{0.03 \times 8 \times 10^3 \times V^2}{2 \times 9.81 \times 0.10}$$

or $$V = 0.688 \text{ m/s}$$

Power transmitted in kW

$$= \frac{wQ(H - h_f)}{1000}$$

Let n be the number of pipes required. Then

$$Q = \left[\frac{\pi}{4} \times (0.10)^2 \times n \times 0.688\right]$$

$$= (5.404 \times 10^{-3})\ n \text{ m}^3/\text{s}$$

By substitution, we have

$$162 = \frac{9810 \times 5.404 \times 10^{-3} \times n \times 665.04}{1000}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore \quad n = 4.6$

$\therefore$ Number of pipes required $= 5$

For maximum power transmission

$$h_f = \frac{H}{3} = \frac{722.87}{3} = 240.96 \text{ m}$$

Thus $$240.96 = \frac{0.030 \times 8 \times 10^3 \times V^2}{2 \times 9.81 \times 0.10}$$

or $$V = 1.404 \text{ m/s}$$

Total discharge through 5 pipes

$$= \left[\frac{\pi}{4} \times (0.10)^2 \times 5 \times 1.404\right]$$

$$= 5.513 \times 10^{-2} \text{ m}^3/\text{s}$$

$\therefore$ Maximum power in kW that can be transmitted in this case

$$= \frac{9810 \times 5.513 \times 10^{-2} \times (722.87 - 240.96)}{1000}$$

$$= 260.63 \text{ kW}$$

Note. The efficiency corresponding to the maximum power transmission is only 66.7%

Example 11.17. *A 0.5 m diameter and 100 m long pipeline carrying 0.5 m^3/s discharge is fitted with a valve at downstream end. Calculate the rise of pressure caused within the pipe due to the valve closure in (i) 1 second, (ii) instantaneously. Take the sonic velocity as 1430 m/s.*

Solution

(i) Time of value closure $t = 1$s.

The time taken by the sound wave to travel to and fro the valve is

$$t' = \frac{2L}{c} = \frac{2 \times 100}{1430} = 0.1399 \text{ s}$$

Since in this case $t' < t$, the valve closure may be considered as gradual.

Thus from Eq. 11.34, the rise of pressure

$$p_i = \frac{wLV}{gt}$$

$w = 1000 \text{ kg/m}^3$ or 9.81 kN/m^3; $L = 100$ m;

$$V = \frac{0.5}{(\pi/4) \times (0.5)^2} = 2.546 \text{ m/s}$$; $g = 9.81 \text{ m/s}^2$; and $t = 1.0_s$

Thus by substitution, we get

$$p_i = \frac{1000 \times 100 \times 2.546}{9.81 \times 1.0} = 2.595 \times 10^4 \text{ kg/m}^2$$

or $$p_i = \frac{9.81 \times 100 \times 2.546}{9.81 \times 1.0} = 2.546 \times 10^2 \text{ kN/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(ii) For instantaneous closure of value, the rise of pressure is given by Eq. 11.37, Allievi formula, as

$$p_i = \frac{wVV_0}{g}$$

$$V_0 = 1430 \text{ m/s}$$

Thus by substitution, we get

$$p_i = \frac{1000 \times 2.546 \times 1430}{9.81} = 3.711 \times 10^5 \text{ kg/m}^2$$

or

$$p_i = \frac{9.81 \times 2.546 \times 1430}{9.81} = 3.641 \times 10^3 \text{ kN/m}^2$$

Example 11.18. *A steel penstock 0.60 m in diameter has a shell thickness of 12 mm. The modulus of elasticity of the pipe shell material is 2.1 × 10⁵ N/mm² [2.1 × 10⁶ kg(f)/cm²] with Poisson's ratio of* $\frac{1}{4}$ *and the volume modulus of elasticity of water is 2.1 × 10³ N/mm² [2.1 × 10⁴ kg(f)/cm²]. The pipe is designed to discharge water at a mean velocity of 2.1 m/s. Determine the water hammer pressure rise caused by sudden closure of the valve at the downstream end :*

(1) neglecting the elasticity of the pipe material; (2) considering the elasticity of the pipe meterial but neglecting the Poisson's ratio; and (3) considering the elasticity of the pipe material including the Poisson's ratio.

Solution

1. In this case water hammer pressure is given by Eq. 11.36 as

$$p_i = V\sqrt{\frac{wK}{g}}$$

SI units

$$p_i = 2.1\sqrt{\frac{9810 \times 2.1 \times 10^3 \times 10^6}{9.81}}$$

$$= 3.043 \times 10^6 \text{ N/m}^2$$

Meteric units

$$p_i = \sqrt{\frac{1000 \times 2.1 \times 10^4 \times 10^4}{9.81}}$$

$$= 3.073\ 10^5 \text{ kg (f)/m}^2$$

2. In this case water hammer pressure is given by Eq. 11.38 as

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K} + \frac{D}{TE}\right)}}$$

SI units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{9810}\left(\frac{1}{2.1 \times 10^3 \times 10^6} + \frac{0.60}{12 \times 10^{-3} \times 2.1 \times 10^5 \times 10^6}\right)}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 2.485 \times 10^6 \text{ N/m}^2$$

Metric units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{1000}\left(\frac{1}{2.1\times10^4\times10^4}+\frac{0.60}{12\times10^{-3}\times2.1\times10^6\times10^4}\right)}}$$

$$= 2.509 \times 10^5 \text{ kg(f)/m}^2$$

3. In this case water hammer pressure is given by Eq. 11.35 as

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2\text{m}}\right)\right]}}$$

SI units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{9810}\left[\frac{1}{2.1\times10^3\times10^6}+\frac{0.60}{12\times10^{-3}\times2.1\times10^5\times10^6}\left(1-\frac{1}{2\times4}\right)\right]}}$$

$$= 2.538\ \ 106 \text{ N/m}^2$$

Metric units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{1000}\left[\frac{1}{2.1\times10^4\times10^4}+\frac{0.60}{12\times10^{-3}\times2.1\times10^6\times10^4}\left(1-\frac{1}{2\times4}\right)\right]}}$$

$$= 2.563 \times 10^5 \text{ kg(f)/m}^2$$

Example 11.19. *What is the maximum permissible velocity in a cast iron pipeline 100 mm diameter and 15 mm thick, which can be suddenly stopped by a valve at the outlet end of the pipe without letting the rise of pressure in the pipe to exceed 1.545 × 10³ kN/m² [15.75 kg (f)/cm²]? Take E for cast iron as 123.606 × 10⁹ N/m² [12.6× 10⁵ kg(f)/cm²]and K for water as 206.01 × 10⁷ N/m² [21 × 10³kg(f)/cm²].*

Solution

From Eq. 11.38, inertia pressure

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K}+\frac{D}{TE}\right)}}$$

SI units

$$p_i = 1.545 \times 10^3 \text{ kN/m}^2 = 1.545 \times 10^6 \text{ N/m}^2$$
$$K = 206.01 \times 10^7 \text{ N/m}^2$$
$$E = 123.606 \times 10^9 \text{ N/m}^2$$
$$D = 100 \text{ mm} = 0.10 \text{ m}$$
$$T = 15 \text{ mm} = 0.015 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by substitution, we get

$$1.545 \times 10^6 = \frac{V}{\sqrt{\frac{9.81}{9810}\left(\frac{1}{206.01\times10^7}+\frac{0.10}{0.015\times123.606\times10^9}\right)}}$$

$$\therefore \quad V = 1.135 \text{ m/s}$$

Metric units

$$p_i = 15.75 \text{ kg(f)/cm}^2 = 15.75 \times 10^4 \text{ kg(f)/m}^2$$
$$K = 21 \times 10^3 \text{ kg(f)/cm}^2 = 21 \times 10^7 \text{ kg(f)/m}^2$$
$$E = 12.6 \times 10^5 \text{ kg(f)/cm}^2 = 12.6 \times 10^9 \text{ kg(f)/m}^2$$
$$D = 100 \text{ mm} = 0.10 \text{ m}$$
$$T = 15 \text{ mm} = 0.015 \text{ m}$$

Thus by substitution, we get

$$15.75 \times 10^4 = \frac{V}{\sqrt{\frac{9.81}{1000}\left(\frac{1}{21\times10^7}+\frac{0.10}{0.015\times12.6\times10^9}\right)}}$$

$$\therefore \quad V = 1.135 \text{ m/s}$$

Example 11.20. *For a pipe network shown in Fig. Ex. 11.20, determine the flow in each pipe. The value of n may be assumed as 2.0.*

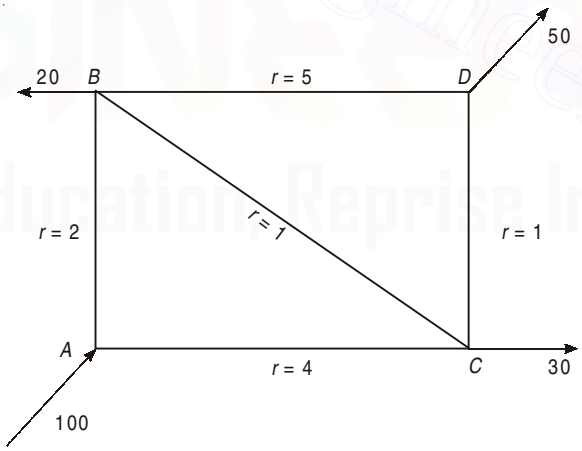


Fig. Ex. 11.20

Solution

For the first trial the distribution is assumed as shown in Fig. 11.20 (a). For this distribution the corrections ΔQ for the loops *ABC* and *BDC* are computed as follows:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Loop ABC

Pipe	rQ^2			$2rQ$		
AB	2×70^2	=	9800	$2 \times 2 \times 70$	=	280
BC	1×35^2	=	1225	$2 \times 1 \times 35$	=	70
AC	-4×30^2	=	−3600	$2 \times 4 \times 30$	=	240
	ΣrQ^2	=	7425	$\Sigma 2rQ$	=	590

$$\therefore \quad \Delta Q = -\frac{7425}{590} \simeq -13$$

Loop BDC

Pipe	rQ^2			$2rQ$		
BD	5×15^2	=	1125	$2 \times 5 \times 15$	=	150
DC	-1×35^2	=	−1225	$2 \times 1 \times 35$	=	70
CB	-1×35^2	=	−1225	$2 \times 1 \times 35$	=	70
	ΣrQ^2	=	−1325	$\Sigma 2rQ$	=	290

$$\therefore \quad \Delta Q = -\frac{(-1325)}{290} \simeq 5$$

Thus applying the above obtained corrections the modified discharges for various pipes are shown in Fig. Ex. 11.20 (b), which is the distribution for the second trial. For this distribution the corrections ΔQ for the loops *ABC* and *BDC* are computed as follows:

Loop ABC

Pipe	rQ^2			$2rQ$		
AB	2×57^2	=	6498	$2 \times 2 \times 57$	=	228
BC	1×17^2	=	289	$2 \times 1 \times 17$	=	34
AC	-4×43^2	=	−7396	$2 \times 4 \times 43$	=	344
	ΣrQ^2	=	−609	$\Sigma 2rQ$	=	606

$$\therefore \quad \Delta Q = -\frac{(-609)}{606} \simeq 1$$

Loop BDC

Pipe	rQ^2			$2rQ$		
BD	5×20^2	=	2000	$2 \times 5 \times 20$	=	200
DC	-1×30^2	=	−900	$2 \times 1 \times 30$	=	60
CB	-1×17^2	=	−289	$2 \times 1 \times 17$	=	34
	ΣrQ^2	=	811	$\Sigma 2rQ$	=	294

$$\therefore \quad \Delta Q = -\frac{811}{294} \simeq -3$$

Thus applying the above obtained corrections the modified discharges for the various pipes are shown in Fig. Ex. 11.20 (c), which is the distribution for the third trial. For this distribution the corrections ΔQ for loops *ABC* and *BDC* are computed as follows:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Loop ABC

Pipe	rQ^2		$2rQ$	
AB	2×58^2	= 6728	$2 \times 2 \times 58$	= 232
BC	1×21^2	= 441	$2 \times 1 \times 21$	= 42
AC	-4×42^2	= − 7056	$2 \times 4 \times 42$	= 336
	ΣrQ^2	= 113	$\Sigma 2rQ$	= 610

$$\therefore \quad \Delta Q = -\frac{113}{610} \simeq 0 \text{ (negligible)}$$

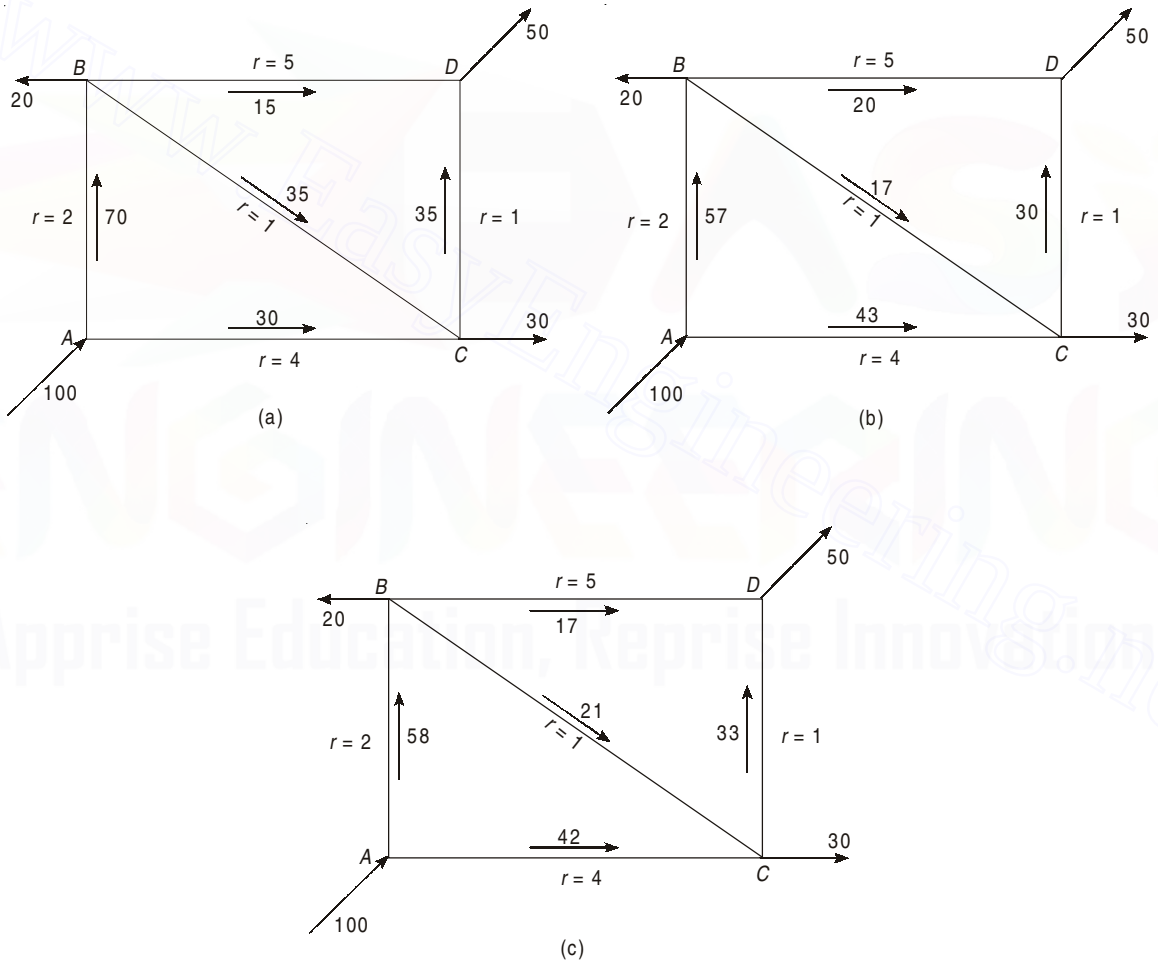


Fig. Ex. 11.20

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Loop BDC

Pipe	rQ^2	$2rQ$
BD	$5 \times 17^2 = 1445$	$2 \times 5 \times 17 = 170$
DC	$-1 \times 33^2 = -1089$	$2 \times 1 \times 33 = 66$
CB	$-1 \times 21^2 = -441$	$2 \times 1 \times 21 = 42$
	$\Sigma rQ^2 = -85$	$\Sigma 2rQ = 278$

$$\therefore \quad \Delta Q = \frac{(-85)}{278} \simeq 0 \text{ (negligible)}$$

Thus Fig. Ex. 11.20 (c) shows the correct distribution of flow for the given network of pipes.

Example 11.21 . *Two reservoirs of large capacity are connected to each other by a straight horizontal pipe 80 mm in diameter and 800 mm long. The difference between the water levels in the two reservoirs is 10 m. The loss of head at pipe entrance is* $0.1 \frac{v^2}{2g}$, *where v is the average velocity in the pipe. The friction factor for the pipe is 0.03. Determine the rate at which water flows through the pipe.*

To increase the rate of flow through the pipe a conical diffuser with an angle of $16°$ *is attached to the pipe as shown in Fig. Ex. 11.18. The loss of head in the diffuser is* $\left[0.3 \frac{(v-V)^2}{2g}\right]$, *where V is the average velocity at the exist section of the diffuser. Determine the exit diameter D and the corresponding axial length L of the diffuser for which the flow rate will be maximum. Compute the discharge under these conditions.*

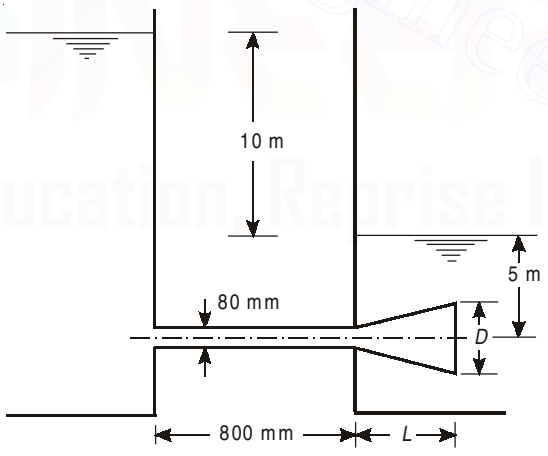


Figure Ex. 11.21

Solution

Applying Bernoulli's equation between the two reservoirs, we get

$$10 = 0.1\frac{v^2}{2g} + \frac{v^2}{2g} + \frac{fLv^2}{2gd}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$10 = \frac{v^2}{2g}\left[1.1+\frac{fL}{d}\right]$$

$f = 0.03$; $L = 800$ mm $= 0.8$ m; and $d = 80$ mm $= 0.08$ m

Thus by substitution, we get

$$10 = \frac{v^2}{2g}\left[1.1+\frac{0.03\times 0.8}{0.08}\right]$$

or $$v = 11.84 \text{ m/s}$$

$\therefore$ Discharge $$Q = \frac{\pi d^2}{4}\times v$$

$$= \frac{\pi\times(0.08)^2}{4}\times 11.84$$

$$= 0.0595 \text{ cumec}$$

When diffuser is attached to the pipe, then applying Bernoulli's equation between the two reservoirs, we get

$$10 = 0.1\frac{v^2}{2g}+\frac{V^2}{2g}+\frac{fLv^2}{2gd}+0.3\frac{(v-V)^2}{2g}$$

If Q is the discharge, then by continuity, we have

$$Q = \frac{\pi d^2}{4}\times v = \frac{\pi D^2}{4}\times V$$

Thus by substitution, we get

$$10 = \frac{Q^2}{(\pi/4)^2\times 2g}\left[\frac{0.1}{d^4}+\frac{1}{D^4}+\frac{fL}{d}\times\frac{1}{d^4}+0.3\left(\frac{1}{d^2}-\frac{1}{D^2}\right)^2\right]$$

or $$Q^2 = \frac{10\times(\pi/4)^2\times 2g}{\left[\frac{0.1}{d^4}+\frac{1}{D^4}+\frac{fL}{d}\times\frac{1}{d^4}+0.3\left(\frac{1}{d^2}-\frac{1}{D^2}\right)^2\right]}$$

Let $\frac{d}{D} = x$, then we have

$$Q^2 = \frac{10\times(\pi/4)^2\times 2g}{\left[\frac{0.1}{d^4}+\frac{fL}{d}\times\frac{1}{d^4}+\frac{x^4}{d^4}+\frac{0.3}{d^4}\left(1-x^2\right)^2\right]}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$Q = \frac{\sqrt{10\times(\pi/4)^2\times 2g}}{\sqrt{\left[\frac{0.1}{d^4}+\frac{fL}{d}\times\frac{1}{d^4}+\frac{1}{d^4}\left\{x^4+0.3(1-x^2)^2\right\}\right]}}$$

For Q to be maximum, $\frac{dQ}{dx}=0$.

Then since f, L and d are constant, we get

$$4x^3+0.3\times 2(1-x^2)(-2x) = 0$$

or $$x = 0.4804$$

$$\therefore \quad D = \frac{d}{0.4804} = 2.082\, d$$

$$d = 0.08 \text{ m}$$

$$\therefore \quad D = 0.08 \times 2.082 = 0.1666 \text{ m}$$

The maximum discharge is given by

$$Q_{max} = \frac{\sqrt{10\times(\pi/4)^2\times 2\times 9.81}}{\sqrt{\frac{0.1}{(0.08)^4}+\frac{0.03\times 0.8}{(0.08)^5}+\frac{1}{(0.08)^4}\left[(0.4804)^4+0.3\left\{1-(0.4804)^2\right\}^2\right]}}$$

$$\therefore \quad Q_{max} = 0.0887 \text{cumec}$$

The axial length L is given by

$$L = \frac{1}{2}(0.1666-0.08)\cot 8°$$

$$= 0.0433 \times 7.1154$$

$$= 0.3081 \text{ m} = 308.1 \text{ mm}$$

SUMMARY OF MAIN POINTS

1. A pipe is a closed conduit used for carrying fluids under pressure. Pipes are commonly circular in section. As pipes carry fluids under pressure, the pipes always run full.
2. The flow of fluid in a pipe be either laminar or turbulent.

   A dimensionless parameter called *Reynolds number* is used to distinguish these two types of flows in a pipe. Reynolds number is given by

   $$\text{Re or } N_R = \frac{\rho VD}{\mu} = \frac{VD}{\nu}$$

   where

   ρ = mass density of the flowing fluid ;
   V = mean or average velocity of flow of fluid;
   D = diameter of the pipe ; a
   μ = absolute or dynamic viscosity or viscosity of the flowing fluid ; and
   ν = kinematic viscosity of the flowing fluid $\left(=\frac{\mu}{\rho}\right)$

   The flow in a pipe will be laminar if the Reynolds number is less than 2000 and turbulent if it is more

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

than 4000. If the Reynolds number is between 2000 and 4000 the flow is considered to be in transition.

3. When a fluid flows through a pipe, certain resistance is offered to the flowing fluid which results in causing a loss of energy or head. The various energy or head losses in pipes may be classified as

   (i) Major losses

   (ii) Minor losses

   The major loss of energy or head is caused by friction and minor losses of energy or head are caused due to sudden expansion or enlargement of pipe, sudden contraction of pipe, bend in pipe and an obstruction in pipe.

4. The loss of energy or head due to friction is given by Darcy–Weisbach equation as

$$h_f = \frac{fLV^2}{2gD}$$

where

f = friction factor ;
L = length of pipe ;
V = mean velocity of flow in pipe ;
D = diameter of pipe ; and
g = acceleration due to gravity.

5. The loss of energy or head due to friction in pipe may also be determined by using the following formulae.

   (i) Chezy's formula, $V = C\sqrt{mi}$ or $V = C\sqrt{RS}$

   (ii) Manning's formula, $V = \frac{1}{n} R^{2/3} S^{1/2}$.

   (iii) Hazen-Willam's formula, $V = 0.85\, C_1 R^{0.63} S^{0.54}$

   where

V = mean velocity of flow in the pipe ;
C = Chezy's coefficient ;
m or R = hydraulic mean depth or hydraulic radius $\left(= \frac{A}{P}\right)$, A being wetted area and P being wetted perimeter of flow section of pipe.
i or S = slope of energy grade line $\left(= \frac{h_f}{L}\right)$, h_f being head loss due to friction and L being the length of pipe ;
n = Manning's roughness coefficient ; and
C_1 = Hazen–Williams coefficient.

Since pipe always runs full

$$m \text{ or } R = \frac{D}{4}$$

where

D = diameter of pipe.

6. (i) The loss of energy or head due to sudden enlargement or expansion is given by

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

where

V_1 = velocity of flow of fluid in smaller pipe ; and
V_2 = velocity of flow of fluid in larger pipe.

(ii) The loss of energy or head due to sudden contraction is given by

$$h_L = 0.5\, \frac{V^2}{2g}$$

where

V = velocity of flow of fluid in smaller pipe.

(iii) The loss of energy or head in a bend is given by

$$h_L = k\frac{V^2}{2g}$$

where

k = a coefficient the value of which mainly depends on the angle of the bend ; and
V = mean velocity of flow of fluid

(iv) The loss of energy or head due to fluid obstruction is given by

$$h_L = \left[\frac{A}{C_c(A-a)} - 1\right]^2 \frac{V^2}{2g}$$

where

A = cross-sectional area of pipe ;
α = maximum cross-sectional area of the obstruction ;
C_c = coefficient of contraction, assumed to vary from 0.60 to 0.66 ; and
V = mean velocity of flow of fluid at unobstructed section of pipe.

(v) The loss of energy or head due to various pipe fittings is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h_L = k\frac{V^2}{2g}$$

(vi) The loss of energy or head at the entrance to a pipe is given by

$$h_L = 0.5\frac{V^2}{2g}$$

(vii) The loss of energy or head at the exit of a pipe is given by

$$h_L = \frac{V^2}{2g}$$

7. The line representing the sum of pressure head and datum head with respect to some datum is known as *hydraulic grade line* (H.G.L).
   The line representing the total energy or head i.e., the sum of pressure head, datum head with respect to some datum and velocity head is known as *energy grade* line or *total energy* line (T.E.L)
8. When several pipes of different lengths and different diameters are connected in series, it is known as compound pipe. The discharge through each pipe connected in series is same. A compouned pipe may be replaced by an equivalent pipe.
   An *equivalent pipe* is a single pipe of uniform diameter and having same length, same discharge and same loss of energy or head as a compound pipe consisting of several pipes of different lengths and different diameters connected in series.
   The diameter of equivalent pipe is given by Dupuit's equation as

$$\frac{L}{D^5} = \left[\frac{L_1}{D_1^5} + \frac{L_2}{D_2^5} + \frac{L_3}{D_3^5} + ...\right]$$

   where
   L = length of equivalent pipe (= L_1 + L_2 + L_3 + ...) ;
   D = diameter of equivalent pipe ;
   $L_1, L_2, L_3, ...$ = lengths of different pipes connected in series ; and
   $D_1, D_2, D_3, ...$ = diameters of different pipes connected in series .
9. When the pipes are connected in parallel, the loss of head in each pipe is same. The discharge in the main pipe is equal to the sum of the discharges in each pipe connected in parallel.
10. For solving the problems of branched pipes i.e., pipes connecting different reservoirs, the three basic equations viz., continuity equation, Bernoulli's equation and Darcy–Weisbach equation are used.
11. A siphon is a long bent pipe which is used to carry water from a reservoir at a higher elevation to another reservoir at a lower elevation when the two reservoirs are separated by a hill or high level ground. The maximum vacuum pressure at the summit of a siphon may be 7.4 m of water.
12. The loss of energy or head due to friction in a tapering pipe of length l and diameters D_1 and D_2 at the ends ($D_1 > D_2$) is given by

$$h_f = \frac{4fQ^2l^5}{2g\lambda^2 D_2^5}\left[\frac{1}{l^4} - \frac{1}{(L+l)^4}\right]$$

    in which

$$l = \frac{D_2 L}{D_1 - D_2}$$

13. The loss of energy or head due to friction in a pipe of length L and diameter D with side tappings through which liquid is removed at a uniform rate of q per unit length of pipe is given by

$$h_f = \frac{f}{D}\frac{V^2}{2g}\frac{Q}{3q}\left[1 - \left(1 - \frac{qL}{Q}\right)^3\right]$$

    where Q is the discharge at the inlet end of the pipe.
    If the end of the pipe is closed so that $(Q - qL) = 0$; then

$$h_f = \frac{f}{D}\frac{V^2}{2g}\frac{L}{3}$$

14. Power transmitted through a pipe is given by

$$P = wQ = (H - h_f)$$

    where
    w = specific or unit weight of water ;
    Q = discharge ;
    H = total head at inlet of pipe ; and
    h_f = loss of head due to friction. The efficiency of power transmission through pipe is given by

$$\eta = \frac{H - h_f}{H}$$

    The power transmitted through a pipe is maximum when $h_f = \frac{H}{3}$, and corresponding

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

efficiency $\eta = \frac{2}{3}$ or 66.67%

15. The velocity of flow at the outlet of the nozzle fitted at the end of a pipe is given by

$$v = \sqrt{\frac{2gh}{\left(1+\frac{fL}{D}\frac{a^2}{A^2}\right)}}$$

where

H = head at the inlet of the pipe ;
L = length of the pipe ;
D = diameter of the pipe ;
a = cross-sectional area of the nozzle outlet ; and
A = cross-sectional area of the pipe.

The power transmitted by the jet issuing from the nozzle is given by

$$P = wQ\,(H - h_f)$$

in which

$$Q = av = AV \text{ ; and}$$

$$h_f = \frac{fLV^2}{2gD} = \frac{fL}{D}\frac{a^2}{A^2}\frac{V^2}{2g}$$

The efficiency of power transmission through nozzle is given as

$$\eta = \left[\frac{1}{1+\frac{fL}{D}\frac{a^2}{A^2}}\right]$$

The power transmitted by a nozzle is maximum when $h_f = \frac{H}{3}$

The diameter of the nozzle transmitting maximum power is given by

$$d = \left(\frac{D^5}{2fL}\right)^{1/4}$$

where

d = diameter of the nozzle ;
D = diameter of the pipe ;
L = length of the pipe, and
f = friction factor

16. When water flowing in a long pipe is suddenly brought to rest by closing the valve or by any similar cause, there will be a sudden rise in pressure and a pressure wave of high intensity will be transmitted along the pipe. This phenomenon of sudden rise in pressure in the pipe is known as *water hammer* or *hammer blow*.

The intensity of pressure rise due to water hammer is given by

(i) When valve is closed gradually

$$p_i = \frac{wLV}{gt}$$

(ii) When valve is closed suddenly and pipe is elastic

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}}$$

(iii) When valve is closed suddenly and pipe is considered rigid

$$p_i = V\sqrt{\frac{wK}{g}}$$

or $$\frac{p_i}{w} = \frac{V}{g}\sqrt{\frac{K}{\rho}} = \frac{VV_o}{g}$$ (Allievi formula)

where

L = length of pipe ;
V = velocity of flow ;
t = time taken to close the valve ;
D = diameter of the pipe ;
K = bulk modulus of water ;
E = Young's modulus of pipe material ;
T = wall thickness of pipe ;
m = poisson's ratio ;
w = specific or unit weight of water ;
ρ = mass density of water ; and
g = acceleration due to gravity.

$V_0 = \sqrt{\frac{K}{\rho}}$ = velocity of propogation of sound wave in the liquid.

17. For solving the problems of pipe network the conditions to be satisfied are as follows :
    (i) The flow into each junction must be equal to the flow out of the junction.
    (ii) In each loop the loss of head due to flow in clockwise direction must be equal to the loss of head due to flow in anticlockwise direction.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) The Darcy–Weisbach equation must be satisfied for flow in each pipe, according to which the loss of head h_f through any pipe discharging at the rate of Q can be expressed as

$$h_f = r Q^n$$

where r is a proportionality factor $\left(= \frac{fL}{2g(\pi/4)^2 D^5} = \frac{fL}{12.10 D^5} \right)$ and n is an exponent having a numerical value ranging from 1.72 to 2.00

PROBLEMS

11.1 Distinguish between laminar flow and turbulent flow in pipes.

11.2 Explain the terms hydraulic gradient and total energy lines.

11.3 What is a compound pipe? How would you determine the equivalent size of a compound pipe?

11.4 Why are the pipes connected in parallel?

11.5 What is a siphon? Where is it used? Explain its working.

11.6 Obtain the condition for maximum efficiency in transmission of power through a pipeline?

11.7 What is meant by water hammer ? Obtain an expression for the rise in pressure in a thin elastic pipe of circular section in which the flow of water is stopped by sudden closure of valve.

11.8 A pipe line, 16 km long, supplies, 40 million litres of water per day to city. The first 5 km length of the pipe is of 1 m diameter and the remaining part is 0.8 m diameter pipe. If the water to the city is to be supplied at a residual head of 15 m of water, calculate the supply head at the inlet end. Neglect minor losses and assume $f = 0.03$ for the entire pipe line. Sketch the hydraulic gradient for the pipe line.
[*Ans.* 35.53 m]

11.9 A pipe 0.15 m diameter taking off from a reservoir suddenly expands to 0.3 m at the end of 16 m and continues for another 15 m. If the head above the inlet of the pipe is 4.88 m, determine the actual velocity at the exit, taking into consideration all the losses. Assume $f = 0.04$ for the complete pipe line. [*Ans.* 1.07 m/s]

11.10 A hydraulic power main 4.5 km long and 0.225 m in diameters is tapped at various points throughout its length at an average rate 0.00135 m^3/sec per 100 m length. Assuming $f = 0.02$, calculate the power lost due to friction in the pipe if (i) the pumping station is situated, at one end of the main, the other end being closed; (ii) pumping station is placed mid-way, the two ends of the main being closed. Start from first principles. [*Ans.* (i) 7.09 kW; (ii) 0.89 kW]

11.11 Two reservoirs are cennected by a pipe 2250 m long and 0.225 m in diameter, the difference in water levels being 7.5 m. Determine the flow through the pipe in litres per minute if $f = 0.03$. Also find the percentage increase in the discharge if for the last 600 m a second pipe of the same diameter is laid along side the first.
[*Ans.* 1666.61 litres/min; 12.1%]

12.12 Two reservoirs are connected by three pipes laid in parallel, their diameters are d, $2d$ and $3d$ respectively, and they are of the same length l. Assuming f to be the same for all pipes, determine the discharge through each of the larger pipes if the smallest pipe is discharging 1 cumec. [*Ans.* 5.66 m^3/sec; 15.6 m^3/sec]

11.13 Water flows out of tank, wherein water stands at El. 140 m; through a pipe 4 km long and 0.3 m diameter upto a point where the elevation is El. 90 m and one-third of the discharge flowing is tapped out to be used by a power machine. The remaining discharge flows down through another pipe, 5 km long and 0.2 m in diameter, into another tank wherein water stands at El. 110 m. Use Darcy's formula $h_f = \left(\frac{LQ^2}{600 D^5} \right)$ with usual notation. Determine the power available at the tapping point for the power machine.
[*Ans.* 6.63 kW (9.01 h.p.)]

11.14 Two reservoirs whose water surface are at an elevation difference of H meters are connected by a pipeline, of diameter D metres and length L metres, which is uniformly tapped all along

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

its length such that the discharge entering it is halved before it reaches the other end. Derive an expression for the discharge Q in cumec entering the pipeline. Use Darcy's formula $h_f = \left(\frac{LQ^2}{600D^5}\right)$. Start from first principles. Neglect minor losses. $\left[Ans. \sqrt{\frac{7200D^5H}{7L}}\right]$

11.15 A pipe 0.2 m diameter and 1800 m long connects two reservoirs one being 30 m below the other. The pipeline crosses a ridge whose summit is 7.5 m above the upper reservoir. What may be the minimum depth of the pipe below the summit of the ridge in order that the pressure at the apex does not fall below 7.5 m vacuum? The length of the pipe from the upper reservoir to the apex is 300 m. Taking $f = 0.032$, determine the rate of flow to the lower reservoir in litres per minute. [*Ans.* 2.37m; 2688 litres/min]

11.16 A turbine operating under a total fall of 36 m is supplied water through 90 m long 0.2 m diameter pipe. If the flow rate is such that 9.5 m of head is lost in friction in the pipe, what will be the power delivered to the turbine? Take $f = 0.022$. [*Ans.*35.43 kW(48.16 h.p.)]

11.17 The water levels in the two reservoirs A and B are respectively 66 m and 61.5 m above datum. A pipe joins each to a common point D where the pressure is 103 kN/m² gauge and height is 45 m above datum. Another pipe connects D to another tank C. What will be the height of water level in C assuming the same value of friction factor for all pipes?

Pipe	*Length*	*Diameter*
AD	2400 m	0.30 m
BD	2700 m	0.45 m
CD	3000 m	0.60 m

[*Ans.* 51.90 m above datum]

11.18 Water is flowing in a long pipe which is 0.15 m diameter and 6.25 mm thick with a velocity of 1.2 m/s and it is suddenly brought to rest by the closing of a valve. Calculate the inertia pressure and the theoretical stress produced in the pipe near the valve, taking K for water as 207×10^4 kN/m² [21.1×10^3 kg(f)/cm²] and E for the pipe material as 206×10^6 kN/m² [21×10^5 kg(f)/cm²].

[*Ans.*155×10^4 N/m² {15.8 kg (f)/cm²}; 1.86×10^4 kN/m² {189.6 kg(f)/cm²}]

11.19 A 0.3 m wrought iron pipeline 560 m long discharges water 100 m below the surface of reservoir. Determine the diameter of the nozzle which will deliver the maximum power. Assume $f = 0.022$ and the coefficient of velocity for the nozzle is 0.97. [*Ans.* 101.2 mm]

11.20 A water main of concrete pipe 3200 m long and 0.3 m diameter discharges into a reservoir at the rate of 10×10^6 litres per day. If this line is gradually closed by operating a valve at the reservoir end in 16 seconds, show that there is a possibility of pipe burst. The safe pressure of concrete pipe is 245.25 kN/m² [2.5 kg(f)/cm²]

[*Ans.* p_i = 327.48 kN/m² {3.34 kg(f)/cm²}, which is greater than safe pressure of concrete pipe and hence there is possibility of pipe burst]

11.21 Show that a long pipe with nozzle at its end has greater velocity of efflux but less discharge as compared to a similar pipe without nozzle at its end.

11.22 Water flows through a horizontal pipe 2 m long, which tapers from a diameter of 0.2 m to 0.15 m in the length 2 m. A constant discharge of 40 litres per second flows through the pipe. Starting from first principles, determine the loss of head due to friction. Take $f = 0.04$.

[*Ans.* 0.0714 m]

11.23 A pipe 0.15 m diameter, 15 m long is connected to the bottom of a tank 15 m long and 12 m wide. The original head over the open end of the pipe is 5 m. Find the time of emptying the tank. Assume the entrance to the pipe to be sharp-edged and $f = 0.01$ in $h_f = \frac{flV^2}{2gd}$.

[*Ans.* 4.52 hrs.]

11.24 Two reservoirs A and C having a difference of level of 15.5 m are connected by a pipeline ABC, the elevation of the point B being 4 m below the level of water in the reservoir A. The length AB of the pipeline is 250 m, the pipe being made of mild steel having a friction coefficient f_1, while

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the length BC is 450 m, the pipe being made of cast iron having a friction coefficient f_2. Both the lengths AB and BC have a diameter 0.20 m. A partially closed valve is located in the length BC at a distance of 150 m from reservoir C. If the flow through the pipeline is 3000 litres/minute, the pressure head at B is 0.5 m and the head loss at the valve is 5.0 m, find the friction coefficients f_1 and f_2. Draw the hydraulic grade line of the pipeline and indicate on the diagram head loss values of significant points. Take into account head loss at entrance and exit points of the pipeline. [*Ans.* $f_1 = 0.0205$, $f_2 = 0.0241$]

11.25 If two pipes of diameter D and d and equal length L are arranged in parallel the loss of head for a flow of Q is h. If the same pipes are arranged in series the loss of head for the same flow Q is H. If $d = 0.5\ D$, find the percentage of total flow through each pipe when placed in parallel and the ratio (H/h). Neglect minor losses and assume f to be constant. [*Ans.* 85%, 15%; 45.7]

11.26 Deduce an expression for loss of head in a pipeline with laterals assuming that velocity changes uniformly from V_1 to V_2 in a length L of the pipeline of constant diameter D and friction factor f.

$$\left[Ans.\ \frac{fL}{6gD}\left(V_1^2 + V_2^2 + V_1V_2\right) \right]$$

[**Note** : The expression obtained in this case is same as given by Eq. 11.22. By considering $\left(Q/\frac{\pi D^2}{4} = V_1\right)$ and $\left[\left(Q - qL\right)\Big/\frac{\pi D^2}{4} = V_2\right]$, we obtain $\left[\left(qL/Q\right) = \left(\left(V_1 - V_2\right)/V_1\right)\right]$ and introducing this in Eq. 11.22 the above noted expression may be obtained.]

11.27 A pipe 0.6 m in diameter takes off from a reservoir 150 m above the datum. The pipe is 5000 m long and is laid completely at the datum level. For the last 1200 m water is drawn by service pipes at a uniform rate of 0.088 cumec per 300 m. Find the pressure at the end of the pipeline (friction factor $f = 0.04$). Take the other end as dead end, i.e., the velocity at the other end as zero.

[*Ans.* 1254.29 kN/m^2 {12.79 kg(f)/cm^2}]

11.28 For supplying water to a colony, it is intended to store water in a large tank whose water surface elevation is presently 85.37 m R.L. This tank is being supplied by a pipeline from a junction J, the friction loss in metres of water in this pipe being $0.6\ Q^{1.95}$, where Q is the discharge in m^3/s. Junction J receives water under pressure and without leakage from (i) a large reservoir I whose water surface elevation is constant at 100.00 m R.L., through a pipe characterised by $h_f = 0.35\ Q^2$; and (ii) a second large reservoir II with its constant water level at 96.00 m R.L., through a pipe characterised by $h_f = 0.46\ Q^{1.85}$. Neglecting losses other than due to pipe friction, determine the incoming flow into the tank in the colony and the piezometric head at the junction J.

[*Ans.* Flow from Tank I = 3.506 m^3/s, from Tank II = 0.797 m^3/s, total flow into the colony Tank = 4.303 m^3/s; Piezometric head at the junction J = 95.698 m R.L.]

11.29 Derive the expression for the pressure rise due to water hammer caused by a sudden closure of the discharge through the pipe if
(1) the pipe elasticity is considered;
(2) the Poisson's ratio is not considered; and
(3) the pipe elasticity is fully neglected.

(1) $p_i = \dfrac{V}{\sqrt{\dfrac{g}{w}\left[\dfrac{1}{K} + \dfrac{D}{TE}\left(1 - \dfrac{1}{2m}\right)\right]}}$;

(2) $p_i = \dfrac{V}{\sqrt{\dfrac{g}{w}\left(\dfrac{1}{K} + \dfrac{D}{TE}\right)}}$

(3) $p_i = V\sqrt{\dfrac{wK}{g}}$

11.30 Two reservoirs with 15 m difference in their water levels are connected by a 300 mm diameter pipe line of 3000 m length. Calculate the discharge. If a parallel pipe line of 300 mm diameter is attached to the last 1500 m length of existing pipe, determine the modified dischage. Take only wall friction into account. Assume $f =$ 0.04 in Darcy–Weisbach formula.

[*Ans.* 0.0606 m^3/s; 0.0767 m^3/s]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Boundary Layer Theory

Chapter **12**

12.1 INTRODUCTION

When a real fluid flows past a solid boundary, a layer of fluid which comes in contact with the boundary surface adheres to it on account of viscosity. Since this layer of fluid cannot slip away from the boundary surface it attains the same velocity as that of the boundary. In other words, at the boundary surface there is no relative motion between the fluid and the boundary. This condition is known as *no slip* condition. If the boundary is moving, the fluid adhering to it will have the same velocity as that of the boundary. However, if the boundary is stationary, the fluid velocity at the boundary surface will be zero. Thus at the boundary surface the layer of fluid undergoes retardation. This retarded layer of fluid further causes retardation for the adjacent layers of the fluid, thereby developing a small region in the immediate vicinity of the boundary surface in which the velocity of flowing fluid increases gradually from zero at the boundary surface to the velocity of the main stream. This region is known as *boundary layer*. In the boundary layer region since there is a larger variation of velocity in a relatively small distance, there exists a fairly large velocity gradient $(\partial v / \partial y)$ normal to the boundary surface. As such in this region of boundary layer even if the fluid has small viscosity, the corresponding shear stress $\tau = \mu (\partial v / \partial y)$, is of appreciable magnitude. Farther away from the boundary this retardation due to the presence of viscosity is negligible and the velocity there will be equal to that of the main stream. The flow may thus be considered to have two regions, one close to the boundary in the boundary layer zone in which due to larger velocity gradient appreciable viscous forces are produced and hence in this region the effect of viscosity is mostly confined; and second outside the boundary layer zone in which the viscous forces are negligible and hence the flow may be treated as non-viscous or inviscid. The concept of boundary layer was first introduced by L. Prandtl in 1904 and since then it has been applied to several fluid flow problems. In the following paragraphs the characteristics of flow in the boundary layer are discussed.

12.2 THICKNESS OF BOUNDARY LAYER

The velocity within the boundary layer increases from zero at the boundary surface to the velocity of the main stream asymptotically. Therefore the thickness of the boundary layer represented by δ (Greek 'delta') is arbitrarily defined as that distance from the boundary surface in which the velocity reaches 99% of the velocity of the main stream. In other words, the boundary layer thickness δ may be considered equal to the distance y from the boundary surface at which $v = 0.99$ V. This definition however gives an

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

approximate value of the boundary layer thickness and hence δ is generally termed an *nominal thickness* of the boundary layer. For greater accuracy the boundary layer thickness is defined in terms of certain mathematical expressions which are the measures of the effect of boundary layer on the flow. Three such definitions of the boundary layer thickness which are commonly adopted are the *displacement thickness* δ*, the *momentum thickness* θ and the *energy thickness* δ_E.

The *displacement thickness* δ* is defined as the distance by which the boundary surface would have to be displaced outwards (as shown in Fig. 12.1) so that the total actual discharge would be same as that of an ideal (or frictionless) fluid past the displaced boundary. Thus, it may be expressed by the equation

$$V\delta^* = \int_0^\infty (V - v)\,dy$$

or

$$\delta^* = \int_0^\infty \left(1 - \frac{v}{V}\right) dy \qquad \ldots (12.1)$$

Another definition for the displacement thickness may be given which is based on the fact that because of the retardation of the flow within the boundary layer, in order to maintain the continuity of flow the streamlines outside the boundary layer are farther from the boundary than they would be if

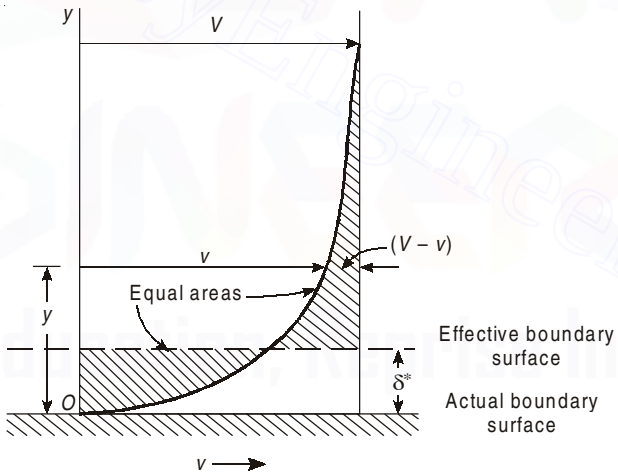


Figure 12.1 *Displacement thickness of boundary layer*

the fluid was ideal or inviscid upto the boundary. The displacement thickness δ* may then be defined as the distance by which the external streamlines are shifted or displaced outwards owing to the formation of the boundary layer.

The retardation of flow due to viscosity in the boundary layer would cause the reduction in the momentum flux. Accordingly the *momentum thickness* θ is defined as the distance from the actual boundary surface such that the momentum flux corresponding to the main stream velocity *V* through this distance θ is equal to the deficiency or loss in momentum due to the boundary layer formation. Thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\rho V^2\theta = \rho\int_0^\infty (V-v)v\,dy$$

or
$$\theta = \int_0^\infty \frac{v}{V}\left(1-\frac{v}{V}\right)dy \qquad ...(12.2)$$

Further the retardation of flow due to viscosity in the boundary layer would cause the reduction in the flux of energy. As such the *energy thickness* δ_E is defined as the distance from the actual boundary surface such that the energy flux corresponding to the main stream velocity V through this distance δ_E is equal to the deficiency or loss of energy due to the boundary layer formation. Thus

$$\frac{1}{2}\rho V^3\delta_E = \frac{1}{2}\rho\int_0^\infty \left(V^2-v^2\right)v dy$$

or
$$\delta_E = \int_0^\infty \frac{v}{V}\left(1-\frac{v^2}{V^2}\right)dy \qquad ...(12.3)$$

12.3 BOUNDARY LAYER ALONG A LONG THIN PLATE AND ITS CHARACTERISTICS

Consider a long thin plate held stationary in the direction parallel to the flow in a uniform stream of velocity V as shown in Fig. 12.2. The plate is said to be held at zero incidence to the velocity of flow and the velocity of flow is known as 'free stream velocity' or 'ambient velocity' or 'potential velocity'. At the leading edge of the plate the thickness of the boundary layer is zero, but on downstream, for the fluid in contact with the boundary the velocity of flow is reduced to zero and at some distance δ from the boundary the velocity is nearly V. Hence a velocity gradient is set up which develops shear resistance to the flow and retards the motion of the fluid. Near the leading edge of the plate the fluid is retarded in a thin layer. In other words, the boundary layer near the leading edge is relatively thin. As this retarded layer of fluid moves downstream, due to continued action of shear resistance more and more fluid is retarded. Thus the thickness of the boundary layer δ, goes on increasing in the downstream direction as shown in Fig. 12.2. The various factors which influence the thickness of the boundary layer forming along a flat smooth plate are noted below.

(1) The boundary layer thickness increases as the distance from the leading edge increases.

(2) The boundary layer thickness decreases with the increase in the velocity of flow of the approaching stream of fluid.

(3) Greater is the kinematic viscosity of the fluid greater is the boundary layer thickness.

(4) The boundary layer thickness is considerably affected by the pressure gradient $(\partial p/\partial x)$ in the direction of flow. In the case of a flat plate placed in a stream of uniform velocity V, the pressure may also be assumed to be uniform i.e., $(\partial p/\partial x) = 0$. However, if the pressure gradient is negative as in the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

case of a converging flow, the resulting pressure force acts in the direction of flow and it accelerates the retarded fluid in the boundary layer. As such the boundary layer growth is retarded in the presence of negative pressure gradient. On the other hand if the pressure gradient is positive as in the case of

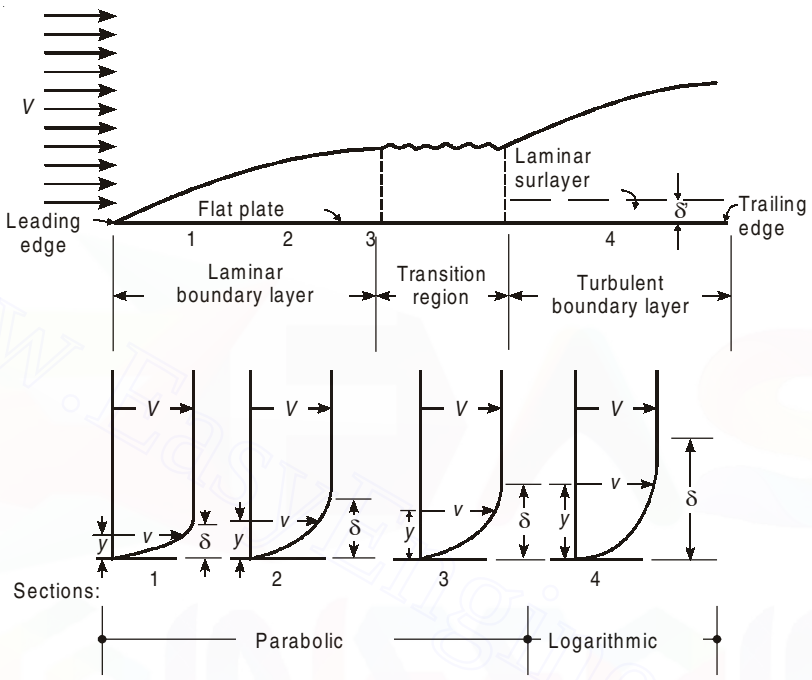


Figure 12.2 *Boundary layer and velocity distribution at successive points along a flat plate*

divergent flow the fluid in the boundary layer is further decelerated and hence assists in thickening of the boundary layer. In the later case back flow and separation may be caused as discussed in Section 12.10.

As the boundary layer develops, upto a certain portion of the plate from the leading edge, the flow in the boundary layer exhibits all the characteristics of laminar flow. This is so irrespective of whether the flow of the incoming stream is laminar or turbulent. This is known as *laminar boundary layer*. If the plate is sufficiently long, then beyond some distance from the leading edge the laminar boundary layer becomes unstable and the flow in the boundary layer exhibits the characteristics between those of laminar and turbulent flow. This region of the boundary layer is usually small and is known as transition region. After the transition region the flow in the boundary layer becomes turbulent. In this portion of the boundary layer there is a rapid increase in its thickness and it is known as *turbulent boundary layer*. If the plate is very smooth, even in the region of turbulent boundary layer, there is a very thin layer just adjacent to the boundary, in which the flow is laminar. This thin layer is commonly known as *laminar sublayer*, and its thickness is represented by δ'.

The velocity distribution in a laminar boundary layer is parabolic $(V - v) \sim (\delta - y)^2$; and for turbulent boundary layer the velocity distribution has been found to follow approximately either the one-

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

seventh power law: $v \sim y^{1/7}$ or it is logarithmic, $v \sim \log y$. For laminar sublayer the velocity distribution is parabolic, but since its thickness δ' is usually very small, a linear distribution can be assumed.

The change of boundary layer from laminar to turbulent mainly depends on the velocity of flow V, of the approaching stream of fluid, the length x measured along the plate from the leading edge, the mass density ρ of fluid and its dynamic viscosity μ. As such the Reynolds number $Re_x = (\rho V x/\mu)$ (the suffix x indicating that it is calculated with the distance x as the characteristic length) becomes a significant parameter in indicating the change of boundary layer from laminar to turbulent. The value of Re_x at which the boundary layer may change from laminar to turbulent varies from 3×10^5 to 6×10^5. In exceptional cases, where the approaching flow is free from any disturbance the values of critical Reynolds number can reach 10^6 or even higher. However, change of boundary layer from laminar to turbulent is also affected by several other factors such as roughness of the plate, plate curvature, pressure gradient and intensity and scale of turbulence.

12.4 BOUNDARY LAYER EQUATIONS

The equations of continuity and motion for the steady flow of an incompressible, inviscid fluid in two dimensions without body forces are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad \ldots \text{(i)}$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} \quad \ldots \text{(ii)}$$

and

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} \quad \ldots \text{(iii)}$$

where x and y are rectangular cartesian coordinates and u and v are the corresponding velocity components.

Now if a viscous fluid is considered then the equation of continuity will be unchanged, but in the equations of motion additional terms will be introduced due to viscous stresses as indicated below.

Consider the boundary layer on a flat plate with x measured along the plate and y normal to it as shown if Fig. l2.3. The only viscous stress τ that need be considered is that acting in the direction parallel to the plate. Thus if a small element of fluid of area $\delta x \ \delta y$ in the xy plane and unit length normal to the xy plane is considered, then on its lower surface there is a force, due to shear stress, equal to $-\tau \delta x$ in the x direction. Likewise on the upper surface there is a force in the x direction due to shear stress as

$$(\tau + \partial \tau)\ \delta x = \left[\tau + \frac{\partial \tau}{\partial y}\delta y\right]\delta x$$

neglecting the terms of higher power of δy.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

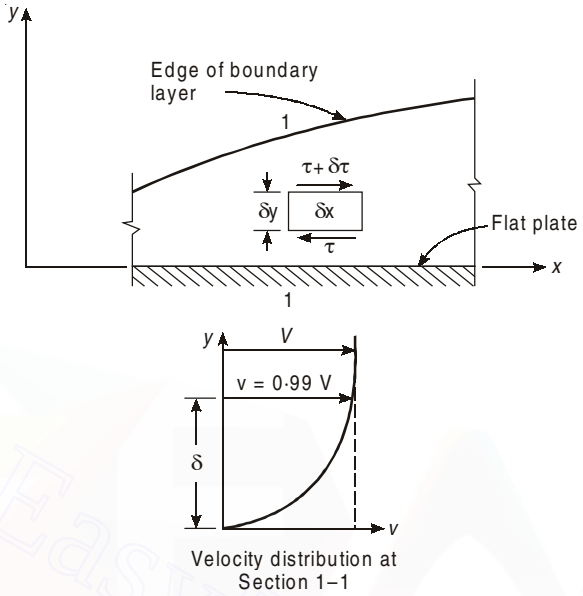


Figure 12.3 *Boundary layer along a flat plate*

The net force in the x direction acting on the element is therefore $(\partial\tau/\partial y)\,\delta x\,\delta y$, and hence as δx and δy tend to be zero, the force per unit mass in the x direction due to viscous stresses becomes

$$\frac{1}{\rho}\frac{\partial \tau}{\partial y}$$

Since the left-hand side of Eq. (ii) noted above represents the acceleration in x direction of a particle of fluid and the right-hand side represents the components of the force per unit mass in that direction due to static pressure, it follows that the above term due to viscous stresses should be added on the right-hand side of the equation to complete the equation of motion in the x direction in the boundary layer. This equation is then modified as

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{1}{\rho}\frac{\partial \tau}{\partial y}$$

Further for viscous flow in the laminar boundary layer from Newton's law of viscosity

$$\tau = \mu\frac{\partial u}{\partial y}$$

and hence the equation of motion becomes

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon\frac{\partial^2 u}{\partial y^2}$$

where υ is kinematic viscosity (μ/ρ).

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The second equation of motion i.e., Eq. (iii) is unchanged by the argument that the shear stress is acting in the x direction only. However, by determining the order of magnitude of each of the terms of Eq. (iii) and neglecting the terms of smaller order of magnitude, Eq. (iii) reduces to

$$-\frac{1}{\rho}\frac{\partial p}{\partial y} = 0.$$

Thus the equations that govern the flow in the steady, two dimensional laminar boundary layer on a flat plate are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad ..(12.4)$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon\frac{\partial^2 u}{\partial y^2} \qquad ...(12.5)$$

and

$$-\frac{1}{\rho}\frac{\partial p}{\partial y} = 0 \qquad ...(12.6)$$

Equations 12.4, 12.5 and 12.6 are generally known as *Prandtl's boundary layer equations* for two-dimensional steady flow of incompressible fluids. In the above analysis, Eq. 12.5 and 12.6 have been derived by adopting a relatively simpler approach. But the same can also be derived directly from the Navier–Stokes equations which are in fact the basic equations of motion for the flow of viscous fluids. The analysis involving the use of Navier–Stokes equations is more accurate and complete, since in this all the viscous stresses are included. However, for deriving Eqs 12.5 and 12.6 from the Navier-Stokes equations the order of magnitude of each of the terms of these equations is determined and the terms of smaller order of magnitude are neglected.

By integrating Eq. 12.6, we get

$$p = f(x) \text{ for steady flow}$$

and

$$p = f(x, t) \text{ for unsteady flow}$$

from which it may be concluded that the pressure p does not vary in the y direction or in other words the pressure remains constant across the boundary layer. Hence the partial derivative$(\partial p/\partial x)$ in Eq. 12.5 can be replaced by the total derivative (dp/dx) and Eq. 12.5 then becomes

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{dp}{dx} + \upsilon\frac{\partial^2 u}{\partial y^2} \qquad ...(12.7)$$

Since outside the boundary layer the fluid may be treated as inviscid (or non-viscous), the Euler's equations of motion may be applied according to which

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x}$$

In the region outside the boundary layer since $v = 0$ and $u = V$ the free stream or ambient velocity of the approaching stream, by substituting these values in the above equation, it becomes

$$V\frac{\partial V}{\partial x} = -\frac{1}{\rho}\frac{\partial p}{\partial x}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$V\frac{\partial V}{\partial x} = -\frac{1}{\rho}\frac{dp}{dx} \qquad ...(12.8)$$

as both V and p are the functions of x only. Integration of Eq. 12.8 leads to the Bernoulli's equation at any section viz.,

$$\frac{p}{\rho} + \frac{V^2}{2} = \text{constant}$$

Further introducing Eq. 12.8 in Eq. 12.7 it becomes

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = V\frac{dV}{dx} + \upsilon\frac{\partial^2 u}{\partial y^2} \qquad ...(12.9)$$

12.5 MOMENTUM INTEGRAL EQUATION OF THE BOUNDARY LAYER

The boundary layer equations obtained in the preceding sections are nonlinear partial differential equations and hence difficult to solve. As such approximate methods for the solution of these equations have been developed for the case for which either the exact solutions cannot be obtained or the same may involve a lot of computations. One such approximate method developed by Theodore von Karman is based on the application of the momentum equation to the boundary layer and the corresponding equation derived by him is known as *momentum integral equation of the boundary layer or von Karman's integral equation*. The momentum integral equation of the boundary layer expresses the relation that must exist between the overall rate of flux of momentum across a section of the boundary layer, the shear stress at the boundary surface and the pressure gradient in the direction of flow. For steady, two-dimensional flow in the boundary layer along a flat plate this equation may be derived as indicated below.

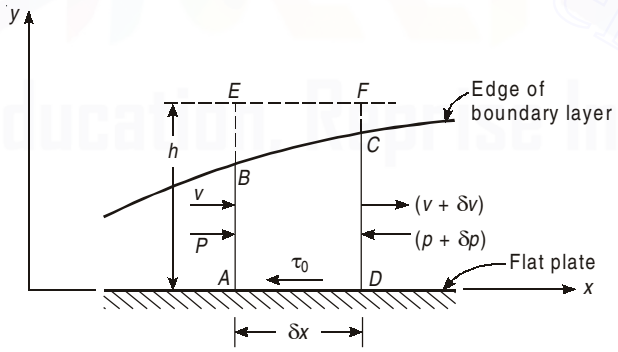


Figure 12.4 *Definition sketch for the derivation of momentum integral equation*

Consider an element of the boundary layer formed along a flat plate, between sections AB and CD, at a small distance δx apart as shown in Fig. 12.4. The boundary layer is of thickness δ and its outer edge is represented by BC. Let AB and CD be extended upto E and F where $AE = DF = h$, such that h is slightly greater than the local boundary layer thickness δ. $AEFD$ may be taken as a suitable control volume for this analysis.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For unit width in the direction perpendicular to the plane of the paper, the rate of mass flow across *AE* into *AEFD*

$$= \int_0^h \rho v \, dy$$

and the corresponding rate of mass flow across *DF* out of *AEFD*

$$= \int_0^h \rho v \, dy + \frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x$$

neglecting the terms of higher power of δx.

Hence the net rate of mass flow across *DF* and *AE*, out of *AEFD*

$$= \frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x \qquad ...(i)$$

If v_h denotes the mean value of velocity in the y direction at height h above the surface over the length *EF* then the rate of mass flow out of *AEFD* across *EF*

$$= \rho v_h \delta x$$

But continuity of mass requires that there is no net rate of change of mass inside *AEFD* and hence

$$\rho v_h = -\frac{d}{dx}\left[\int_0^h \rho v dy\right] \qquad ...(ii)$$

Now for the fluid in *AEFD*, the balance of rate of change of momentum in the x direction and components of applied force in that direction is considered.

The rate of transport of momentum in the x direction across *DF* minus the rate of transport of momentum in the x direction across *AE* is

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right]\delta x$$

The rate of transport of momentum in the x direction across *EF* out of *AEFD* is

$$\rho v_h V \delta x = -V \frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x$$

from Eq. (ii), where V represents the velocity (in the x direction) of the main stream outside the boundary layer at section *AB*.

The force in the x direction on *AEFD* due to the pressure

$$= -h\delta p = -h \frac{dp}{dx} \delta x$$

and that due to the shear stress at the boundary surface

$$= -\tau_0 \delta x$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where τ_0 is the mean shear stress over the length *AD*.

Thus equating the net increase in the rate of transport of momentum to the sum of the forces acting in the *x* direction, we have

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right]\delta x - V\frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x = -h\frac{dp}{dx}\delta x - \tau_0 \delta x$$

Dividing both the sides of the above equation by δx and taking the limit $\delta x \to 0$, we get

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right] - V\frac{d}{dx}\left[\int_0^h \rho v dy\right] = -h\frac{dp}{dx} - \tau_0 \quad \ldots (12.10)$$

By introducing Eq. 12.8 in Eq. 12.10 it becomes

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right] - V\frac{d}{dx}\left[\int_0^h \rho v dy\right] = h\rho V\frac{dV}{dx} - \tau_0$$

Further the above equation can be written in the following form

$$\frac{d}{dx}\left[\int_0^h \rho v(v-V)dy\right] + \frac{dV}{dx}\left[\int_0^h \rho v dy\right] = h\rho V\frac{dV}{dx} - \tau_0$$

or $$\frac{d}{dx}\left[\int_0^h \rho v(V-v)dy\right] + \frac{dV}{dx}\left[\int_0^h \rho (V-v)dy\right] = \tau_0$$

Since $(V-v)$ becomes zero at the edge of the boundary layer the upper limit to both integrals in the above equation may be changed to ∞. Then from the Eqs. 12.1 and 12.2 defining the displacement thickness δ^* and momentum thickness θ the above equation simplifies to

$$\frac{d}{dx}(\rho V^2\theta) + \frac{dV}{dx}(\rho V\delta^*) = \tau_0$$

or $$\frac{d}{dx}(V^2\theta) + \frac{dV}{dx}(V\delta^*) = \frac{\tau_0}{\rho} \quad \ldots (12.11)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 12.11 is the *momentum integral equation* of the boundary layer which forms the basis for approximate methods of solving boundary layer problems. Since in the derivation of this equation no assumption has been made regarding the nature of flow in the boundary layer, it is applied to both laminar as well as turbulent boundary layers. Further it may be noted that if for a flat plate at zero incidence $(\partial p/\partial x) = 0$ then from Eq. 12.8, $(\partial V/\partial x)$ or $(dV/dx) = 0$ and hence Eq. 12.11 reduces to

$$\frac{\tau_0}{\rho} = \frac{d}{dx}(V^2\theta)$$

or
$$\frac{\tau_0}{\rho V^2} = \frac{d\theta}{dx} \qquad ...(12.12)$$

The momentum integral Eq. 12.11 may also be derived by integration of boundary layer Eq. 12.9.

12.6 LAMINAR BOUNDARY LAYER

Inside the boundary layer since the viscous forces are predominant, it is reasonable to assume that the inertial and viscous forces are of the same order of magnitude in a laminar boundary layer. The inertial forces per unit volume is given by $(\rho v \partial v/\partial x)$ which is proportional to $(\rho V^2/x)$ for the case of a flat plate. Similarly the viscous force per unit volume is $(\partial\tau/\partial y)$ which for laminar flow becomes $(\partial/\partial y)(\mu \partial v/\partial y)$ i.e., $(\mu \partial^2 v/\partial y^2)$. In this boundary layer since $(\partial v/\partial y) \sim (V/\delta)$, hence $(\partial\tau/\partial y) \sim (\mu V/\delta^2)$. Thus if these forces are proportional to each other, then

$$\rho \frac{V^2}{x} = k'\mu \frac{V}{\delta^2}$$

or
$$\frac{\delta^2}{x^2} = \frac{k'\mu}{\rho V x} = \frac{k'}{\mathrm{Re}_x}$$

$$\therefore \quad \frac{\delta}{x} = \frac{k}{\sqrt{Re_x}}; \delta = k\sqrt{\frac{x\upsilon}{V}} \qquad ...(12.13)$$

in which k' and k are constants. By exact analytical solution of the boundary layer equations Blasius has obtained the value of the constant k as 5. Thus

$$\frac{\delta}{x} = \frac{5}{\sqrt{\mathrm{Re}_x}}; \text{ or } \delta = 5\sqrt{\frac{x\upsilon}{V}} \qquad ...(12.14)$$

Further the expression for the shear stress τ_0 can also be obtained as indicated below.

Since in the boundary layer

$$\left(\frac{\partial v}{\partial y}\right)_{y=0} \sim \frac{V}{\delta}; \tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=0} \sim \mu\frac{V}{\delta}$$

Introducing the value of δ from Eq. 12.13 in the above expression it becomes

$$\tau_0 = \text{constant}\sqrt{\frac{V^3\rho\mu}{x}} \qquad ...(12.15)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 12.15 can also be expressed as

$$\frac{\tau_0}{\frac{\rho V^2}{2}} = c_f = \frac{\text{constant}}{\sqrt{\frac{\rho V x}{\mu}}} = \frac{\text{constant}}{\sqrt{\text{Re}_x}} \qquad ...(12.16)$$

where the coefficient c_f is known as *local drag coefficient**. In Eq. 12.16 the value of the constant of proportionality has been obtained by Blasius by exact analytical solution of the boundary layer equations as 0.664. Thus Eq. 12.16 becomes

$$\frac{\tau_0}{\frac{\rho V^2}{2}} = \frac{0.664}{\sqrt{\text{Re}_x}} \qquad ...(12.17)$$

Further the total horizontal force F_D (or skin friction drag) acting on one side of the plate on which laminar boundary layer is developed can be obtained as

$$F_D = \int_0^L \tau_0 B dx = \left(0.664\ \rho V^{3/2} \upsilon^{1/2} L^{1/2} B\right)$$

in which B is the width of the plate and L is the length of the plate.

The average drag coefficient C_f may be obtained as

$$C_f = \frac{(F_D / BL)}{(\rho V^2 / 2)} = \frac{1.328}{\sqrt{Re_L}} \qquad ...(12.18)$$

in which $$Re_L = \frac{\rho V L}{\mu}$$

Further from the exact analytical solution of the boundary layer equations by Blasius the following expressions for the displacement thickness and the momentum thickness have been obtained.

$$\frac{\delta^*}{x} = \frac{1.729}{\sqrt{\text{Re}_x}} \qquad ...(12.19)$$

and

$$\frac{\theta}{x} = \frac{0.664}{\sqrt{\text{Re}_x}} \qquad ...(12.20)$$

The above noted equations for the laminar boundary layer along a flat plate can also be derived by using Eq. 12.12. For this it is essential to assume a suitable velocity distribution for the boundary layer. Thus as an example if it is assumed that the velocity distribution across a section of the boundary layer is linear with y upto the edge of the boundary layer, then we have

$$\frac{v}{V} = \frac{y}{\delta} \text{ for } y \le \delta$$

* For definitions of drag and drag coefficient see Chapter 18.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and the boundary conditions are at $y = 0$, $v = 0$; and $y = \delta$, $v = V$, where V is the velocity of the approaching stream outside the boundary layer.

Thus $$\tau_0 = \mu \left(\frac{\delta v}{\delta y}\right)_{y=0} = \frac{\mu V}{\delta}$$

$$\delta^* = \int_0^{\delta}\left(1-\frac{v}{V}\right)\delta y = \int_0^{\delta}\left(1-\frac{y}{\delta}\right)\delta y = \frac{\delta}{2}$$

and $$\theta = \int_0^{\delta}\frac{v}{V}\left(1-\frac{v}{V}\right)\delta y = \int_0^{\delta}\frac{y}{\delta}\left(1-\frac{y}{\delta}\right)\delta y = \frac{\delta}{6}$$

Substituting these values in Eq. 12.12 yields

$$\frac{1}{6}\frac{d\delta}{dx} = \frac{\mu}{\rho V\delta} = \frac{\upsilon}{V\delta}$$

or $$\frac{d}{dx}(\delta^2) = \frac{12\upsilon}{V}$$

Integrating the above expression, we get

$$\delta^2 = \frac{12\upsilon x}{V} + C$$

where C is constant of integration.

Since at $x = 0$, $\delta = 0$, hence $C = 0$. Thus

$$\delta^2 = \frac{12\upsilon x}{V}$$

or $$\frac{\delta^2}{x^2} = \frac{12\upsilon}{Vx} = \frac{12}{\mathrm{Re}_x}$$

$\therefore$ $$\frac{\delta}{x} = \sqrt{\frac{12}{\mathrm{Re}_x}} = \frac{3.464}{\sqrt{\mathrm{Re}_x}} \quad \ldots (12.21)$$

where $$\mathrm{Re}_x = \frac{Vx}{\upsilon}$$

Further local drag coefficient c_f may be obtained as

$$c_f = \frac{\tau_0}{(\rho V^2/2)} = \frac{2\mu}{\rho V\delta} = \frac{0.5774}{\sqrt{\mathrm{Re}_x}} \quad \ldots (12.22)$$

Skin friction drag F_D on one side of the plate having laminar boundary layer is obtained as

$$F_D = \int_0^L \tau_0 B dx = \left(0.5774\ \rho V^{3/2}\upsilon^{1/2}L^{1/2}B\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where B and L are the width and the length of the plate respectively.

The average drag coefficient C_f may be obtained as

$$C_f = \frac{(F_D / BL)}{(\rho V^2 / 2)} = \frac{1.1548}{\sqrt{Re_L}} \quad ...(12.23)$$

in which $$\mathrm{Re}_L = \frac{VL}{\upsilon}$$

The momentum thickness is given by

$$\frac{\theta}{x} = \frac{\delta}{6x} = \frac{0.5774}{\sqrt{\mathrm{Re}_x}} \quad ...(12.24)$$

and the displacement thickness is given by

$$\frac{\delta^*}{x} = \frac{\delta}{2x} = \frac{1.732}{\sqrt{\mathrm{Re}_x}} \quad ...(12.25)$$

It may thus be noted that the results obtained by assuming a linear velocity distribution in the boundary layer, are fairly close to the results obtained by Blasius by exact solution of the boundary layer equations. However, by assuming certain other velocity distributions in the boundary layer better agreement with the exact solution can be obtained. Moreover, in laminar boundary layer the flow exhibits the characteristics of laminar flow, and hence the velocity distribution at any section in the laminar boundary layer will follow parabolic law.

12.7 TURBULENT BOUNDARY LAYER

Turbulent boundary layers are usually thicker than laminar ones. Further as a result of intermingling of fluid particles between different layers of the fluid in a turbulent boundary layer the velocity distribution in a turbulent boundary layer is much more uniform than that in a laminar boundary layer. However, in a turbulent boundary layer, near the boundary large velocity change occurs in a relatively small vertical distance, and hence at the boundary the velocity gradient $(\partial v/\partial y)$ is steeper in a turbulent boundary layer than in a laminar boundary layer. The velocity distribution in a turbulent boundary layer follows a logarithmic law i.e., $v \sim \log y$, which can also be represented by a power law of the type

$$\frac{v}{V} = \left(\frac{y}{\delta}\right)^n \quad ... (12.26)$$

The value of the exponent n is approximately $(1/7)$ for moderate Reynolds number ($Vx/\upsilon < 10^7$ for a flat plate) and decreases somewhat with increasing Reynolds number. Although Eq. 12.26 satisfactorily describes the velocity distribution for most of the region of turbulent boundary layer, it cannot apply at the boundary itself because $(\partial v/\partial y) = (1/7)V\delta^{-1/7}\, y^{-6/7} = \infty$ when $y = 0$. However, immediately adjacent to the boundary there is laminar sublayer, which is so thin that its velocity distribution profile may be taken as linear (instead of parabolic) and tangential to the 'seventh root' profile at the point where the laminar sublayer merges with the turbulent part of the boundary layer.

The expressions for δ, c_f and C_f for the turbulent boundary layer are as noted below:

$$\frac{\delta}{x} = \frac{0.376}{(\mathrm{Re}_x)^{1/5}} \quad ... (12.27)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$c_f = \frac{0.059}{(\mathrm{Re}_x)^{1/5}} \quad \text{... (12.28)}$$

$$C_f = \frac{0.074}{(\mathrm{Re}_L)^{1/5}} \quad \text{... (12.29)}$$

Equation 12.29 is applicable only for values of Reynolds number Re_L ranging from 5×10^5 to 10^7. For values of Re_L ranging between 10^7 to 10^9, H. Schlichting assumed a logarithmic velocity distribution for the flow in the boundary layer and obtained the semi-empirical relation as noted below.

$$C_f = \frac{0.455}{(\log_{10} \mathrm{Re}_L)^{2.58}} \quad \text{...(12.30)}$$

A comparison of Eqs. 12.13 and 12.27 indicates that in the case of laminar boundary layer the boundary layer thickness δ increases as $x^{1/2}$, but in the case of turbulent boundary layer δ increases as $x^{4/5}$. Hence the thickness of the turbulent boundary layer increases much faster than that of laminar boundary layer.

Equation 12.18 for the average drag coefficient is applicable only if the boundary layer is laminar for the entire length of the plate, that is when the value of Re_L is less than 5×10^5. Similarly Eq. 12.29 (or 12.30) is applicable if, the boundary layer is turbulent for the entire length of the plate, and the value of Re_L is within the range of 5×10^5 to 10^7 (or 10^7 to 10^9). However, when the plate is of such a length that for some distance from the leading edge of the plate the boundary layer is laminar and it becomes turbulent for the remaining portion of the plate, then combined relations are required. Hence for such cases where the plate is covered with both laminar and turbulent boundary layers, Eq. 12.31 as proposed by Prandtl may be used to compute the average drag coefficient C_f

$$C_f = \frac{0.074}{\mathrm{Re}_L^{1/5}} - \frac{A}{\mathrm{Re}_L} \quad \text{... (12.31)}$$

The constant A in Eq. 12.31 depends on the value of the Reynolds number Re_x at which the laminar boundary layer becomes turbulent. The values of A for various values of critical Reynolds number are as given below:

$Critical\,(Re)_x$	3×10^5	5×10^5	10^6	3×10^6
$Constant A$	1050	1700	3300	8700

It may, however, be mentioned that since for most of the cases the value of critical Re_x may be taken as 5×10^5, Eq. 12.31 then becomes

$$C_f = \frac{0.074}{\mathrm{Re}_L^{1/5}} - \frac{1700}{\mathrm{Re}_L} \quad \text{... (12.31 a)}$$

Further Eq. 12.31 is applicable for values of Re_L upto 10^7. But for higher values of Re_L ranging from 10^7 to 10^9 the following equation can be used.

$$C_f = \frac{0.455}{(\log_{10} \mathrm{Re}_L)^{2.58}} - \frac{A}{\mathrm{Re}_L} \quad \text{... (12.32)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in which value of the constant A again depends on the critical Re_x and is the same as given in the above table. Equation 12.32 is known as *Prandtl–Schlichting Equation*. It may, however, be mentioned that Eq. 12.32 is valid in the entire range of Re_L from 5×10^5 upto 10^9, and it agrees with the Eq. 12.31 upto $Re_L = 10^7$.

12.8 LAMINAR SUBLAYER

If the plate is very smooth, even in the zone of turbulent boundary layer, there exists a very thin layer immediately adjacent to the boundary, in which the flow is laminar. This thin layer is commonly known as *laminar sublayer*, and its thickness is represented by δ'. As stated earlier the exponential as well as the logarithmic velocity distribution cannot apply at the boundary itself, although these satisfactorily describe the velocity distribution for most of the region of turbulent boundary layer. This discrepancy may, however, be explained by the existence of the laminar sublayer. Since the flow in the laminar sublayer is laminar, it will follow a parabolic velocity distribution law, which can be approximately taken to be linear as δ' is very small, and hence exponential or logarithmic velocity distribution law cannot be applied within the laminar sublayer. Thus in this case there will be a gradual change from parabolic velocity distribution very close to the boundary to exponential or logarithmic velocity distribution and the intersection of the parabolic and exponential or logarithmic velocity distribution curves may be arbitrarily taken as the limit of the laminar sublayer. Nikuradse's experimental studies have shown that

$$\delta' = \frac{11.6\upsilon}{\sqrt{(\tau_0/\rho)}} = \frac{11.6\upsilon}{V_*} \qquad ...(12.33)$$

in which $V^* = \sqrt{(\tau_0/\rho)}$ is known as shear or friction velocity. As indicated in Chapter 14 the relative magnitudes of the thickness of laminar sublayer δ' and the average height of roughness projections k facilitates the classification of the boundaries as hydrodynamically smooth and rough.

12.9 BOUNDARY LAYER ON ROUGH SURFACES

In the preceding sections the development of boundary layer along smooth plates has been considered. However, in most practical applications connected with the boundary layer development on flat plates such as ships, airplane wings, turbine blades etc., the surfaces cannot be considered smooth. As such the flow past a rough plate is of much practical interest. For a rough plate if k is the average height of roughness projections on the surface of the plate and δ is the thickness of the boundary layer, then the relative roughness (k/δ) is a significant parameter indicating the behaviour of the boundary surface. For k remaining constant, (k/δ) decreases along the plate because δ increases in the downstream direction. As a result the front portion of the plate will behave differently from its rear portion as far as the influence of roughness on drag is concerned. If it is assumed that the boundary layer is turbulent from the leading edge of the plate, the front portion of the plate will act as hydrodynamically rough, followed by a transition region and the downstream portion of the plate will be hydrodynamically smooth if the plate is sufficiently long. The limits between these three regimes are determined by the value of a dimensionless roughness parameter $(V_* k_s/\upsilon)$ as indicated below:

Hydrodynamically smooth : $\frac{V_* k_s}{\upsilon} < 5$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Transition : $5 < \frac{V_* k_s}{\upsilon} < 70$

Hydrodynamically rough : $\frac{V_* k_s}{\upsilon} > 70$

(or completely rough)

in which k_s is equivalent sand grain roughness defined as that value of the roughness which would offer the same resistance to the flow past the plate as that due to the actual roughness on the surface of the plate. Although the above noted limits have been determined for flow through pipes by Nikuradse on the basis of the results of his experiments on the pipes made artificially rough by gluing sand grains of uniform size on the inner surface of the pipe, but the same may also be applied to the flow past a plate. The introduction of the equivalent sand grain roughness k_s instead of the average height of the roughness projections k is due to the fact that as compared to k, k_s is a more representative parameter of the actual roughness. This is so because usually the actual roughness projections on the surface of the plate are of various heights and they do not have any uniform pattern and hence such roughnesses cannot be truely represented in terms of the average height of the roughness projections. Moreover, the consideration of k_s also facilitates the use of Nikuradse's results, which are in terms of the sand grain roughness, in the analysis of flow past rough plates.

In the completely rough regime the local drag coefficient c_f and the average drag coefficient C_f are given by the following expressions

$$c_f = \left[2.87 + 1.58 \log_{10}\left(\frac{x}{k_s}\right)\right]^{-2.5} \quad \text{... (12.34)}$$

$$C_f = \left[1.89 + 1.62 \log_{10}\left(\frac{L}{k_s}\right)\right]^{-2.5} \quad \text{... (12.35)}$$

which are valid for $10^2 < \left(\frac{L}{k_s}\right) < 10^6$.

12.10 SEPARATION OF BOUNDARY LAYER

As mentioned in Section 12.3 the boundary layer thickness is considerably affected by the pressure gradient in the direction of flow. If the pressure gradient ($\partial p/\partial x$) is zero, then the boundary layer continues to grow in thickness along a flat plate. With the decreasing pressure in the direction of flow i.e., with negative pressure gradient, the boundary layer tends to be reduced in thickness. However, with the pressure increasing in the direction of flow i.e., with positive (or adverse) pressure gradient, the boundary layer thickens rapidly. The adverse pressure gradient plus the boundary shear decreases the momentum in the boundary layer and if they both act over a sufficient distance they cause the fluid in the boundary layer to come to rest i.e., the retarded fluid particles, cannot, in general penetrate too far into the region of increased pressure owing to their small kinetic energy. Thus, the boundary layer is deflected sideways from the boundary, separates from it and moves into the main stream. This phenomenon is called *separation*.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

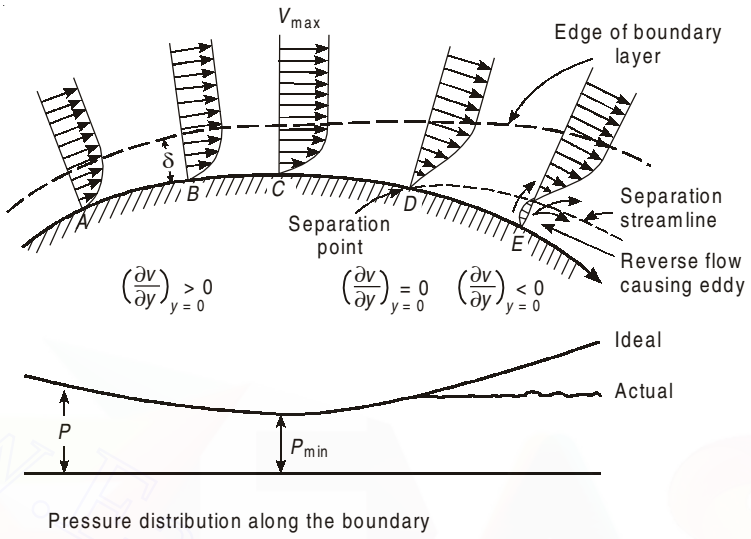


Figure 12.5 *Separation of boundary layer*

Consider flow of fluid over a curved surface as shown in Fig. 12.5. As the fluid flows round the surface it is accelerated over the left hand section until at point *C* the velocity just outside the boundary layer is a maximum. At this section the pressure is minimum as shown by the graph below the surface. Thus, from *A* to *C* the pressure gradient $(\partial p/\partial x)$ is negative and the net pressure force on an element of fluid in the boundary layer is in the direction of flow, which counteracts to some extent the slowing down effect of the boundary on the flowing fluid and thus the rate at which the boundary layer thickens is less than that for a flat plate with zero pressure gradient (at a corresponding value of Re_x).

However, beyond *C* the pressure increases, and hence the net pressure force on an element of fluid in the boundary layer opposes the forward flow. Thus at a certain distance on the downstream of point *C* the fluid near the boundary surface is soon brought to a standstill. The value of the velocity gradient $(\partial v/\partial y)$ at the boundary surface is then zero as at point *D*. The fluid is no longer able to follow the contour of the curved surface and it separates from it. The separation of the flowing fluid from the boundary first occurs at a point where $(\partial v/\partial y)$ at the boundary becomes zero and this point is known as *separation point*. On downstream of the separation point *D*, a further retardation of the fluid close to the boundary can even have a reverse or back flow as at point *E* shown in Fig. 12.5. If all the points below which a reverse flow occurs are joined by a smooth curve, a line dividing the forward and reverse flows is obtained which is known as *separation streamline*. In a region in between the boundary surface and the separation streamline, as a result of the reverse flow, large irregular eddies are formed in which much energy is dissipated as heat. This region of disturbed fluid usually extends for some distance on the downstream. Since the energy of the eddies is dissipated as heat, the pressure downstream remains approximately the same as at the separation point.

Separation occurs with both laminar and turbulent boundary layers, but laminar boundary layer is more susceptible to earlier separation than turbulent boundary layer. This is so because in a laminar boundary layer the increase of velocity with distance from the boundary surface is less rapid, and the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

adverse pressure gradient can more rapidly halt the slow moving fluid close to the boundary surface. On the other hand in a turbulent boundary layer the velocity distribution is much more uniform than in a laminar boundary layer because of intense lateral mixing. As a result relatively higher velocity prevails within a turbulent boundary layer, which reduces tendency of separation.

Separation of the boundary layer greatly affects the flow as a whole. In particular the formation of a wake zone of disturbed fluid on the downstream, in which the pressure is approximately constant and much less than that on the upstream, gives rise to boundary forces. As indicated in Chapter 18, the net force in the direction of flow caused by such pressure difference is known as pressure drag or form drag, the magnitude of which depends on the form or shape of the boundary and the resulting flow pattern obtained due to separation. Further the flow in a divergent passage or diffuser is another example in which the separation of the flow may be caused due to adverse pressure gradient prevailing there unless the angle of divergence is very small.

Since the separation of the boundary layer gives rise to additional resistance to flow, attempts should be made to avoid separation by some means. One of these methods may be by developing such boundary shapes for which the separation is as far as possible eliminated. The flow past a streamlined object such as airfoil is one such example. Apart from this the separation may also be avoided by adopting suitable method of controlling the boundary layer which are described in the next section.

12.11 METHODS OF CONTROLLING THE BOUNDARY LAYER

Several methods of controlling the boundary layer have been developed which are briefly described below.

12.11.1 Motion of Solid Boundary

The formation of the boundary layer is due to the difference between the velocity of the flowing fluid and that of the solid boundary. As such it is possible to eliminate the formation of a boundary layer by causing the solid boundary to move with the flowing fluid. Such a motion of the boundary may be achieved in the simplest ways by rotating a circular cylinder lying in a stream of fluid [see Fig. 12.6 (a)], so that on the upper side of the cylinder, where the fluid as well as the cylinder move in the same direction, the boundary layer does not form and hence the separation is completely eliminated. However, on the lower side of the cylinder, where the fluid motion is opposite to that of the cylinder, separation would occur.

12.11.2 Acceleration of the Fluid in the Boundary Layer

This method consists of supplying additional energy to the particles of fluid which are being retarded in the boundary layer. This may be achieved either by injecting fluid into the region of boundary layer from the interior of the body with the help of some suitable device as shown in Fig. 12.6 (b); or by diverting a portion of the fluid of the main stream from the region of high pressure to the retarded region of boundary layer through a slot provided in the body as in the case of the slotted wing shown in Fig. 12.6 (c). However, a disadvantage of this method is that if the fluid is injected into a laminar boundary layer, it undergoes a transition to turbulent boundary layer which results in an increased skin friction drag.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

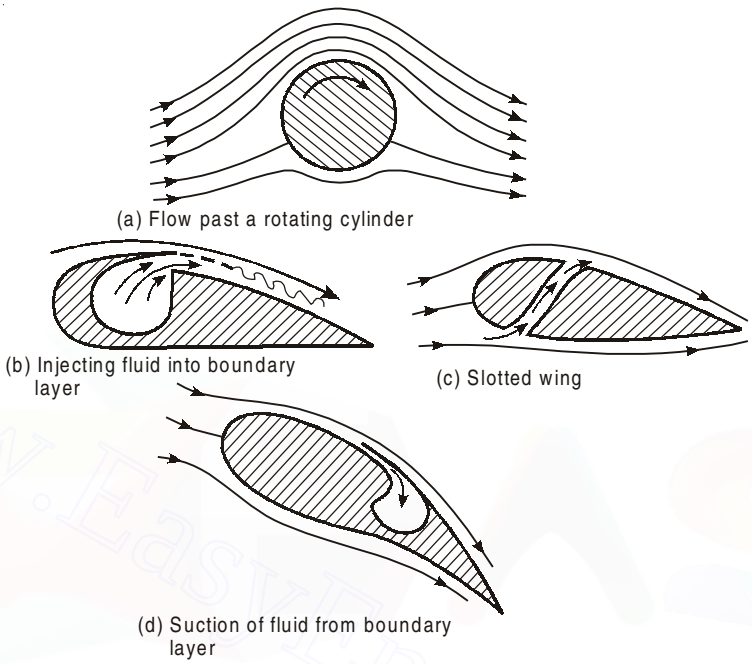


Figure 12.6 *Methods of controlling the boundary layer*

12.11.3 Suction of the Fluid from the Boundary Layer

In this method the slow moving fluid in the boundary layer is removed by suction through slots or through a porous surface as shown in Fig. 12.6 (d), so that on the downstream of the point of suction a new boundary layer starts developing which is able to withstand an adverse pressure gradient and hence separation is prevented. Moreover, the suction of the fluid from the boundary layer also greatly delays its transition from laminar to turbulent due to which skin friction drag is reduced.

12.11.4 Streamlining of Body Shapes

By the use of suitably shaped bodies the point of transition of the boundary layer from laminar to turbulent can be moved downstream which results in the reduction of the skin friction drag. Furthermore by streamlining of body shapes the separation may be eliminated.

ILLUSTRATIVE EXAMPLES

Example 12.1. *The velocity distribution in the boundary layer is given as*

$$\frac{v}{V} = \frac{3}{2}\eta - \frac{1}{2}\eta^2$$

in which $\eta = (y/\delta)$. *Compute* (δ^*/δ) and (θ/δ).

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution $$\delta^* = \int_0^{\infty}\left(1-\frac{v}{V}\right)dy$$

or $$\delta^* = \int_0^{\delta}\left(1-\frac{v}{V}\right)dy + \int_{\delta}^{\infty}\left(1-\frac{v}{V}\right)dy$$

But outside the boundary layer $(v/V) = 1$, and hence

$$\delta^* = \int_0^{\delta}\left(1-\frac{v}{V}\right)dy$$

$$\frac{v}{V} = \frac{3}{2}\eta - \frac{1}{2}\eta^2 \text{ and } dy = \delta d\eta$$

Thus $$\delta^* = \delta\int_0^1\left(1-\frac{3}{2}\eta+\frac{1}{2}\eta^2\right)d\eta = \frac{5}{12}\delta$$

$\therefore$ $$\frac{\delta^*}{\delta} = \frac{5}{12}$$

Similarly $$\theta = \int_0^{\delta}\frac{v}{V}\left(1-\frac{v}{V}\right)dy$$

or $$\theta = \delta\int_0^1\left(\frac{3}{2}\eta-\frac{1}{2}\eta^2\right)\left(1-\frac{3}{2}\eta+\frac{1}{2}\eta^2\right)d\eta$$

$$= \frac{19}{120}\delta$$

$\therefore$ $$\frac{\theta}{\delta} = \frac{19}{120}$$

Example 12.2. *Given that a laminar boundary layer at zero pressure gradient over a flat plate is described by the velocity profile*

$$\frac{v}{V_o} = \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)$$

determine the momentum correction coefficient (or factor) and the energy correction coefficient (or factor).

Also show that boundary layer thickness δ, wall shear stress τ_0 and coefficient of drag C_D are given by

$$\delta = \frac{4.795x}{\sqrt{Re_x}};\ \tau_0 = \frac{0.328\rho V_0^2}{\sqrt{Re_x}};\ C_D = \frac{1.312}{\sqrt{Re_L}}$$

where symbols have their usual meaning.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Momentum correction coefficient (or factor) is given by

$$\beta = \frac{1}{AV^2}\int_A v^2 dA$$

Mean velocity $$V = \frac{Q}{A} = \frac{\int_A v dA}{A}$$

Considering unit width of the plate $dA = 1 \times dy$, and $A = (1 \times \delta)$. Thus

$$V = \frac{\int_0^{\delta} V_0 \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)(1\times dy)}{(1\times\delta)}$$

or $$V = \frac{V_0}{\delta}\int_0^{\delta} \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right) dy$$

or $$V = \frac{2V_0}{\pi}$$

Thus $$\beta = \frac{\pi^2}{4V_0^2} \times \frac{1}{(1\times\delta)}\int_0^{\delta}\left[V_0 \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]^2 (1\times dy)$$

or $$\beta = \frac{\pi^2}{4\delta}\int_0^{\delta} \sin^2\left(\frac{\pi}{2}\frac{y}{\delta}\right) dy$$

or $$\beta = \frac{\pi^2}{8} = 1.234$$

Energy correction coefficient (or factor) is given as

$$\alpha = \frac{1}{AV^3}\int_A v^3 dA$$

or $$\alpha = \frac{\pi^3}{8V_0^3} \times \frac{1}{(1\times\delta)}\int_0^{\delta}\left[V_0 \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]^3 (1\times dy)$$

or $$\alpha = \frac{\pi^3}{8\delta}\int_0^{\delta} \sin^3\left(\frac{\pi}{2}\frac{y}{\delta}\right) dy$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\alpha = \frac{\pi^2}{6} = 1.645$$

From Eq. 12.12 we have

$$\frac{\tau_0}{\rho V_0^2} = \frac{d\theta}{dx} \qquad ...(i)$$

At the boundary

$$\tau_0 = \mu \left(\frac{\delta v}{\delta y}\right)_{y=0}$$

$$\left(\frac{\delta v}{\delta y}\right)_{y=0} = \frac{\pi}{2}\frac{V_0}{\delta}\left[\cos\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]_{y=0} = \frac{\pi}{2}\frac{V_0}{\delta}$$

$$\tau_0 = \mu \frac{\pi}{2}\frac{V_0}{\delta} \qquad ...(ii)$$

From Eq. 12.2

$$\theta = \int_0^{\infty} \frac{v}{V_0}\left(1 - \frac{v}{V}\right) dy$$

Outside the boundary layer since $v = V_0$,

$$\theta = \int_0^{\delta} \frac{v}{V_0}\left(1 - \frac{v}{V_0}\right) dy$$

or $$\theta = \int_0^{\delta} \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\left[1 - \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right] dy$$

or $$\theta = \frac{2\delta}{\pi}\left(1 - \frac{\pi}{4}\right)$$

By substituing the values of τ_0 and θ in Eq. (i) we get

$$\frac{1}{\rho V_0^2}\left(\mu \frac{\pi}{2}\frac{V_o}{\delta}\right) = \frac{2}{\pi}\left(1 - \frac{\pi}{4}\right)\frac{d\delta}{dx}$$

or $$\delta\, d\delta = \frac{11.4976\, \mu dx}{\rho V_0} \qquad ...(iii)$$

Since δ is a function of x only, integration of Eq. (iii) gives

$$\frac{\delta^2}{2} = \frac{11.4976\, \mu dx}{\rho V_0} + \text{const.}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\delta = 4.795\sqrt{\frac{\mu x}{\rho V_0}}$$

or $$\delta = \frac{4.795x}{\sqrt{\text{Re}_x}}$$

where $$\text{Re}_x = \frac{\rho V_0 x}{\mu}$$

Substituing the value of δ in Eq. (ii) yields

$$\tau_0 = \mu \frac{\pi}{2} \frac{V_0}{4.795} \sqrt{\frac{\rho V_0}{\mu x}}$$

or $$\tau_0 = 0.328 \sqrt{\frac{\rho \mu V_0^3}{x}} \quad \ldots \text{(iv)}$$

or $$\tau_0 = \frac{0.328 \rho V_0^2}{\sqrt{\text{Re}_x}}$$

where $$\text{Re}_x = \left(\frac{\rho V_0 x}{\mu}\right)$$

The drag F_D on one side of the plate of unit width is

$$F_D = \int_0^L \tau_0 dx$$

Introducing the value of τ_o from Eq. (iv) we have

$$F_D = \int_0^L 0.328 \sqrt{\frac{\rho \mu V_0^3}{x}}\, dx$$

or $$F_D = 0.656 \sqrt{\rho \mu V_0^3 L} \quad \ldots \text{(v)}$$

The drag can be expressed in terms of a drag coefficient C_D times the stagnation pressure ($\rho V_0^2/2$) and the area of plate L (per unit width). Thus

$$F_D = C_D \frac{\rho V_0^2}{2} L$$

Introducing the value of F_D from Eq. (v) we have

$$0.656 \sqrt{\rho \mu V_0^3 L} = C_D \frac{\rho V_0^2}{2} L$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$C_D = \frac{1.312}{\sqrt{\text{Re}_L}}$$

where $$\text{Re}_L = \frac{\rho V_0 L}{\mu}$$

Example 12.3. *Air flows over a flat plate 1 m long at a velocity of 6 m/s. Determine (a) the boundary layer thickness at the end of the plate, (b) shear stress at the middle of the plate, (c) total drag per unit length on the sides of the plate. Take* $\rho = 1.226\ kg/m^3$ *(0.125 msl/m³) and* $\upsilon = 0.15 \times 10^{-4}\ m^2/s$ *(0.15 stokes) for air.*

Solution

$$\text{Re}_L = \frac{VL}{\upsilon} = \frac{6 \times 1}{0.15 \times 10^{-4}} = 4.0 \times 10^5$$

Hence the boundary layer is laminar over the entire length of the plate

$$\therefore \quad \delta = 5\sqrt{\frac{x\upsilon}{V}}$$

$$= 5\sqrt{\frac{1 \times 0.15 \times 10^{-4}}{6}} = 7.91 \times 10^{-3}\ \text{m} = 7.91\ \text{mm}$$

SI units

$$\tau_0 = c_f \frac{\rho V^2}{2}\ ; \text{and } c_f = \frac{0.664}{\sqrt{\text{Re}_x}}$$

For the middle of the plate $x = 0.5$ m

Thus $$\text{Re}_x = \frac{6 \times 0.5}{0.15 \times 10^{-4}} = 2.0 \times 10^5$$

and $$c_f = \frac{0.664}{\sqrt{2.0 \times 10^5}} = 1.485 \times 10^{-3}$$

$$\therefore \quad \tau_0 = \frac{1.485 \times 10^{-3} \times 1.226 \times (6)^2}{2} = 32.77 \times 10^{-3}\ \text{N/m}^2$$

Further $$C_f = \frac{1.328}{\sqrt{\text{Re}_L}} = \frac{1.328}{\sqrt{4.0 \times 10^5}} = 2.1 \times 10^{-3}$$

and $$F_D = 2 \times BL \times C_f \frac{\rho V^2}{2}$$

$$\therefore \quad F_D = 2 \times (1 \times 1) \times 2.1 \times 10^{-3} \times \frac{1.226 \times (6)^2}{2}$$

$$= 92.69 \times 10^{-3}\ \text{N}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Metric units

$$\tau_0 = c_f \frac{\rho V^2}{2} \text{ ; and } c_f = \frac{0.664}{\sqrt{\text{Re}_x}}$$

For the middle of the plate $x = 0.5$ m

Thus $$\text{Re}_x = \frac{6 \times 0.5}{0.15 \times 10^{-4}} = 2.0 \times 10^5$$

and $$c_f = \frac{0.664}{\sqrt{2.0 \times 10^5}} = 1.485 \times 10^{-3}$$

$$\therefore \quad \tau_0 = \frac{1.485 \times 10^{-3} \times 0.125 \times (6)^2}{2}$$

$$= 3.34 \times 10^{-3} \text{ kg(f)/m}^2$$

Further $$C_f = \frac{1.328}{\sqrt{Re_L}} = \frac{1.328}{\sqrt{4.0 \times 10^5}} = 2.1 \times 10^{-3}$$

and $$F_D = 2 \times BL \times C_f \frac{\rho V^2}{2}$$

$$\therefore \quad F_D = 2 \times (1 \times 1) \times 2.1 \times 10^{-3} \times \frac{0.125 \times (6)^2}{2}$$

$$= 9.45 \times 10^{-3} \text{ kg (f)}$$

Example 12.4. *A smooth two-dimensional flat plate is exposed to a wind velocity of 100 km per hour. If laminar boundary layer exists upto a value of Re_x equal to 3×10^5, find the maximum distance upto which laminar boundary layer persists, and find its maximum thickness. Assume kinematic viscosity of air as 1.49×10^{-5} m^2/s.*

Solution

$$V = \frac{100 \times 10^3}{3600} = 27.78 \text{ m/s}$$

$$\text{Re}_x = \frac{Vx}{\upsilon}$$

or $$3 \times 10^5 = \frac{27.78 \times x}{1.49 \times 10^{-5}}$$

$$\therefore \quad x = 0.161 \text{ m}$$

$$\delta = 5\sqrt{\frac{x\upsilon}{V}}$$

$$= 5\sqrt{\frac{0.161 \times 1.49 \times 10^{-5}}{27.78}}$$

$$= 1.47 \times 10^{-3} \text{ m} = 1.47 \text{ mm}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 12.5. *Find the power required to tow lengthwise a plate 1.2 m wide and 3 m long at a velocity of 2.4 m/s in water at 23°C. Make allowance for the fact that the boundary layer will change from laminar to turbulent over the plate. υ for water at 23°C is 0.9×10^{-6} m^2/s and $\rho = 1000$ kg/m^3 (10^2 msl/m^3).*

Solution

$$\text{Re}_L = \frac{VL}{\upsilon} = \frac{2.4 \times 3}{0.9 \times 10^{-6}} = 8.0 \times 10^6$$

$$C_f = \frac{0.074}{(\text{Re}_L)^{1/5}} - \frac{1700}{\text{Re}_L}$$

or

$$C_f = \left[\frac{0.074}{(8.0 \times 10^6)^{1/5}} - \frac{1700}{8.0 \times 10^6}\right] = 2.87 \times 10^{-3}$$

The drag on the plate is given by

$$F_D = 2C_f \times BL \times \frac{\rho V^2}{2}$$

SI units

$$F_D = \left[2 \times 2.87 \times 10^{-3} \times 1.2 \times 3 \times \frac{1000 \times (2.4)^2}{2}\right]$$

$$= 59.51 \text{ N}$$

$\therefore$ Power required $= 59.51 \times 2.4$

$= 143 \text{ W} = 0.143 \text{ kW}$

Metric units

$$F_D = \left[2 \times 2.87 \times 10^{-3} \times 1.2 \times 3 \times \frac{102 \times (2.4)^2}{2}\right]$$

$$= 6.07 \text{ kg(f)}$$

$\therefore$ Power required $= \dfrac{6.07 \times 2.4}{75} = 0.194$ h.p.

Example 12.6. *Find the ratio of skin friction drag on the front two-third and rear one-third of a flat plate kept in a uniform stream at zero incidence. Assume the boundary layer to be turbulent over the entire plate.*

Solution

$$C_f = \frac{0.074}{(\text{Re}_L)^{1/5}}$$

For the front two-third portion of the plate

$$Re_L = \frac{V(2/3)L}{\upsilon}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $$C_f = \frac{0.074}{\left[\frac{V(2/3)L}{\upsilon}\right]^{1/5}}$$

∴ Drag for the front two-third portion of the plate is

$$F_{D1} = C_f \times BL \times \frac{\rho V^2}{2}$$

$$= \frac{0.074}{\left[\frac{V(2/3)L}{\upsilon}\right]^{1/5}} \times B \times (2/3)\, L \times \frac{\rho V^2}{2}$$

Similarly drag for the entire plate is

$$F_D = \frac{0.074}{\left[\frac{VL}{\upsilon}\right]^{1/5}} \times B \times L \times \frac{\rho V^2}{2}$$

∴ Drag for the rear one-third portion of the plate is

$$F_{D2} = (F_D - F_{D1})$$

$$= \frac{0.074}{\left[\frac{VL}{\upsilon}\right]^{1/5}} \times B \times L \times \frac{\rho V^2}{2} \times \left[1 - \frac{2}{3}\left(\frac{3}{2}\right)^{1/5}\right]$$

∴ $$= \frac{F_{D_1}}{F_{D_2}} = \frac{\frac{2}{3}\left(\frac{3}{2}\right)^{1/5}}{\left[1 - \frac{2}{3}\left(\frac{3}{2}\right)^{1/5}\right]} = 2.61$$

Example 12.7. *Calculate the friction drag on a plate 0.15 m wide and 0.45 m long placed longitudinally in a stream of oil flowing with a free stream velocity of 6 m/s. Also find the thickness of the boundary layer and shear stress at the trailing edge. Sp. gr. of oil is 0.925 and its kinematic viscosity is 0.9 × 10^{-4} m^2/s (0.9 stokes).*

Solution

$$\mathrm{Re}_L = \frac{6 \times 0.45}{0.9 \times 10^{-4}} = 3.0 \times 10^4$$

Hence the boundary layer is laminar over the entire length of the plate.

Thus $$C_f = \frac{1.328}{\sqrt{\mathrm{Re}_L}} = \frac{1.328}{\sqrt{3.0 \times 10^4}} = 7.667 \times 10^{-3}$$

SI units

The friction drag for both sides of the plate is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F_D = 2 \times BL \times C_f \frac{\rho V^2}{2}$$

$$= 2 \times (0.15 \times 0.45) \times 7.667 \times 10^{-3} \times (0.925 \times 1000) \times \frac{(6)^2}{2}$$

$$= 17.233 \text{ N}$$

Thickness of the boundary layer at the trailing edge is given by Eq. 12.14 for $x = L$ as

$$\delta = 5\sqrt{\frac{L\upsilon}{V}}$$

$$= 5\sqrt{\frac{0.45 \times 0.9 \times 10^{-4}}{6}} = 13 \times 10^{-3} \text{ m} = 13 \text{ mm}$$

Shear stress at the trailing edge is given by Eq. 12.17 for $x = L$ as

$$\tau_0 = \frac{0.664 \times \rho V^2}{2\sqrt{\text{Re}_L}}$$

$$= \frac{0.664 \times (0.925 \times 1000) \times (6)^2}{2 \times \sqrt{3.0 \times 10^4}} = 63.83 \text{ N/m}^2$$

Metric units

The friction drag for both sides of the plate is

$$F_D = 2 \times BL \times C_f \frac{\rho V^2}{2}$$

$$= 2 \times (0.15 \times 0.45) \times 7.667 \times 10^{-3} \times (0.925 \times 102) \times \frac{(6)^2}{2}$$

$$= 1.758 \text{ kg(f)}$$

Thickness of the boundary layer at the trailing edge is given by Eq. 12.14 for $x = L$ as

$$\delta = 5\sqrt{\frac{L\upsilon}{V}}$$

$$= 5\sqrt{\frac{0.45 \times 0.9 \times 10^{-4}}{6}} = 1.3 \times 10^{-2} \text{ m} = 1.3 \text{ cm}$$

Shear stress at the trailing edge is given by Eq. 12.17 for $x = L$, as

$$\tau_0 = \frac{0.664 \times \rho V^2}{2\sqrt{\text{Re}_L}}$$

$$= \frac{0.664 \times (0.925 \times 102) \times (6)^2}{2 \times \sqrt{3.0 \times 10^4}} = 6.51 \text{ kg(f)/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 12.8. *Assuming that the velocity distribution in the boundary layer is given by*

$$\frac{v}{V} = \left(\frac{y}{\delta}\right)^{1/7}$$

calculate $\frac{\delta^*}{\delta}$, $\frac{\theta}{\delta}$ *and* $\frac{\delta_E}{\delta}$. *If at a certain section, free stream velocity V was observed to be 10 m/s and the thickness of the boundary layer as 25 mm, then calculate the energy loss per unit length due to the formation of the boundary layer. Take* $\rho = 1.226\ kg/m^3$ $(0.125\ msl/m^3)$.

Solution

The displacement thickness is given by Eq. 12.1 as

$$\delta^* = \int_0^\delta \left(1 - \frac{v}{V}\right) dy$$

$$= \int_0^\delta \left[1 - \left(\frac{y}{\delta}\right)^{1/7}\right] dy = \frac{\delta}{8}$$

$$\therefore \quad \frac{\delta^*}{\delta} = \frac{1}{8} = 0.125$$

The momentum thickness is given by Eq. 12.2 as

$$\theta = \int_0^\delta \frac{v}{V}\left(1 - \frac{v}{V}\right) dy$$

$$= \int_0^\delta \left(\frac{y}{\delta}\right)^{1/7} \left[1 - \left(\frac{y}{\delta}\right)^{1/7}\right] dy = \frac{7}{72}\delta$$

$$\therefore \quad \frac{\theta}{\delta} = \frac{7}{72} = 0.097$$

The energy thickness is given by Eq. 12.3 as

$$\delta_E = \int_0^\delta \frac{v}{V}\left(1 - \frac{v^2}{V^2}\right) dy$$

$$= \int_0^\delta \left(\frac{y}{\delta}\right)^{1/7} \left[1 - \left(\frac{y}{\delta}\right)^{2/7}\right] dy = \frac{7}{40}\delta$$

$$\therefore \quad \frac{\delta_E}{\delta} = \frac{7}{40} = 0.175$$

The loss of energy per unit length due to the formation of the boundary layer is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$E_L = \frac{1}{2}\rho V^3 \delta_E$$

SI units

$$E_L = \frac{1}{2} \times 1.226 \times (10)^3 \times \frac{7}{40} \times 25 \times 10^{-3}$$
$$= 2.682 \text{ N.m/s}$$

Metric units

$$E_L = \frac{1}{2} \times 0.125 \times (10)^3 \times \frac{7}{40} \times 25 \times 10^{-3}$$
$$= 0.273 \text{ kg(f)-m/s}$$

Example 12.9. *It is required to determine the frictional drag of a submarine. The length of the hull is 75 m and its surface area is 3000 m^2. The submarine is travelling at a constant speed of 5 m/s. Critical Reynolds number at which the flow in the boundary layer changes from laminar to turbulent is 5×10^5. Assuming that the boundary layer at the leading edge is laminar, obtain the frictional drag and the power required to propel the submarine at 5 m/s. Take $\upsilon = 1 \times 10^{-6}$ m^2/s (0.01 cm^2/s) and $\rho = 1000$ kg/m^3 (102 msl/m^3).*

Solution

$$\text{Re}_L = \frac{VL}{\upsilon} = \frac{5 \times 75}{1 \times 10^{-6}} = 3.75 \times 10^8$$

Since at leading edge boundary layer is laminar, it changes from laminar to turbulent on the surface of the submarine.

As critical Reynolds number = 5×10^5

$$C_f = \frac{0.074}{(\text{Re}_L)^{1/5}} - \frac{1700}{\text{Re}_L}$$

$$= \left[\frac{0.074}{(3.75 \times 10^8)^{1/5}} - \frac{1700}{3.75 \times 10^8}\right]$$

$$= 1.422 \times 10^{-3}$$

SI units

The frictional drag is

$$F_D = C_f \rho A \frac{V^2}{2}$$

$$= 1.422 \times 10^{-3} \times 1000 \times 3000 \times \frac{(5)^2}{2}$$

$$= 53.325 \times 10^3 \text{ N} = 53.325 \text{ kN}$$

Power required $= (53.325 \times 10^3 \times 5)$

$$= 266.625 \times 10^3 \text{ W} = 266.625 \text{ kW}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Metric units

The frictional drag is

$$F_D = C_f \rho A \frac{V^2}{2}$$

$$= 1.422 \times 10^{-3} \times 102 \times 3000 \times \frac{(5)^2}{2}$$

$$= 5.44 \times 10^3 \text{ kg (f)}$$

Power required $= \dfrac{5.44 \times 10^3 \times 5}{75} = 362.7$ h.p.

SUMMARY OF MAIN POINTS

1. When a real fluid flows past a soild boundry, a layer of fluid adheres to the boundary surface due to viscosity of the fluid, and a small region is developed in the vicinity of the boundary surface in which the velocity of flowing fluid increases gradually from zero at the boundary surface to the velocity of the main stream or free stream velocity. This region is known as *boundry layer*, in which there exists a large velocity gradient ($\partial v / \partial y$) normal to the boundary surface.
2. The nominal thickness of the boundary layer δ is considered equal to the distance y from the boundary surface at which the velocity v reaches 99% of the free stream velocity V , i.e., $v = 0.99$ V.
3. The *displacement thickness* δ^* is defined as the distance by which the boundary surface would have to be displaced outwards so that the total actual discharge would be same as that of an ideal fluid past the displaced boundary. It may be expressed by the equation

$$\delta^* = \int_0^\infty \left(1 - \frac{v}{V}\right) dy$$

4. The *momentum thickness* θ is defined as the distance from the actual boundary surface such that the momentum flux corresponding to the main stream velocity V through this distance θ is equal to the deficiency or loss of momentum due to the boundary layer formation. It may be mathematically express by the equation

$$\theta = \int_0^\infty \frac{v}{V}\left(1 - \frac{v}{V}\right) dy$$

5. The *energy thickness* δ_E is defined as the distance from the actual boundary surface such that the energy flux corresponding to the main stream velocity V through this distance δ_E is equal to the deficiency or loss of energy due to the boundary layer formation. It may be mathematically expressed by the equation

$$\delta_E = \int_0^\infty \frac{v}{V}\left(1 - \frac{v^2}{V^2}\right) dy$$

6. Von-Karman momentum integral equation of the boundary layer is given as

$$\frac{\tau_0}{\rho V^2} = \frac{d\theta}{dx}$$

where

τ_0 = shear stress at the boundary surface.

7. In a laminar boundary layer the flow exhibits all the characteristics of laminar flow. The velocity distribution in a laminar boundary layer is parabolic. The laminar boundary layer exists when the Reynolds number $\text{Re}_x = \dfrac{Vx}{\nu}$ is less than 5×10^5,

where

V = Free stream velocity of flowing fluid ;

x = distance from the leading edge of the solid boundary ; and

ν = kinematic viscosity of the flowing fluid.

The thickness of laminar boundary layer is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\delta = \frac{5x}{\sqrt{\text{Re}_x}}$$

8. From the exact analytical solution of the boundary layer equations by Blasius the following expressions for the displacement thickness and the momentum thickness have been obtained.

$$\delta^* = \frac{1.729x}{\sqrt{\text{Re}_x}}$$

$$\theta = \frac{0.664x}{\sqrt{\text{Re}_x}}$$

9. The average drag coefficient C_f obtained by Blasius by exact solution of the boundary layer equations is given by

$$C_f = \frac{1.328}{\sqrt{\text{Re}_L}} = \frac{1.328}{\sqrt{VL/\nu}}$$

where

L = length of the plate.

Note : The expressions for δ, δ^*, q and C_f have been obtained by considering a laminar boundary layer formed along a long thin plate held stationary in a uniform stream in the direction parallel to the flow.

10. In turbulent boundary layer the flow exhibits the characteristics of turbulent flow. The velocity distribution in a turbulent boundary layer follows a lagarithmic law i.e., $v \sim \log y$, which can also be represented by a power law of the type

$$\frac{v}{V} = \left(\frac{y}{\delta}\right)^n$$

The value of the exponent n is approximately (1/7) for moderate Reynolds number $\left(\text{Re}_x = \frac{Vx}{v} < 10^7\right.$ for a flate plate.

Turbulent boundary layer is usually thicker than laminar boundary layer.

11. The thickness of turbulent boundary layer is given by

$$\delta = \frac{0.376x}{(\text{Re}_x)^{1/5}}$$

12. The average drag coefficient C_f for turbulent boundary layer developed along a long thin plate held stationary in a uniform stream in the direction parallel to the flow is given by

$$C_f = \frac{0.074}{(\text{Re}_L)^{1/5}}$$

13. If the plate is covered with both laminar and turbulent boundary layers the drag coefficient C_f is given by

$$C_f = \frac{0.074}{(\text{Re}_L)^{1/5}} - \frac{1700}{\text{Re}_L}$$

14. If the plate is smooth, even in the zone of turbulent boundary layer, there exists a very thin layer immediately adjacent to the boundary, in which the folw is laminar. This thin layer is commonly known as *Laminar Sublayer*. The thickness of laminar sublayer is represented by δ' which is given by

$$\delta' = \frac{11.6\,\nu}{V_*}$$

in which $V_* = \sqrt{(\tau_0/\rho)}$ is known as shear or friction velocity.

15. With the pressure increasing in the direction of flow i.e., with positive pressure gradient, the boundary layer separates from the boundary surface, and back flow and formation of eddies takes place due to which a great loss of energy occur. This phenomenon is dalled *separation of flow*.

16. The conditions for attached flow, separation and detached flow are :

(i) For attached flow

$$\left(\frac{\partial v}{\partial y}\right)_{y=0} > 0 \quad ; \quad \frac{\partial p}{\partial x} < 0$$

(ii) For separation

$$\left(\frac{\partial V}{\partial y}\right)_{y=0} = 0; \quad \frac{\partial p}{\partial x} = 0$$

(iii) For detached flow

$$\left(\frac{\partial V}{\partial y}\right)_{y=0} < 0 \; ; \; \frac{\partial p}{\partial x} > 0$$

17. The various methods of controlling the formation as well as separation of boundary layer are as follows :

(i) Motion of solid boundary.

(ii) Acceleration of fluid in the boundary layer.

(iii) Suction of fluid from the boundary layer.

(iv) Streamlining of body shapes.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

PROBLEMS

12.1 Explain the characteristics of laminar and turbulent boundary layer.

12.2 What are the factors affecting the boundary layer thickness?

12.3 What do you understand by displacement thickness and momentum thickness?

12.4 For a linear velocity distribution in the boundary layer on a flat plate show that $(\delta^*/\theta) = 3$, and $(\delta E/\delta) = 0.25$.

12.5 Discuss the phenomenon of separation in a diverging flow.

12.6 Discuss (i) the concept of the boundary layer with reference to fluid motion over a flat plate; (ii) phenomenon of separation for flow over curved surfaces; (iii) the prevention of separation.

12.7 Define boundary layer and explain the fundamental causes of its existence. Also discuss the various methods of controlling the boundary layer.

12.8 For the following types of velocity distribution obtain the values of (δ^*/δ) and (θ/δ).

(i) $\frac{v}{V} = 2\eta - \eta^2$; (ii) $\frac{v}{V} = 2\eta - 2\eta^3 + \eta^4$ where $\eta = (y/\delta)$.

$$\left[\text{Ans. (i) } \frac{1}{3}, \frac{2}{15}; \quad \text{(ii)} \frac{3}{10}, \frac{37}{31}\right]$$

12.9 If a laminar boundary layer at zero pressure gradient over a flat plate is described by the velocity profile

$$\frac{v}{V_0} = \frac{3}{2}\eta - \frac{\eta^3}{2}$$

in which $\eta = (y/\delta)$, show that boundary layer thickness δ, wall shear stress τ_0 and coefficient of drag C_D are given by

$$d = \frac{4.65x}{\sqrt{\text{Re}_x}}; \; t_0 = \frac{0.322\rho V_0^2}{\sqrt{\text{Re}_x}}; \; C_D = \frac{1.328}{\sqrt{\text{Re}_L}}$$

12.10 A smooth flat plate 1 m wide and 1.5 m long is rowed lengthwise through still air with a velocity of 10 m/s. Assuming the boundary layer to be fully laminar, estimate its thickness at the trailing edge. Mass density and kinematic viscosity of the air are 1.216 kg/m^3 (0.124 msl/m^3) and 0.15×10^{-4} m^2/s (0.15 cm^3/s) respectively. Also calculate the shear stress at that point.

[*Ans.* 7.5 mm; 40.37×10^{-3} N/m^2{41.17×10^{-4} kg(f)/m^2}]

12.11 Find the ratio of skin friction drag on the front half and rear half portions of a flat plate kept in a uniform stream of zero incidence. Assume the boundary layer to be turbulent over the entire plate. [*Ans.* 1.35]

12.12 Air (kinematic viscosity 0.15×10^{-4} m^2/s) flows at 10 m/s past a smooth rectangular flat plate 0.3 m wide and 3 m long. Assuming that the turbulence level in the oncoming stream is low and that transition occurs at $\text{Re}_x = 5 \times 10^5$, calculate the ratio of the total drag when the flow is parallel to the length of the plate to the value when the flow is parallel to the width. [*Ans.* 1.08]

12.13 A smooth flat plate 2.4 m long and 0.9 m wide moves lengthwise at 6 m/s through still atmospheric air of density 1.226 kg/m^3 (0.125 msl/m^3) and kinematic viscosity 1.49×10^{-5} m^2/s. Assuming the boundary layer to be entirely laminar, calculate the boundary layer thickness at the trailing edge of the plate, the shear stress half-way along the plate and the power required to move the plate. What power would be required if the boundary layer was made turbulent at the leading edge?

[*Ans.*12.2 mm; 2.108×10^{-2} N/m^2 {2.149×10^{-3} kg(f)/m^2}; 0.773 × 10^{-3} kW (1.05×10^{-3} h.p.); 2.688 × 10^{-3} kW (3.655×10^{-3} h.p.)]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Laminar Flow

Chapter 13

13.1 INTRODUCTION

In laminar flow the fluid particles move along straight parallel paths in layers or laminae, such that the paths of the individual fluid particles do not cross those of the neighbouring particles. It occurs at low velocity so that forces due to viscosity predominate over the inertial forces. The viscosity of fluid induces relative motion within the fluid as the fluid layers slide over each other, which in turn gives rise to shear stresses. The magnitude of the viscous shear stress so produced, varies from point to point, being maximum at the boundary and gradually decreasing with increase in the distance from the boundary. The shear stress so produced result in developing a resistance to flow. In order to overcome the shear resistance to flow the pressure drops from section to section in the direction of flow, so that a pressure gradient exists. An expression relating shear and pressure gradients in laminar flow may be developed, which may be used to analyse the various cases of laminar flow as indicated below.

13.2 RELATION BETWEEN SHEAR AND PRESSURE GRADIENTS IN LAMINAR FLOW

Consider a free-body of fluid having the form of an elementary parallelopiped of length δx, width δz, thickness δy as shown in Fig. 13.1. On account of relative motion between different layers of fluid, the velocity distribution is non-uniform. Thus the fluid layer *abcd* is moving at a higher velocity than the layer below it and hence the layer *abcd* exerts a shear stress in the positive direction on the lower layer. The lower layer on the other hand, exerts an equal and opposite shear stress on the layer *abcd*. Similarly it can be seen that a shear stress is exerted on the layer *a′b′c′d′* by the layer above it in the positive x-direction. The magnitudes of shear stresses on the layers *abcd* and *a′b′c′d′* will be different. Thus if τ represents the shear stress on the layer *abcd* then the shear stress on the layer *a′b′c′d′* is equal to

$$\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right).$$

For two-dimensional steady flow there will be no shear stresses on the vertical faces *abb′a′* and *cdd′c′*. Thus the only forces acting on the parallelopiped in the direction of flow x will be the pressure and shear forces as shown in Fig. 13.1. The net shear force acting on the parallelopiped

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \left[\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right)\delta x \delta z - \tau \delta x \delta z\right] = \frac{\partial \tau}{\partial y}\delta x \delta y \delta z.$$

If the pressure intensity on face *add'a'* is p, and since there exists a pressure gradient in the direction of flow, the pressure intensity on the face *bcc'b'* will be $\left(p + \frac{\partial p}{\partial x}\delta x\right)$. The net pressure force acting on the parallelopiped

$$= \left[p\delta y \delta z - \left(p + \frac{\partial p}{\partial x}\delta x\right)\delta y \delta z\right] = -\left(\frac{\partial p}{\delta x}\right)\delta x \delta y \delta z.$$

For steady and uniform flow, there being no acceleration in the direction of motion, the sum of these forces in the x-direction must be equal to zero. Thus

$$\left(\frac{\partial \tau}{\partial y}\right)\delta x\, \delta y\, \delta z \; - \; \left(\frac{\partial p}{\partial x}\right)\delta x, \delta y, \delta z = 0$$

or
$$\frac{\partial \tau}{\partial y} = \frac{\partial p}{\partial x} \qquad ...(13.1)$$

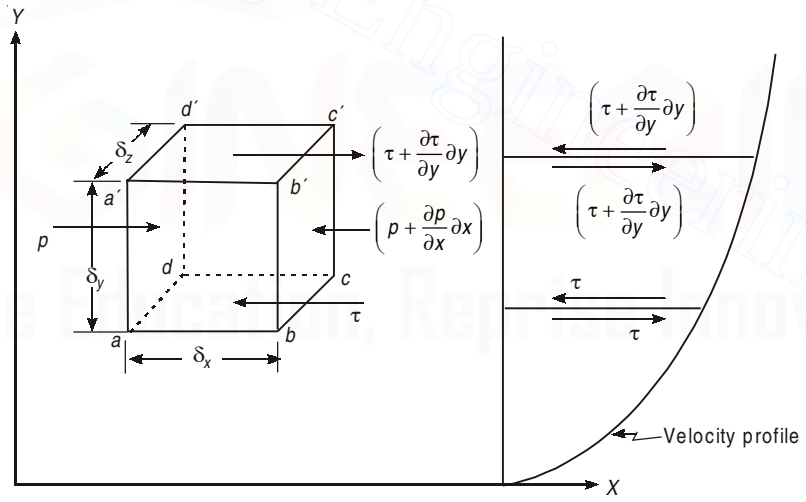


Figure 13.1 *Forces on a fluid element in laminar flow*

Equation 13.1 indicates that in a steady uniform laminar flow the pressure gradient in the direction of flow is equal to the shear stress gradient in the normal direction. Further for steady uniform flow, since acceleration is absent, it is apparent that the pressure gradient $(\partial p/\partial x)$ is independent of y and the shear stress gradient $(\partial \tau/\partial y)$ is independent of x.

Moreover, for viscous fluids according to Newton's law of viscosity $\tau = \mu(\partial v/\partial y)$. Thus substituting for τ in Eq. 13.1 the following differential equation for laminar flow is obtained

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\mu \frac{\delta^2 v}{\delta y^2} = \frac{\partial p}{\partial x} \quad \ldots (13.1)$$

By integrating Eq. 13.1 or 13.2 the problems of steady uniform laminar flow can be analysed.

13.3 STEADY LAMINAR FLOW IN CIRCULAR PIPES—HAGEN-POISEUILLE LAW

Figure 13.2 shows a horizontal circular pipe having laminar flow of fluid through it. Consider two sections 1 and 2 of this pipe L distance apart. Let p_1 and p_2 be the average intensities of pressure acting at the two sections 1 and 2 respectively and let V be the mean velocity of flow of fluid in the pipe. Applying Bernoulli's equation between the two sections 1 and 2,

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + Z_2 + h_f$$

where h_f is the loss of head due to resistance to flow between the sections 1 and 2. Since $V_1 = V_2 = V$ and $Z_1 = Z_2$,

$$\text{loss of head} = h_f = \left(\frac{p_1}{w} - \frac{p_2}{w} \right)$$

i.e., pressure gradient must exist in the direction of flow in order to overcome the resistance to flow and the loss of head is proportional to pressure drop $(p_1 - p_2)$ occurring in the length L of the pipe.

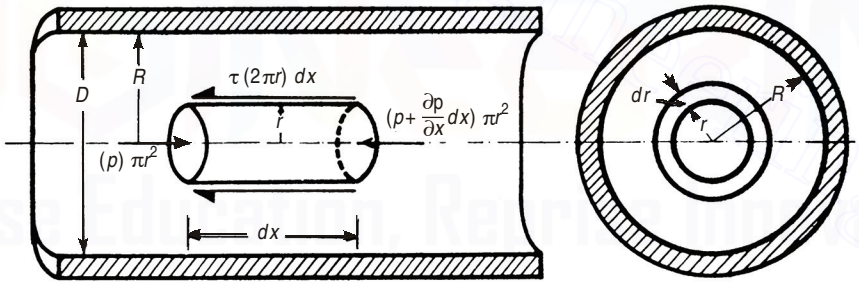


Figure 13.2 *Laminar flow through a circular pipe*

A small concentric cylindrical fluid element of radius r and length dx is chosen as free-body as shown in Fig. 13.2. Since the flow is steady and the size of the cross-section does not change every particle of fluid moves without acceleration. Therefore the summation of forces on the free-body in the direction of motion must be equal to zero. The forces acting on the fluid element in the direction of motion are normal pressure forces over the end areas and shear forces over the curved surface of the cylinder. If p is the pressure intensity on the left face then as indicated above since a pressure gradient must exist in the direction of flow in order to overcome the shear resistance, the pressure intensity on the right face will be $\left(p + \frac{\partial p}{\partial x} dx \right)$. The shear stress τ on the periphery of the cylinder will be acting in the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

direction opposite to that of the flow of fluid and on account of symmetry its magnitude will be constant on whole of the periphery. The total pressure forces on the left and the right faces of the cylinder are $(p)\pi r^2$ and $\left(p+\frac{\partial p}{\partial x}dx\right)\pi r^2$ respectively and total shear force acting on the periphery of the cylindrical element is $\tau(2\pi r)dx$. Since the summation of all these forces in the x-direction must be equal to zero,

$$p\left(\pi r^2\right)-\left(p+\frac{\partial p}{\partial x}dx\right)\pi r^2-\tau(2\pi r)dx = 0$$

or $$\left(-\frac{\partial p}{\partial x}dx\right)\pi r^2-\tau(2\pi r)dx = 0$$

Dividing the above equation by the volume of the element $(\pi r^2)dx$ and further simplifying it, we get

$$\tau = -\frac{\partial p}{\partial x}\frac{r}{2} \qquad ...(13.3)$$

Equation 13.3 shows that the shear stress τ varies linearly along the radius of the pipe (Fig. 13.3). At the centre of the pipe since $r = 0$, the shear stress τ is zero and at the pipe wall, since $r = R$ the maximum value of the stress τ_0 is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} \qquad ...(13.4)$$

The negative sign for the pressure gradient $\left(\frac{\partial p}{\partial x}\right)$ indicates a decrease of fluid pressure in the direction of flow. The pressure must decrease in the direction of flow in a horizontal pipe because pressure force is the only means available to compensate for the resistance to the flow, the potential and kinetic energies remain constant. Thus the term $\left(-\frac{\partial p}{\partial x}\right)$ is positive. Equation 13.3 however holds good for both laminar and turbulent flow, since in deriving it no assumptions were made about the nature of the flow.

Since the pipe is uniform the pressure gradient will be constant and hence

$$\left(-\frac{\partial p}{\partial x}\right) = \left(\frac{p_1-p_2}{L}\right) = \frac{wh_f}{L}$$

Introducing the above expression in Eq. 13.3, we get

$$\tau = \frac{wh_f}{2L}r \qquad ...(13.5)$$

In laminar flow the shear stress τ is entirely due to viscous action and therefore it may be evaluated by using Newton's law of viscosity, according to which $\tau = \mu\left(\frac{\partial v}{\partial y}\right)$; in which v is the velocity of flow

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

at a distance y from the pipe wall. Since in the present case $y = (R - r)$, it follows that $dy = -dr$ and the expression for τ then becomes

$$\tau = -\mu \frac{\partial v}{\partial r} \qquad ...(13.6)$$

in which the negative sign is on account of particular choice of coordinates and predicts mathematically that velocity v decreases as the radius r increases.

Substituting the value of τ from Eq. 13.6 in Eq. 13.3, we have

$$-\mu \frac{\partial v}{\partial r} = -\frac{\partial p}{\partial x}\frac{r}{2}$$

or

$$\frac{\partial v}{\partial r} = \frac{1}{\mu}\frac{\partial p}{\partial x}\frac{r}{2} \qquad ...(13.7)$$

For steady uniform flow since the drop in pressure depends only on the distance x and is independent of r, the pressure gradient $(\partial p/\partial x)$ in the direction of flow must have a constant value. The integration of Eq. 13.7 with respect to r then yields the expression for the velocity distribution for laminar flow in a circular pipe as

$$v = \frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right) r^2 + C \qquad ...(13.8)$$

The constant of integration C can be evaluated from the boundary condition that at $r = R$, the velocity $v = 0$.

$$\therefore \quad C = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right) R^2$$

Equation 13.8 then becomes

$$v = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2 - r^2) \qquad ...(13.9)$$

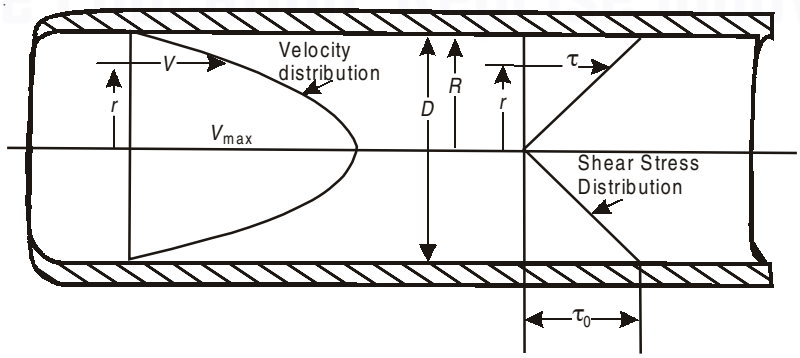


Figure 13.3 *Velocity and shear-stress distribution for laminar flow in a circular pipe*

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 13.9 shows that in laminar flow through circular pipes, velocity of flow varies parabolically, and the surface of velocity distribution is a paraboloid of revolution as shown in Fig. 13.3. The maximum velocity v_{max} occurs at the axis of the pipe and has the magnitude of

$$v_{max} = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)R^2 \qquad ...(13.10)$$

by combining Eq. 13.9 and 13.10 the expression for the velocity distribution can also be written in terms of the maximum velocity as

$$v = v_{max}\left[1-\left(\frac{r}{R}\right)^2\right] \qquad ...(13.11)$$

The discharge Q passing through any cross-sectio of a circular pipe can be obtained by integrating a small discharge passing through an elementary ring of thickness dr, considered at a radial distance r, out of the cross sectional area of the pipe, as shown in Fig. 13.2.

Since $dQ = vdA = v(2\pi r)\,dr$

$$= \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2 - r^2)\,(2\pi r)\,dr$$

By integrating both sides, we get

$$Q = \frac{\pi}{2\mu}\left(-\frac{\partial p}{\partial x}\right)\int_0^R (R^2 - r^2)\,r\,dr$$

or
$$Q = \frac{\pi}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^4 \qquad ...(13.12)$$

If D is the diameter of the pipe, then Eq. 13.12 expressed in terms of D becomes

$$Q = \frac{\pi}{128\mu}\left(-\frac{\partial p}{\partial x}\right)D^4 \qquad ...(13.13)$$

The mean velocity of flow V, which is equal to (Q/A) or $(Q/\pi R^2)$, is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^2 = \frac{1}{32\mu}\left(-\frac{\partial p}{\partial x}\right)D^2 \qquad ...(13.14)$$

Comparison of Eqs 13.10 and 13.14 reveals that

$$V = \frac{1}{2}\,v_{max} \qquad ...(13.15)$$

The relation between the mean and the maximum velocities of flow, given in Eq. 13.15, also follows from the geometry of a paraboloid of revolution, since it is known that the volume of a paraboloid of revolution is equal to half the volume of the circumscribing cylinder.

The point where the local velocity is equal to the mean velocity V, can be located by combining Eqs 13.11 and 13.15. Thus

$$v = v_{max}\left[1-\left(\frac{r}{R}\right)^2\right] = V = \frac{1}{2}\,v_{max}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$r = \frac{1}{\sqrt{2}} R = 0.707R \quad \text{...(13.16)}$$

That is the mean velocity of flow V occurs at a radial distance of $(0.707R)$ from the centre of the pipe.

Rearranging Eq. 13.14 to solve for pressure drop, we have

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{32\mu V}{D^2}$$

or $$\frac{p_1 - p_2}{L} = \frac{32\mu V}{D^2}$$

or $$(p_1 - p_2) = \frac{32\mu VL}{D^2} = \frac{128\mu QL}{\pi D^4} \quad \text{...(13.17)}$$

Equation 13.17 is known as Hagen–Poiseuille equation for laminar flow in the circular pipes. It was first determined experimentally by a German Engineer G.H.L. Hagen in 1839 and almost simultaneously but independently by J.L.M. Poiseuille, a French physician in 1840. The striking feature of Hagen–Poiseuille equation is that it involves no empirical coefficient or experimental factor of any kind, except for the physical properties such as viscosity and specific weight of the flowing fluid.

Further if h_f represents the drop in pressure head, then

or $$\left.\begin{aligned} h_f &= \frac{p_1 - p_2}{w} = \frac{32\mu VL}{wD^2} \\ h_f &= \frac{64}{Re}\left(\frac{LV^2}{2gD}\right) = \left(\frac{128\mu QL}{w\pi D^4}\right) \end{aligned}\right\} \quad \text{...(13.18)}$$

where Reynolds number $\mathrm{Re} = (\rho DV/\mu)$ and w is the specific weight of the flowing fluid. It may again be noted that the drop in pressure head h_f is nothing but the head loss over length L of the pipe, which varies directly as the first power of the velocity of flow of fluid and inversely as the square of the diameter of the pipe.

Several investigators have verified the validity of the Hagen–Poiseuille equation, and hence it is often employed for experimental determination of the fluid viscosity. From Eq. 13.17, we have

$$\mu = \frac{(p_1 - p_2)D^2}{32VL} = \frac{(p_1 - p_2)D^4\pi}{128QL} \quad \text{... (13.19)}$$

The pressure drop $(p_1 - p_2)$ in a fixed length L of a pipe of diameter D for any given rate of laminar flow can be measured. Thus all the quantities on the right hand side of Eq. 13.19 will be known, from which the value of the coefficient of viscosity μ of the fluid can be obtained.

The loss of head due to frictional resistance in a long straight pipe of length L and diameter D may also be expressed by Darcy–Weisbach equation as

$$h_f = \frac{p_1 - p_2}{w} = f\frac{L}{D}\frac{V^2}{2g} \quad \text{... (13.20)}$$

where f is friction factor and V is mean velocity of flow. Equating the two values of h_f given by Eqs. 13.18 and 13.20, we obtain

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{fLV^2}{2gD} = \frac{32\mu VL}{wD^2}$$

or
$$f = \frac{64\mu}{\rho VD} = \frac{64}{\text{Re}} \qquad ...(13.21)$$

Thus the value of friction factor f, for laminar flow of fluid is given by Eq. 13.21.

Further from Eq. 13.4

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} = \left(\frac{p_1 - p_2}{L}\right)\frac{D}{4}$$

By substituting for $(p_1 - p_2)$ from Eq. 13.20 in the above expression, we get

$$\tau_0 = \frac{(wh_f)D}{4L} = \frac{wfV^2}{8g} = \frac{\rho fV^2}{8} \qquad ...(13.22)$$

where ρ is mass density of fluid.

From Eq. 13.22,we obtain

$$\sqrt{\frac{\tau_0}{\rho}} = V_* = V\sqrt{\frac{f}{8}} \qquad ...(13.23)$$

Since the expression $\left[\sqrt{(\tau_0/\rho)}\right]$ has the dimensions of velocity, it is known as shear velocity and is denoted by V_*.

In order to maintain steady uniform laminar flow through pipes, certain power is required to overcome the resistance to flow. Since power is rate of doing work, it is equivalent to the product of force and velocity. The resistance to flow is compensated by the pressure gradient$(-\partial p/\partial x)$ in the direction of flow. The pressure gradient may be considered as the average force per unit volume of fluid. Thus if A and L are the area and length of pipe respectively, then the total force is $(-\partial p/\partial x) \times AL$, and with the mean velocity of flow V the required power is

$$P = \left(-\frac{\partial p}{\partial x}\right)ALV = Q(p_1 - p_2) \qquad ...(13.24)$$

since discharge $Q = AV$, and $\left(-\frac{\partial p}{\partial x}\right) = \frac{(p_1 - p_2)}{L}$.

13.4 LAMINAR FLOW THROUGH INCLINED PIPES

When the pipe having laminar flow of fluid is inclined as shown in Fig. 13.4 in addition to pressure and viscous forces, gravity forces will also become effective. Considering a small concentric cylindrical fluid element, if specific weight of the fluid is w, the weight of the element is $[w(\pi r^2)dx]$ acting vertically downward, and its component in the direction of flow becomes $[w(\pi r^2)dx]\cos\theta$, in which x-direction is taken parallel to the axis of the pipe. Now if z-direction is taken vertically upward, let dz represent a change in elevation corresponding to length dx of the fluid element.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Then from Fig. 13.4, we have

$$\cos\theta = -\frac{dz}{dx} = -\frac{\partial z}{\partial x}$$

in which the minus sign has been considered because the elevation z decreases as the distance x inceases. The component of the weight of the fluid element may thus be written as

$$[w(\pi r^2)dx]\cos\theta = [w(\pi r^2)dx]\left(-\frac{dz}{dx}\right) = -w(\pi r^2)dz.$$

The other forces acting on the fluid element are the shear and pressure forces as shown in Fig. 13.4.

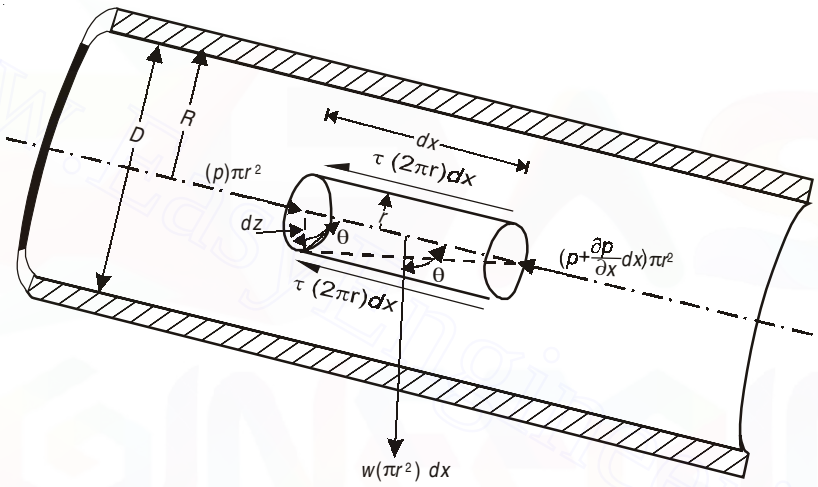


Figure 13.4 *Laminar flow through an inclined pipe*

In the absence of any acceleration, sum of all the forces acting on the element in the direction of flow must be equal to zero. Thus

$$P\pi r^2 - \left(P + \frac{\partial p}{\partial x}dx\right)\pi r^2 - w\pi r^2 dz - \tau(2\pi r)\,dx = 0$$

or
$$\tau = -\frac{\partial}{\partial x}(p + wz)\frac{r}{2} = -w\frac{\partial}{\partial x}\left(\frac{p}{w} + z\right)\frac{r}{2} \quad \text{... (13.25)}$$

or
$$\tau = -w\frac{\partial h}{\partial x}\frac{r}{2} \quad \text{... (13.25 a)}$$

in which $h = [(p/w) + z]$ represents piezometric head. Comparison of Eq. 13.25 with Eq. 13.3 would reveal that in the case of inclined pipes the term $\partial(p + wz)/\partial x$ replaces the pressure gradient terms $\partial p/\partial x$. Thus Eq. 13.25 is general expression for the distribution of shear stress for laminar flow in circular pipes, from which Eq. 13.3 can be derived since for horizontal pipe $(\partial z/\partial x) = 0$. For pipes according to Newton's law of viscosity $\tau = -\mu(\partial v/\partial r)$ as indicated in Section 13.3. Thus by substitution Eq. 13.25 becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$-\mu \frac{\partial v}{\partial r} = -w \frac{\partial h}{\partial x} \frac{r}{2}$$

or
$$\frac{\partial v}{\partial r} = \frac{w}{\mu} \frac{\partial h}{\partial x} \frac{r}{2} \quad \text{...(13.26)}$$

Again for steady uniform flow the piezometric head gradient $(\partial h/\partial r)$ in the direction of flow must have constant value which depends on the distance x and is independent of r. Thus by integrating Eq. 13.26 with respect to r and applying the boundary condition that at $r = R$ the value of $v = 0$, we obtain

$$v = \frac{1}{4\mu} \times w \left(-\frac{\partial h}{\partial x}\right)(R^2 - r^2) \quad \text{... (13.27)}$$

The other expressions for this case can be similarly obtained by introducing $(w\partial h/\partial x)$ in place of $(\partial p/\partial x)$ and $w(h_1 - h_2)$ in place of $(p_1 - p_2)$ in the various other expressions of Section 13.3.

13.5 LAMINAR FLOW THROUGH ANNULUS

Consider steady laminar flow of fluid through annular space between two concentric circular tubes as shown in Fig. 13.5. Let the radius of the inner surface of the outer tube be R_1 and that of the outer surface of the inner tube be R_2. A fluid element having a shape of small concentric cylindrical sleeve of length dx and thickness dr considered at a radial distance r is chosen as free body. The forces acting on the fluid element in the direction of flow are normal pressure forces over the end areas and shear forces over inner and outer curved surfaces of the cylindrical element, which are shown in Fig. 13.5.

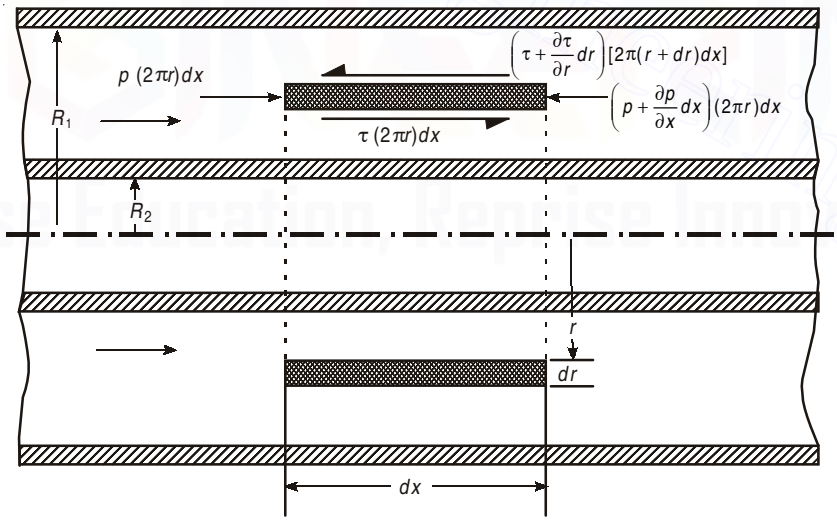


Figure 13.5 *Laminar flow through an annulus*

Again since the flow is steady and uniform, the summation of forces on the free body in the direction of flow must be zero. Thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left[p(2\pi r)\,dr-\left(p+\frac{\partial p}{\partial x}dx\right)2\pi r dr\right]+\left[\tau(2\pi r)\,dx-\left(\tau+\frac{\partial \tau}{\partial r}dr\right)\times 2\pi(r+dr)dx\right]=0$$

$$\text{or}\quad \left[\left(-\frac{\partial p}{\partial x}dx\right)2\pi r dr-\left(\frac{\partial \tau}{\partial r}dr\right)2\pi r dx\right]-\left[\tau(2\pi dx)\,dr+\left(\frac{\partial \tau}{\partial r}dr\right)(2\pi dx)dr\right]=0$$

Dividing through by the volume of the element $(2\pi r)\,dr\,dx$ and neglecting the last term which is a differential of higher order than the first three, we get

$$\left(\frac{\partial p}{\partial x}+\frac{\partial \tau}{\partial r}+\frac{\tau}{r}\right) = 0$$

In the above expression τ is a function of r only, and p is a function of x only. The last two terms may be combined to provide

$$\frac{\partial p}{\partial x}+\frac{1}{r}\frac{\partial}{\partial r}(\tau r) = 0. \qquad ...(13.28)$$

Since $\frac{\partial p}{\partial x}$ is not a function of r, Eq. 13.28 can be integrated with respect to r. Thus, we have

$$\left(\frac{\partial p}{\partial x}\right)\frac{r^2}{2}+\tau r = C_1$$

where C_1 is the constant of integration.

Since $\tau = -\mu(\partial v/\partial r)$, by substitution, we have

$$\left(\frac{\partial p}{\partial x}\right)\frac{r^2}{2}-\mu r\,\frac{\partial v}{\partial r} = C_1$$

Dividing the above expression throughout by r and again integrating it with respect to r, we get

$$\left(\frac{\partial p}{\partial x}\right)\frac{r^2}{4}-\mu v = C_1 \log_e r + C_2 \qquad ...(13.29)$$

in which C_2 is the second constant of integration.

The two constants of integration can be evaluated by applying the boundary conditions that at $r = R_1$ and $r = R_2$, $v = 0$. Thus the following expression for the velocity distribution in an annulus can be readily obtained :

$$v = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)\times\left[R_1^2-r^2-\frac{\left(R_1^2-R_2^2\right)}{\log_e(R_1/R_2)}\log(R_1/r)\right] \qquad \ldots (13.30)$$

In order to locate the point where maximum velocity occurs, we differentiate Eq. 13.30 with respect to r and equate it to zero. Thus, we have

$$\frac{\partial v}{\partial r} = 0 = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)\left[-2r+\frac{1}{r}\frac{\left(R_1^2-R_2^2\right)}{\log_e(R_1/R_2)}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \qquad r = \left[\frac{R_1^2 - R_2^2}{2\log_e(R_1/R_2)}\right]^{1/2} \qquad \ldots(13.31)$$

By substituting this value of r in Eq. 13.30 the value of maximum velocity may be obtained.

The discharge Q passing through any cross-section of the annulus is

$$Q = \int_{R_2}^{R_1} 2\pi r\, v\, dr$$

$$= \left(-\frac{\pi}{8\mu}\right)\left(\frac{\partial p}{\partial x}\right)\left[R_1^4 - R_2^4 - \frac{(R_1^2 - R_2^2)^2}{\log_e(R_1/R_2)}\right] \qquad \ldots(13.32)$$

The mean velocity of flow through the annulus is given by,

$$V = \frac{Q}{\pi(R_1^2 - R_2^2)} = \left(-\frac{1}{8\mu}\right)\left(\frac{\partial p}{\partial x}\right)\left[\left(R_1^2 + R_2^2 - \frac{(R_1^2 - R_2^2)}{\log_e(R_1/R_2)}\right)\right] \qquad \ldots(13.33)$$

It will be observed that if $R_2 = 0$ the problem becomes the same as laminar flow through a circular pipe and the above equations reduce to the same from as derived in Section 13.3.

If the tubes are inclined the relevant expressions for the velocity distribution and the discharge for an inclined annulus can be obtained by replacing $(\partial p/\partial x)$ by $[\partial(p + wz)/\partial x)$ as discussed in Section 13.4.

13.6 LAMINAR FLOW BETWEEN PARALLEL PLATES–BOTH PLATES AT REST

Consider laminar flow of fluid between two fixed parallel flat plates located at a distance B apart, as shown in Fig. 13.6. A small rectangular element of the fluid of length δx is considered as free-body as shown in Fig. 13.6. Let the lower face of the element be at a distance y from the lower plate and here let the velocity be v. At the upper face of the element which is at a distance of $(y + \delta y)$ from the lower plate, let the velocity be $(v + \delta v)$.

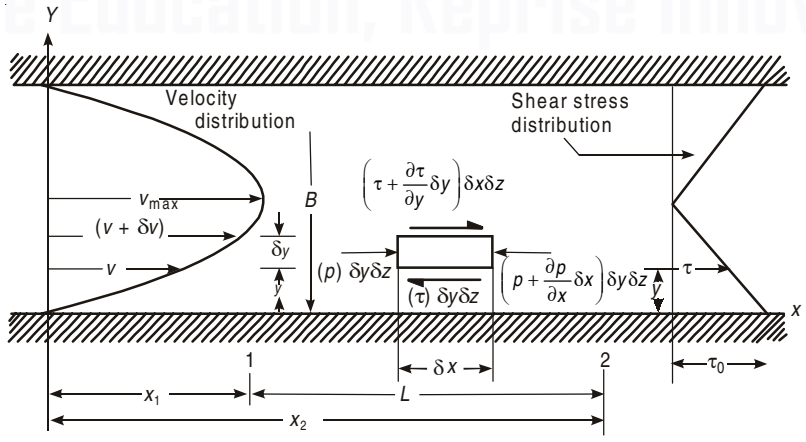


Figure 13.6 *Laminar flow between two fixed parallel flat plates*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If δv is positive, the faster moving fluid just above the upper face of the element exerts a forward force on the upper face. Similarly, the slower moving fluid adjacent to the lower face tends to retard its motion, i.e., it exerts a backward force on the lower face. Thus there are shear stresses of magnitude τ on the lower face and $\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right)$ on the upper face of the element in the directions as shown in Fig. 13.6. In order to balance the shearing forces in the fluid a pressure gradient in the direction of flow must be maintained. Thus if p is the pressure intensity at the left face of the element, then $\left(p + \frac{\partial p}{\partial x}\delta x\right)$ will be the pressure intensity on the right face of the element.

If the width of the element in the direction perpendicular to the paper is δz, the total force acting on the element towards the right is

$$\left[p - \left(p + \frac{\partial p}{\partial x}\delta x\right)\right] \partial y\ \partial z \ + \ \left[\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right) - \tau\right] \partial x\ \partial z$$

But for steady and uniform flow, there is no acceleration and hence this total force must be equal to zero.

$$\therefore \qquad -\frac{\partial p}{\partial x}\delta x\,\delta y\,\delta z + \frac{\partial \tau}{\partial y}\delta x\,\delta y\,\delta z = 0$$

Dividing by the volume of the element ($\delta x\,\delta y\,\delta z$), we get

$$\frac{\partial p}{\partial x} = \frac{\partial \tau}{\partial y}$$

which is same as Eq. 13.1 derived above. Again according to Newton's law of viscosity for laminar flow the shear stress $\tau = \mu \frac{\partial v}{\partial y}$. Hence by substitution in the above equation the following differential equation for laminar flow is obtained, which is same as Eq. 13.2.

$$\frac{\partial p}{\partial x} = \mu \frac{\partial^2 v}{\partial y^2}$$

Since $\left(\frac{\partial p}{\partial x}\right)$ is independent of y, integrating the above equation twice with respect to y gives

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2 \qquad ...(13.34)$$

The two constants of integration C_1 and C_2 may be evaluated by means of two boundary conditions. There being no slip of fluid at the two solid boundary surfaces of the plates,

$$v = 0 \text{ at } y = 0$$

$$\therefore \qquad C_2 = 0$$

and $$v = 0 \text{ at } y = B$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \qquad C_1 = -\frac{B}{2\mu}\left(\frac{\partial p}{\partial x}\right)$$

Introducing these values in Eq. 13.34 the following equation for the velocity distribution is obtained

$$v = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2) \qquad ...(13.35)$$

Equation 13.35 indicates that the velocity distribution curve for the laminar flow between parallel flat plates is a parabola with its vertex being mid-way between the plates. The negative sign for the pressure gradient indicates that there is a drop in pressure in the direction of flow.

The maximum velocity v_{max} which occur at the mid-point between the plates, can be evaluated by substituting $y = B/2$ in Eq. 13.35.

Thus

$$v_{max} = \frac{B^2}{8\mu}\left(-\frac{\partial p}{\partial x}\right) \qquad ...(13.36)$$

The discharge q of the laminar flow per unit width of the plate can be obtained by integrating a small discharge dq passing through strip dy.

Since $dq = v\,dy$

$$\therefore \qquad q = \int_0^B v\,dy = \int_0^B \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)\left(By - y^2\right)dy$$

or
$$q = \frac{B^3}{12\mu}\left(-\frac{\partial p}{\partial x}\right) \qquad ...(13.37)$$

The mean velocity of flow V is given by

$$V = \frac{q}{B} = \frac{B^2}{12\mu}\left(-\frac{\partial p}{\partial x}\right) \qquad ...(13.38)$$

Comparison of Eqs. 13.36 and 13.38 reveals that

$$V = \frac{2}{3}v_{max} \qquad ... (13.38a)$$

The relation between the mean and the maximum velocities of flow given in Eq. 13.38 (a) also follows from the geometry of a parabola, since it is known that the area of a parabola is equal to two-thirds the area of the enclosing rectangle.

Rearranging Eq. 13.38 to solve for the pressure gradient, we get

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{12\mu V}{B^2} \qquad ... (13.39)$$

Again since the flow is steady and uniform, the pressure drop $(p_1 - p_2)$ between any two sections 1 and 2 in the direction of flow at a distance $L = (x_2 - x_1)$ apart can be determined by integrating Eq. 13.39 with respect to x.

Thus
$$(p_1 - p_2) = \int_{p_1}^{p_2}(-\partial p) = \int_{X_1}^{X_2}\frac{12\mu V}{B^2}(\partial x)$$

or
$$(p_1 - p_2) = \frac{12\mu V}{B^2}(x_2 - x_1)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$(p_1 - p_2) = \frac{12\mu VL}{B^2} \quad \text{... (13.40)}$$

If h_f represents the drop in pressure head, then

$$h_f = \frac{p_1 - p_2}{w} = \frac{12\mu VL}{wB^2} \quad \text{...(13.41)}$$

In which w is the specific weight of the flowing fluid. It may be noted from Eq.13.41 that the drop in the pressure head (or the loss of head) for laminar flow varies linearly with the first power of velocity.

The distribution of shear stress in the flowing fluid across any section of the parallel plates may be determined by substituting Eq. 13.35 into Newton's law of viscosity. Thus

$$\tau = \mu \frac{\partial v}{\partial y}$$

or $$\tau = \mu \frac{\partial}{\partial y}\left[\frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2)\right]$$

or $$\tau = \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right) \quad \text{... (13.42)}$$

From Eq. 13.42 it is obvious that the shear stress varies linearly with the distance from the boundary. It has the same maximum value at either boundary (i.e., at $y = 0$ or $y = B$) and decreases linearly with the distance from the boundary, with the result that it is equal to zero at the centre line between the two plates $\left(\text{i.e., at } y = \frac{B}{2}\right)$. The shear stress distribution is shown in Fig. 13.6. If τ_0 represents the magnitude of the boundary shear stress, then from Eq. 13.42, for $y = 0$, we have

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2} \quad \text{... (13.43)}$$

It may be noted from Eq. 13.43 that the boundary shear stress is directly proportional to the pressure gradient and the boundary spacing.

If the plates are inclined, gravity forces will also become effective. This effect can be accounted for by considering the piezometric head $h\left(= \frac{p}{w} + z\right)$ instead of pressure head. The various expressions relevant to this case may thus be obtained by inserting $\left(w\frac{\partial h}{\partial x}\right)$ in place of $\left(\frac{\partial p}{\partial x}\right)$ in the various expressions derived for the flow between parallel flat plates placed horizontally.

13.7 LAMINAR FLOW BETWEEN PARALLEL FLAT PLATES—ONE PLATE MOVING AND OTHER AT REST—COUETTE FLOW

Consider laminar flow between two parallel flat plates located at a distance B apart, such that the lower plate is at rest and the upper plate moves uniformly in its own plane with a velocity V as shown in Fig. 13.7.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The flow of this type is known as Couette flow after M.F.A. Couette. The general expression for the velocity distribution for laminar flow between two parallel flat plates as derived earlier [Eq. 13.34] is also applicable for Couette flow. Thus

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2 \qquad \text{... (13.44)}$$

with the boundary conditions

$$v = 0 \text{ at } y = 0$$

$$\therefore \quad C_2 = 0$$

and $$v = V \text{ at } y = B$$

$$\therefore \quad C_1 = \frac{V}{B} - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)B$$

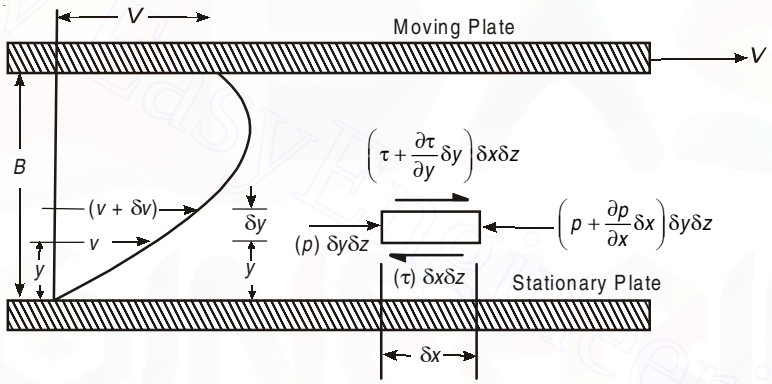


Figure 13.7 *Couette flow*

Hence, substituting the values of C_1 and C_2 in Eq. 13.44, it yields the following equation for the velocity distribution for Couette flow

$$v = \frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2) \qquad \text{...(13.45)}$$

It is thus observed that the velocity distribution in Couette flow depends on both V and $\left(\frac{\partial p}{\partial x}\right)$.

However, the pressure gradient $\left(\frac{\partial p}{\partial x}\right)$ in this case may be either positive or negative.

In particular case when $\left(\frac{\partial p}{\partial x}\right)$ equals zero, there is no pressure gradient in the direction of flow, then, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$v = V\frac{y}{B} \qquad ...(13.46)$$

which shows that the velocity distribution is linear. This particular case is known as *simple* (*or plain*) *Couette flow* or *simple shear flow*. Now, if V equals zero, Eq. 13.45 reduces to Eq. 13.35. Thus it may be stated that the general case of Couette flow is a superposition of the simple Couette flow over a laminar flow between two fixed parallel flat plates.

The distribution of shear stress across any section for the case of Couette flow may also be determined by substituting Eq. 13.45 into Newton's law of viscosity. Thus

$$\tau = \mu\frac{\partial v}{\partial y}$$

or
$$\tau = \mu\frac{\partial}{\partial y}\left[\frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)\right]$$

or
$$\tau = \mu\frac{V}{B} + \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right) \qquad ... (13.47)$$

Evidently in this case also the shear stress varies linearly with the distance from the boundary. However, as shown in Fig. 13.8, in this case the shear stress distribution at any section is asymmetrical with the values as given below.

At $y = 0$

$$\tau = \tau_{0_1} = \left[\frac{\mu V}{B} + \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2}\right]$$

At $y = \frac{B}{2}$

$$\tau = \frac{\mu V}{B}$$

At $y = \left[\frac{B}{2} + \frac{\mu V}{B}\left(-\frac{1}{\partial p/\partial x}\right)\right]$

$$\tau = 0$$

At $y = B$

$$\tau = \tau_{0_2} = \left[\frac{\mu V}{B} - \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2}\right]$$

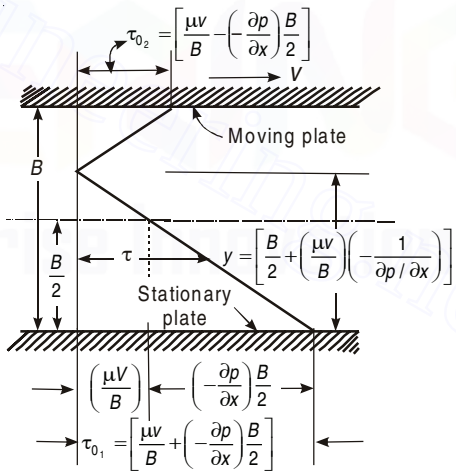


Figure 13.8 *Shear stress distribution for Couette flow*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

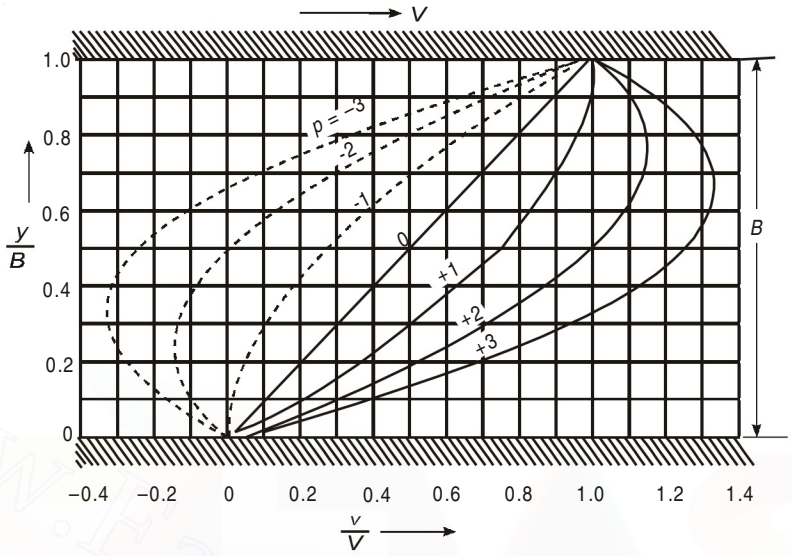


Figure 13.9 *Non-dimensional velocity distribution curves for Couette flow*

The velocity distribution for a general Couette flow can be transformed to the following non-dimensional form.

$$\frac{v}{V} = \frac{y}{B} + \frac{B^2}{2\mu V}\left(-\frac{\partial p}{\partial x}\right)\left(1-\frac{y}{B}\right)\frac{y}{B} \quad \text{... (13.48)}$$

If the constant term $\left[\frac{B^2}{2\mu V}\left(-\frac{\partial p}{\partial x}\right)\right]$ is replaced by P, called dimensionless pressure gradient, then the above expression becomes

$$\frac{v}{V} = \frac{y}{B} + P\left(1-\frac{y}{B}\right)\frac{y}{B} \quad \text{... (13.48a)}$$

A family of velocity distribution curves can be plotted in terms of (v/V) and (y/B) for different values of P. These are shown in Fig. 13.9. Positive values of P indicate a pressure drop in the direction of flow and negative values of P indicate an increase in fluid pressure in the direction of flow. It will be observed that for $P < (-1)$, for some layers of fluid there exists a backward flow, that is, opposite to the direction of V. This is on account of an adverse pressure gradient in the fluid. Further it may be noted that for a given value of P the maximum value of (v/V) occurs at a point where the shear stress is equal to zero, i.e., at a distance $y = \left[\frac{B}{2} + \frac{\mu V}{B}\left(-\frac{1}{\partial p/\partial x}\right)\right]$ above the lower plate.

The Couette flow with a pressure gradient has its application to the hydrodynamic theory of lubrication. The flow of lubricating oil in the narrow space in between journal and bearing is somewhat identical with Couette flow with a pressure gradient, which has been discussed in Section 13.11.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

13.8 LAMINAR FLOW OF FLUID IN AN OPEN CHANNEL

Consider steady, uniform, laminar flow in a very wide open channel. Let D be the depth of flow and $S_0 = (-\partial z/\partial x) = (\sin\theta)$, be the slope of the bed of the channel. In this case there is flow with a free surface, the pressure at every section will be same and therefore no pressure gradient is available to maintain the flow. However, in such a case the flow is maintained by the drop in the potential energy, i.e., the resistance to flow will be compensated by the gravity force. The required drop in potential energy will be provided by the bed slope of the channel. Equation 13.2 can be employed to analyse this case also, but the term $(\partial p/\partial x)$ should be replaced by $w(\partial z/\partial x)$, where w is the specific weight of the fluid. Thus for this case

$$w\frac{\partial z}{\partial x} = \mu\frac{\partial^2 v}{\partial y^2}$$

Again since the flow is uniform $(\partial z/\partial x)$ is independent of y. Thus integrating the above equation with respect to y, we get

$$\frac{\partial v}{\partial y} = \frac{w}{\mu}\left(\frac{\partial z}{\partial x}\right)y + C_1 \qquad ...(13.49)$$

The constant of integration C_1 may be evaluated by applying the boundary condition that at the free surface the velocity is maximum. Thus for $y = D$, $(\partial v/\partial y) = 0$

$$C_1 = -\frac{w}{\mu}\left(\frac{\partial z}{\partial x}\right)D$$

Introducing the value of C_1 in the above expression it becomes

$$\frac{\partial v}{\partial y} = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)(D-y) \qquad ... (13.50)$$

Again integrating it with respect to y the expression for the velocity distribution for laminar flow in an open channel is obtained as

$$v = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy - \frac{y^2}{2}\right) + C_2 \qquad ... (13.51)$$

The second constant of integration C_2 may be evaluated by applying the boundary condition that the velocity of flow is zero at the bed of the channel. Thus at $y = 0$, $v = 0$, therefore, $C_2 = 0$.

The velocity distribution then becomes

$$v = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy - \frac{y^2}{2}\right) \qquad ... (13.52)$$

This is a quadratic equation indicating that the velocity of uniform laminar flow in a wide open channel has a parabolic distribution. Integrating this equation from $y = 0$ to $y = D$ and dividing the result by D, the average velocity is obtained as

$$V = \frac{1}{D}\int_0^D v\,dy = \frac{wS_0D^2}{3\mu} = \frac{gS_0D^2}{3\upsilon}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and the discharge per unit width is obtained as

$$q = \frac{wS_0D^3}{3\mu} = C_L D^3 \quad ...(13.53)$$

where $C_L = (wS_0/3\mu)$, a coefficient involving slope and viscosity.

Rearranging the above expression for the potential head gradient, we get

$$S_0 = \left(-\frac{\partial z}{\partial x}\right) = \frac{3\mu V}{wD^2} \quad ...(13.54)$$

Again since the flow is steady and uniform, the above expression can be integrated with respect to x to provide the potential head drop $(z_1 - z_2)$ between any two sections 1 and 2 in the direction of flow at a distance L apart. Thus

$$(z_1 - z_2) = \frac{3\mu VL}{wD^2} \quad ...(13.55)$$

If h_f represents the loss of head in between the sections 1 and 2 then since $h_f = (z_1 - z_2)$ and hence

$$h_f = \frac{3\mu VL}{wD^2} \quad ...(13.55\ a)$$

The distribution of shear stress in the flowing fluid across any section of the channel may be determined by substituting the value of $(\partial v/\partial y)$ in Newton's law of viscosity. Thus we obtain

$$\tau = w\left(-\frac{\partial z}{\partial x}\right)(D - y) = wS_0(D - y) \quad ...(13.56)$$

From this expression it is evident that the shear stress varies linearly with distance from the boundary, having a maximum value at the bed and a minimum value of zero at the free surface. The value of the bed shear stress τ_0 is obtained by substituting $y = 0$ in the above expression. Thus we obtain

$$\tau_0 = wS_0D \quad ...(13.57)$$

13.9 LAMINAR FLOW THROUGH POROUS MEDIA

Another phenomenon that exhibits laminar flow is that of flow of water or any other liquid through porous materials which they saturate. Consider a circular pipe of length L and cross-sectional area A completely filled with porous material of grain diameter d, as shown in Fig. 13.10. Let Q be the discharge of liquid of specific weight w, flowing through the pipe. The flow will take place through the interstices or pores of the porous material which form numerous little conduits. Since it is not possible to analyse the flow through such individual conduit the porous material as a whole is considered in which the passage for liquid is bounded by the impervious solid sand grains. Thus the apparent (or superficial) mean velocity V is equal to (Q/A), but this will not be the actual mean velocity of flow of liquid. This is because part of the cross-sectional area of pipe is blocked by the solid grains of the porous material and the liquid flows only through the pores. Thus if η is the effective ratio of the area of pores A_p to the total area A [i.e., $\eta = (A_p/A)$], then the actual mean velocity of flow of liquid through the pores is given by (Q/A_p) which is same as $(Q/\eta A)$ or (V/η). Since the area of the pores A_p is smaller than the total area A, the actual mean velocity of flow through the pores will be greater than the apparent mean velocity V. However, the actual mean velocity as well as the pore size being quite small

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the flow will be laminar, and therefore, the loss of head when liquid flows through a porous media can be determined by using the general expression for head loss in laminar flows as indicated below.

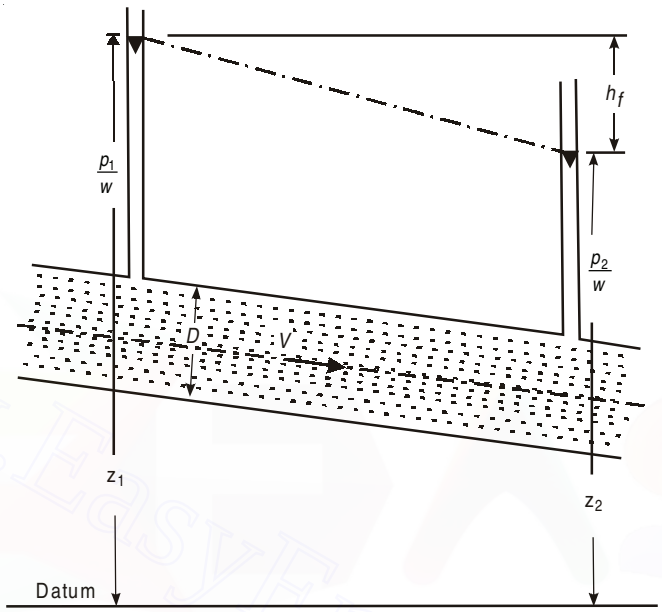


Figure 13.10 *Flow through porous media*

As derived in Section 13.3 the loss of head for laminar flow through a pipe is given by Eq. 13.18 as

$$h_f = \frac{32\mu VL}{wD^2}$$

Similarly the loss of head for laminar flow through parallel plates is given by Eq. 13.41 as

$$h_f = \frac{12\mu VL}{wB^2}$$

Hence the general expression for the loss of head for laminar flow through a passage of any shape may be expressed as

$$h_f = \frac{k\mu VL}{wD^2} \quad \ldots (13.58)$$

where h_f is the loss of head in length L, V is the mean velocity of flow, D is a characteristic length representing the geometry of the passage, and k is a constant the value of which depends on the shape of the passage.

In the case of flow through porous media the average size of the pores may be taken equal to the grain diameter d_s. Thus by substituting (V/η) for V and d_s for D in Eq. 13.58, the loss of head for laminar flow through a porous media is given by

$$h_f = \frac{k\mu VL}{w\eta d_s^2} \quad \ldots (13.59)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which can be rearranged for velocity V as

$$V = \left(\frac{w\eta d_s^2}{k\mu}\right)\left(\frac{h_f}{L}\right)$$

or $$V = Ki \quad \ldots (13.60)$$

where i is the hydraulic gradient and K is known as coefficient of permeability which has the dimensions of velocity. Equation 13.60 expresses Darcy's Law in its simplest form, which was first discovered experimentally by Henri Darcy, a French hydraulic engineer in 1856. Darcy's Law has been found to be quite useful in the analysis of various problems which invlove flow of liquid through porous media.

13.10 LAMINAR FLOW AROUND A SPHERE–STOKES' LAW

Laminar flow may also exist around a solid body which moves through a fluid of infinite extent, in such a manner that viscosity is the only fluid property affecting the pattern of motion and the accelerative effects are so small that they may be neglected. The practical examples in this context may be that of a dust particle falling in the air or a fine sand grain falling in water. When a body moves through any fluid, it experiences a resistance which acts in the direction opposite to that of the motion of the body. The resistance called *drag*, depends on the size of the object, the velocity with which it moves and the viscosity of the fluid.

In 1815 G.G. Stokes developed analytically an expression for the resistance F_D experienced by a sphere of diameter D, moving with a constant velocity V (i.e., there is no acceleration) in a fluid of viscosity μ, which is given by

$$F_D = 3\,\pi\mu VD \quad \ldots (13.61)$$

It is known as Stokes' Law which has been verified experimentally and is found to hold good for values of Reynolds number (VD/υ) less than 0.2.

If the sphere is dropping down through the fluid under its own weight at a constant velocity V, then the buoyant force plus the resistance to its motion must be equal to its weight. Thus we obtain

$$\left(\frac{\pi}{6}D^3\right)w + F_D = \left(\frac{\pi}{6}D^3\right)w_s$$

or $$F_D = \left(\frac{\pi}{6}D^3\right)(w_s - w)$$

in which w is specific weight of the fluid, w_s is specific weight of the sphere and $\left(\frac{\pi}{6}D^3\right)(w_s - w)$ represents the submerged weight of the sphere. Introducing the value of F_D from Eq. 13.61 in the above expression and solving for V, we obtain

$$V = \frac{D^2}{18\mu}(w_s - w) \quad \ldots (13.62)$$

The velocity V given by Eq. 13.62 is called terminal fall velocity, which is defined as the velocity attained by a body in falling through a fluid at rest, when the drag on the body is equal to the submerged weight of the body. As indicated later, the viscosity μ of a fluid can be determined

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

experimentally, if in Eq. 13.62 the fall velocity V, of a sphere of known diameter D, moving down through the fluid is measured.

The drag F_D for any body moving with velocity V in a fluid of mass density ρ may be expressed as

$$F_D = C_D A \frac{\rho V^2}{2}$$

in which C_D is known as coefficient of drag, and A is the projected area of the body perpendicular to the direction of flow. For the sphere $F_D = 3\pi\mu VD$ and $A = (\pi D^2/4)$; thus by substituting for F_D and A in the above equation, we get

$$C_D = 24\left(\frac{\mu}{\rho VD}\right) = \frac{24}{\text{Re}} \qquad \ldots(13.63)$$

Equation 13.63 represents the drag coefficient* for sphere on the basis of Stokes' Law. Experiments have shown that Eq. 13.63 holds good for Reynolds number less than 0.2 provided the sphere is moving in an infinite fluid. However, if the fluid through which the sphere moves is not infinite in extent but is confined within a container of finite dimensions, then the resistance to motion is increased and in such a case the drag coefficient is given by the following modified expression

$$C_D = \frac{24}{\text{Re}}\left[1 + 2.1\frac{D}{D_1}\right]$$

where D_1 is the smallest lateral dimension of the container and D is the diameter of the sphere.

13.11 LUBRICATION MECHANICS

One of the most important applications of laminar flow of viscous fluids arises in the lubrication of various types of bearings. The theory of lubrication was initiated by Reynolds who showed that *two parallel or nearly parallel surfaces can slide one over the other with but slight frictional resistance even under great normal pressure, provided a film of viscous fluid is maintained between them.* This is the working principle of various bearings with lubricating oil as viscous fluid. Although high velocities may be involved in the lubricating systems, the thickness of the film of the lubricant is so small that the Reynolds number is far below its critical value for that system. Therefore, laminar motion may always be assumed in the analysis of lubrication problems.

The primary function of the lubricant is to separate the bearing surfaces so that there is no direct contact between properly finished surfaces of two machine elements which are in relative motion. Further, if the film of lubricant is to keep the bearing surfaces apart when they are in relative motion it must be capable of sustaining a load, which is made possible by the pressure developed in the oil film. However, as indicated later the bearings can support a load only if the surfaces of the relatively moving parts are inclined to one another.

Two common types of bearings, viz, *slipper bearing* and *journal bearing* are described below.

13.11.1 Slipper bearing

In the simplest form a slipper bearing consists of a slipper or slide block moving over a horizontal bearing plate as illustrated in Fig. 13.11. The plane surface of the slipper is kept slightly inclined over

* For definition of drag coefficient see Chapter 18.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the plane surface of the bearing plate which is horizontal. For the purpose of simple analysis it is assumed that the bearing plate is infinite in extent and the slipper has infinite width in the direction

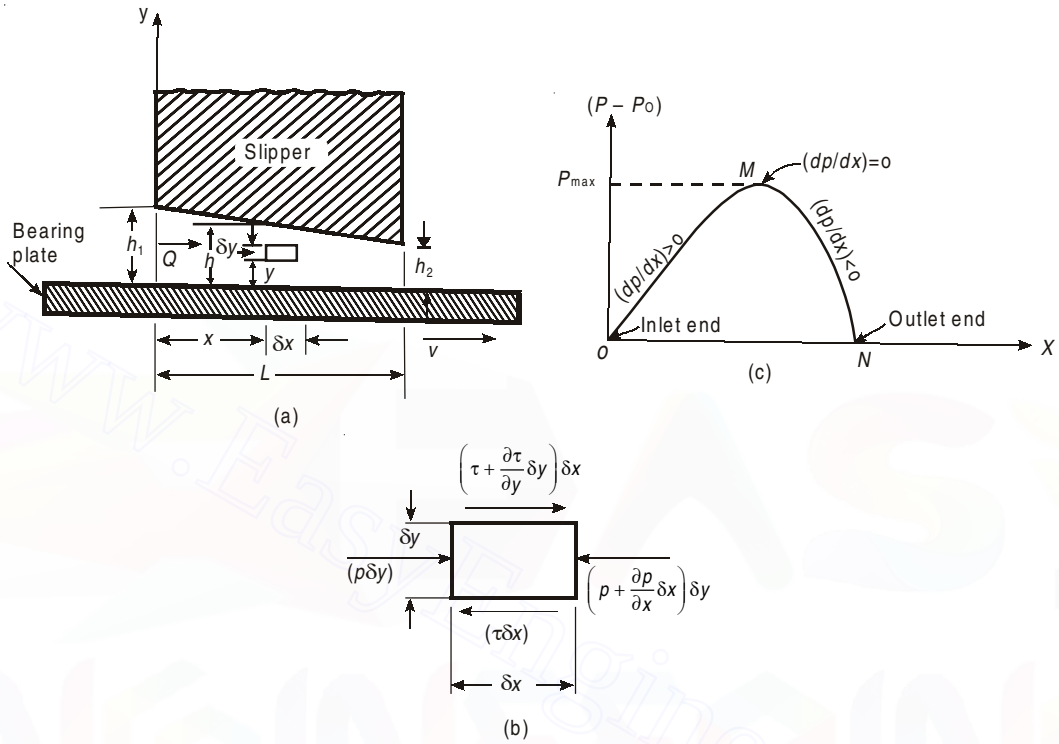


Figure 13.11 *Slipper bearing–a simplified sketch*

perpendicular to the plane of the paper, so that the flow of the lubricant may be considered two dimensional.

Actually the slipper or slide block is moving at a constant velocity V, but in order to simplify the analysis the principle of relative motion is applied to the entire system to bring the slipper at rest while the bearing plate moves past it with a constant velocity V in the x-direction. The behaviour of the lubricant in this case, therfore, corresponds to laminar flow between parallel plates with one plate moving, i.e., Couette flow as discussed in Section 13.7. However, to recapitulate the detailed analysis is given here again.

Consider a rectangular element of fluid situated at (x, y) as shown in Fig. 13.11 (a). This element is shown enlarged in Fig. 13.11 (b) together with the pressure and shear forces acting on its faces. For steady and uniform flow, since there is no acceleration, the sum of all the forces acting on the fluid element must be equal to zero. Hence for unit width of the element in the direction perpendicular to the paper,

$$p\delta y - \left(p + \frac{\partial p}{\partial x}\delta x\right)\delta y + \left(\tau + \frac{\partial \tau}{\partial y}\delta y\right)\delta x - \tau \delta x = 0.$$

or

$$\frac{\partial p}{\partial x}\delta x\,\delta y = \frac{\partial \tau}{\partial y}\delta x\,\delta y$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{\partial p}{\partial x} = \frac{\partial \tau}{\partial y}$$

Since according to Newton's law of viscocity $\tau = \mu(\partial v / \partial y)$, the above expression becomes

$$\frac{\partial p}{\partial x} = \mu \frac{\partial^2 v}{\partial y^2} \qquad ...(13.63\ a)$$

Again since $(\partial p / \partial x)$ is independent of y, integrating Eq. 13.63 (a) twice with respect to y gives

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2 \qquad ...(13.64)$$

which is same as Eq. 13.34 derived earlier. Equation 13.64 is a general equation which may be applied to this case as well. The constants of integration C_1 and C_2 can be evaluated by applying the following boundary conditions:

at $y = 0, v = V$ $\quad \therefore \; C_2 = V$

at $y = h, \; v = 0$ $\quad \therefore \; C_1 = -\frac{V}{h} - \frac{h}{2\mu}\frac{\partial p}{\partial x}$

Thus introducing the values of constants in Eq. 13.64 it becomes

$$v = V\left(1 - \frac{y}{h}\right) + \frac{h^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left(\frac{y}{h}\right)\left(\frac{y}{h} - 1\right) \qquad ...(13.65)$$

By comparison it may be seen that Eq. 13.65 is exactly same as Eq. 13.45 for the general case of Couette flow.

Considering the flow through an elementary strip of unit breadth and thickness dy

$$dQ = v\,dy$$

$$\therefore \quad Q = \int_0^h v\,dy = \text{constant}$$

$$= \int_0^h \left[V\left(1 - \frac{y}{h}\right) + \frac{h^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left(\frac{y}{h}\right)\left(\frac{y}{h} - 1\right)\right] dy$$

$$= \frac{Vh}{2} - \frac{1}{12\mu}\frac{\partial p}{\partial x} h^3$$

Solving for $(\partial p / \partial x)$, we obtain

$$\frac{\partial p}{\partial x} = \frac{12\mu}{h^3}\left(\frac{Vh}{2} - Q\right) \qquad ...(13.66)$$

From Fig. 13.10 it is seen that

$$h = h_1 - \frac{x}{L}(h_1 - h_2) = (h_1 - mx)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where $$m = \left(\frac{h_1 - h_2}{L}\right)$$

By substituting the value of h in Eq. 13.66, we obtain

$$\frac{\partial p}{\partial x} = 12\mu\left[\frac{V}{2(h_1 - mx)^2} - \frac{Q}{(h_1 - mx)^3}\right] \quad ..(13.67)$$

Integarating the above expression the pressure distribution along the slipper is given by

$$p = 12\mu\left[\frac{V}{2m(h_1 - mx)} - \frac{Q}{2m(h_1 - mx)^2}\right] + C_3$$

The pressure intensities at the two ends of the slipper are known since atmospheric pressure prevails at the ends, i.e., at $x = 0$, $p = 0$ and $x = L$, $p = 0$. Thus by substitution

$$C_3 = -\frac{6\mu V}{mh_1} + \frac{6\mu Q}{mh_1^2}$$

and $$Q = \frac{Vh_1h_2}{h_1 + h_2} \quad ...(13.68)$$

By substituting the values of C_3 and Q the pressure distribution along the slipper is given by

$$p = \frac{6\mu Vx(L - x)(h_1 - h_2)}{Lh^2(h_1 + h_2)} \quad ...(13.69)$$

The variation of pressure along the slipper as represented by Eq. 13.69 is shown in Fig. 13.11. Further by equating $(\partial p/\partial x)$ to zero it can be readily shown that the maximum pressure intensity p_{max} occurs at

$$x = \frac{h_1(h_1 - h_2)}{m(h_1 + h_2)} = \frac{Lh_1}{(h_1 + h_2)} \quad ...(13.70)$$

Introducing Eq. 13.70 in Eq. 13.69 the maximum pressure intensity is obtained as

$$p_{max} = \frac{3\mu V(h_1 - h_2)L}{2h_1h_2(h_1 + h_2)} \quad ...(13.71)$$

It is on account of the pressure developed in the oil film that the bearing supports the load. As shown in Fig. 13.11 the pressure increases from the zero (or atmospheric) pressure at the inlet end O to a maximum value at some point M, and then decreases again to zero (or atmospheric) at the outlet end N. The slope of the pressure distribution $(\partial p/\partial x)$ must therefore change from positive between O and M to negative between M and N.

From Eq. 13.66 it will be clear that if $(\partial p/\partial x)$ has to change from positive value to negative, $\left(\frac{Vh}{2} - Q\right)$ must change from positive between O and M to negative between M and N. Since Q is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

constant at all sections along the oil film it follows that the essential conditions for the development of pressure in the oil film are

(i) between O and M, $h > (2Q/V)$

(ii) at M, $h = (2Q/V)$

and (iii) between M and N, $h < (2Q/V)$

It, therefore, follows that for a bearing to sustain a load the thickness of the oil film h must decrease in the direction of flow of the lubricant.

The total force on the bearing surface per unit length can be obtained by integrating the pressure intensity over the width L.

Thus
$$P = \int_0^L p\,dx = \int_0^L \frac{6\mu V x(L-x)(h_1 - h_2)}{L(h_1+h_2)(h_1 - mx)^2}\,dx.$$

$$\therefore \quad P = \frac{6\mu VL^2}{(h_1 - h_2)^2}\left[\log_e(h_1/h_2) - \frac{2(h_1 - h_2)}{(h_1+h_2)}\right] \qquad ...(13.72)$$

It is evident from this expression that the total load P will be zero when $h_1 = h_2$, that is when the two surfaces are parallel. *In other words the moving parts of the bearing will sustain load only if the surfaces of the relatively moving parts are inclined to one another.*

The maximum load P_{max} is computed with Eq. 13.72 when $h_1 = 2.2\,h_2$ for which

$$P_{max} = \frac{0.16\,\mu VL^2}{h_2^2} \qquad ...(13.73)$$

In the above analysis it has been assumed that the bearing plate is of infinite extent and the slipper has infinite width, but the real bearings used in actual practice have a finite width. As such for real bearings the flow is three-dimensional. Michell has investigated the three-dimensional flow problem for bearing pads of finite width. In such cases the oil flows not only in the direction of motion of the bearing but also towards the sides, thus causing a decrease in the load or bearing capacity. According to Michell's analysis the load on the bearing of finite width B is reduced according to the following table:

B/L	∞	1	1/3
Load	P	0.422P	0.031 P

In the above table the total load P supported by the slipper bearing is as given by Eq. 13.72.

13.11.2 Journal Bearing

As shown in Fig. 13.12 a journal bearing consists of a sleeve which is wrapped partially or completely around a rotating shaft or journal. It is designed to support a radial load. It is possible to design a journal bearing to operate with its rubbing surfaces separated by lubricant film and still carry a substantial load. The journal bearing adjusts itself automatically to these conditions as discussed below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When the loaded journal is at rest, there is a metal to metal contact at the lowest point as shown in Fig. 13.12 (a). As the journal begins to rotate say in clockwise direction it moves up the bearing in an anticlockwise direction as shown in Fig. 13.12 (b). The lubricant then adheres to the surface of the journal which now rolls on a lubricated surface. When the speed of journal is sufficiently high, the

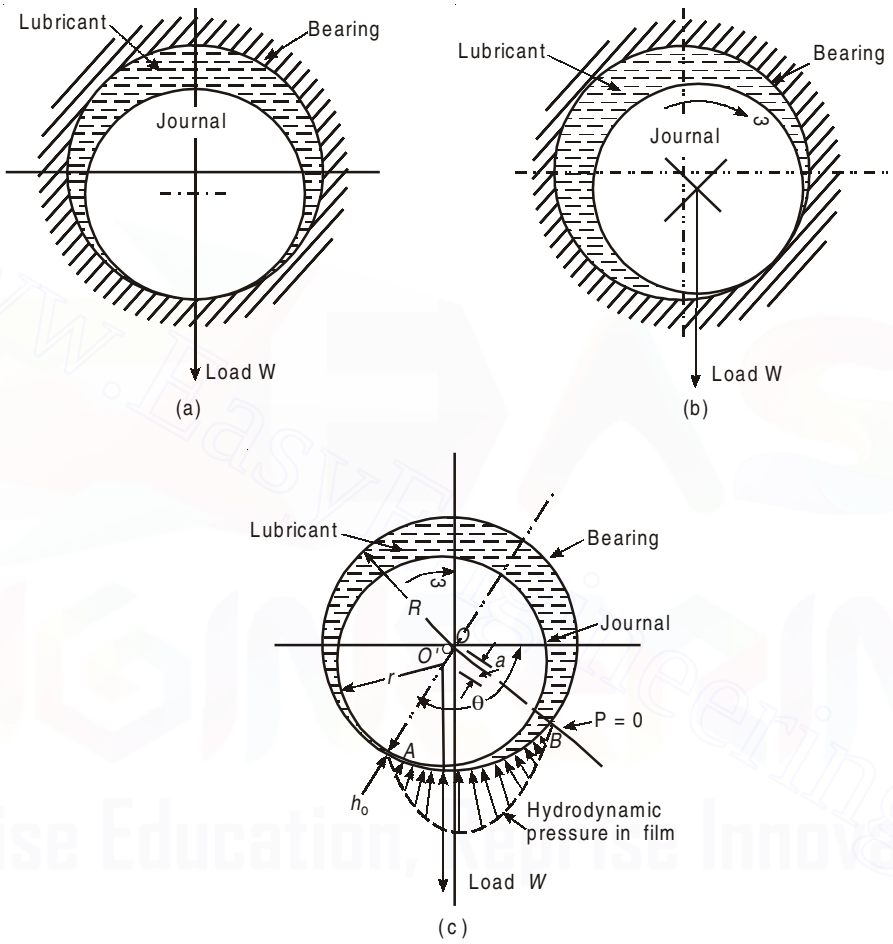


Figure 13.12 *Journal bearing*

lubricant will automatically be drawn between the surfaces, thus floating the journal in the bearing and the centre of the journal moves to the left of the centre of the bearing as shown in Fig. 13.12 (c). Once the condition shown in Fig. 13.12 (c) is reached, the convergent film of lubricant gets established and a positive pressure is developed which can support the load.

A polar diagram of film pressures is shown representing the extent of the arc of actual load carrying lubricant film. The leading edge of the film may be established by an oil distribution groove such as at *B* in Fig. 13.12 (c). The trailing edge of the load carrying film is considered as terminating at the point of closest approach between the shaft (or journal) and bearing. Beyond this point the clearance space is no longer converging, but begins to diverge. The film thickness at the point of closest approach is designated as h_o. Point *O* represents the centre of the bearing and *O'* the centre of the shaft. The radius of the shaft is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

r and the radius of the bearing is R. Distance OO' is indicated by a and it is the eccentric distance. The ratio of a for the actual operating conditions to the radial clearance in the bearing is called the eccentricity ratio ε. For light loads and high operating speeds the centre of the shaft and the bearing tend to coincide and a approaches zero. Thus ε also approaches zero. For heavy loads or extremes of operation such as low speed or very low viscosity the journal becomes more eccentric in the bearing and the distance a increases. The limit is where the journal just begins to make the metallic contact with the bearing or the distance a equals the radial clearance. The value of the eccentricity ratio ε is then 1.

Although the lubricant film in this case is curved, the same equations as derived for the case of slipper bearing can be applied for the journal bearing also, but with proper modifications as noted below.

The radial clearance in the bearing is c, which equals mr, where m is the clearance modulus of the bearing in *mm/mm*. The value of m usually range between 0.0005 and 0.003, with typical values for industrial type bearing running from 0.001 to 0.0025.

The value of ∂x in Eq. 13.66 is replaced by $r\partial\theta$ (Fig. 13.12c) also

$$h_0 + a = c = mr$$

but $$a = \varepsilon mr$$

so $$h_0 = mr\,(1-\varepsilon)$$

Also $$h + a\cos\theta = c = mr$$

so $$h = mr\,(1-\varepsilon\cos\theta).$$

Thus substituting these values in Eq. 13.66 it becomes

$$\frac{\partial p}{r\partial\theta} = 6\mu V\left[\frac{1}{m^2r^2(1-\varepsilon\cos\theta)^2} - \frac{2Q}{Vm^3r^3(1-\varepsilon\cos\theta)^3}\right]$$

or
$$\frac{\partial p}{\partial\theta} = \frac{6\mu Vr}{c^2}\left[\frac{1}{(1-\varepsilon\cos\theta)^2} - \frac{2Q}{Vc(1-\varepsilon\cos\theta)^3}\right] \quad ...(13.74)$$

Integrating the above equation, we obtain

$$p = \frac{6\mu Vr}{c^2}\left[\frac{\varepsilon(2-\varepsilon\cos\theta)\sin\theta}{(2+\varepsilon^2)(1-\varepsilon\cos\theta)^2}\right] \quad ...(13.75)$$

The above expression has been obtained by using the boundary conditions that $p = 0$ at $\theta = 0$ and the term $(2Q/V)$ has been determined by the condition that the pressure returns to its original value when θ has increased by 2π. Further in the above expressions $V = r\omega = (2\pi Nr/60)$, where N is the speed of the journal in rev./min.

To obtain the total load that can be carried by the bearing when it has an eccentricity ratio ε we must integrate the pressure over the convergent portion of the oil, that is between points A and B. This is because in the crown of the bearing a negative pressure will be established which will contribute very little to the load carrying function of the bearing, since the positive oil-film pressures are generally much larger than these negative pressures.

13.11.3 Properties of Lubricant

Oils and greases are the most widely used lubricants. Viscosity is the most important single property involved in lubrication in engineering applications. A detailed discussion of viscosity of fluids is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

given in Chapter 1. It is known that viscosity μ of common lubricating oils, like that of most liquids, falls rapidly with rising temperature. The degree to which the viscosity depends on temperature is called *viscosity index* (V.I.) of the oil. Oils which show the smallest change with temperature are assigned a higher value of V.I. say 100 and those which show the greatest change have a V.I. of zero. In between these two extreme categories of oils, there are several other oils with varying values of V.I. A lubricant with high V.I. is always desirable, but it is especially necessary for the systems which are exposed to extreme ranges of temperature.

In addition to viscosity there are some other properties of lubricants which are also required to be considered to judge the suitability of any particular lubricant. However, a detailed discussion of these may be seen in any of the books on lubricants.

13.12 DASH-POT MECHANISM

A dash-pot is a device used for damping vibrations of machines, which may be achieved by making use of a fluid of fairly high viscosity. The simplest kind of a dash-pot is shown in Fig. 13.13. It consists of a piston which may move in a concentric cylinder, the diameter of which is only slightly greater than that of the piston. The cylinder contains a viscous oil, the quantity of which should be sufficient to cover the top of the piston. The piston is connected to the machine component whose movement is to be restrained. If the piston is caused to move downwards, oil is displaced from underneath it. This displaced oil must move to the space above the piston and its only route is through the small annular clearance between the piston and the cylinder. If the oil is highly viscous and the clearance between the piston and the cylinder is small the flow of oil upwards through the clearance space is laminar, and it occurs simply as a result of the pressure developed underneath the piston. Obviously, more viscous is the oil and smaller is the clearance between the piston and the cylinder, the greater is the pressure required to produce a particular movement of oil, thus greater is the force required to move the piston or greater is the resistance to its motion.

Similarly for an upward movement of the piston, oil must flow downwards through the clearance space, since an upward movement of the piston reduces the pressure underneath it and the resulting difference of pressure betwen the top and bottom of the piston causes the downward flow of oil. Consider a force F applied to the piston which causes it to move downwards and, therefore, the oil flow is upward through the clearance space. If the clearance space is small, then the piston and cylinder may be considered as two flat plates of breadth equal to the circumferential length of the piston πD, length L and distance apart equal to the clearance c, through which the oil flows. Thus Eq. 13.34 may be applied to this case also according to which

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2$$

Applying the boundary conditions:

when $y = 0,\ v = 0 \quad \therefore\ C_2 = 0$

when $y = c,\ v = -V_p$

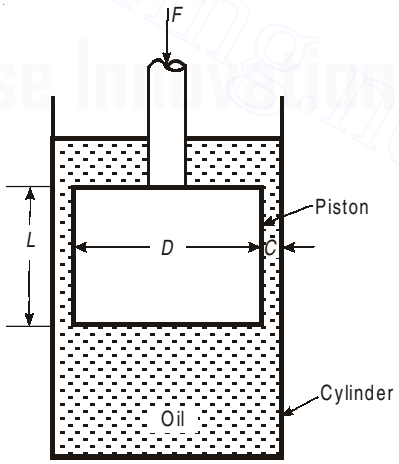


Figure 13.13 *Dash pot–a simplified sketch*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(where V_p is the uniform downward velocity of the piston and hence considered negative).

$$\therefore \quad C_1 = -\frac{V_p}{c} - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{c}{2}$$

$$\therefore \quad v = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy) - \frac{V_p y}{c} \qquad ...(13.76)$$

Considering the flow through an elemental strip of breadth πD and thickness dy, then

$$dQ = v(\pi D)\, dy$$

$$= \pi D\left[\frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy) - V_p\frac{y}{c}\right] dy$$

or $$Q = \pi D\int_0^C \left[\frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy) - V_p\frac{y}{c}\right] dy$$

$$Q = \pi D\left[-\left(\frac{\partial p}{\partial x}\right)\frac{c^3}{12\mu} - \frac{V_p c}{2}\right] \qquad ...(13.77)$$

Equation 13.77 represents the steady rate of flow of oil through the clearance space, which must be exactly equal to the rate at which oil is being displaced by the piston, i.e., $(\pi D^2/4)\, V_p$.

Thus

$$\pi D\left[-\left(\frac{\partial p}{\partial x}\right)\frac{c^3}{12\mu} - \frac{V_p c}{2}\right] = \frac{\pi D^2}{4} V_p$$

or $$V_p\left(\frac{D}{2} + c\right) = -\frac{\partial p}{\partial x}\frac{c^3}{6\mu} \qquad ...(13.78)$$

But $\frac{\partial p}{\partial x} = -\frac{p}{L}$, where p is the pressure drop over the length of the piston.

$$\therefore \quad p = \frac{6\mu V_p L}{c^3}\left(\frac{D}{2} + c\right) \qquad ...(13.79)$$

Due to this pressure difference an upward force will be exerted on the piston which will be opposing its downward motion, and it is given by

$$F_v = p\frac{\pi D^2}{4} = \pi\mu V_p L\left[\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)^2\right] \qquad ...(13.80)$$

In addition to this force there will be shear force (i.e., skin friction drag) exerted by the oil on the piston which will also be resisting its downward motion. The shear stress at the piston wall is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=c}$$

From Eq. 13.34

$$\frac{\partial v}{\partial y} = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right) y + C_1$$

As obtained above

$$C_1 = -\frac{V_p}{c} - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{c}{2}$$

$$\therefore \quad \frac{\partial v}{\partial y} = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right) y - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{c}{2} - \frac{V_p}{c}$$

when $y = c$

$$\left(\mu\frac{\partial v}{\partial y}\right)_{y=c} = \frac{\partial p}{\partial x}\frac{c}{2} - \frac{\mu V_p}{c}$$

$$\therefore \quad \tau_0 = \frac{\partial p}{\partial x}\frac{c}{2} - \frac{\mu V_p}{c}$$

But from Eq. 13.78

$$\frac{\partial p}{\partial x} = -\frac{6\mu V_p}{c^3}\left(\frac{D}{2} + c\right)$$

$$\therefore \quad \tau_0 = -\left[\frac{3\mu V_p}{c^2}\left(\frac{D}{2} + c\right) + \frac{\mu V_p}{c}\right]$$

Thus shear force resisting the motion of the piston is given by

$$F_s = (\pi DL\tau_0) = \pi DL\left[\frac{3\mu V_p}{c^2}\left(\frac{D}{2} + c\right) + \frac{\mu V_p}{c}\right]$$

Note: The negative sign of shear stress has been omitted since it merely denotes that the shear force opposes the motion of the piston

$$\therefore \quad F_s = \pi\mu V_p L\left[\frac{3}{2}\left(\frac{D}{c}\right)^2 + 4\left(\frac{D}{c}\right)\right] \qquad ...(13.81)$$

Therefore, the total force required to move the piston at a uniform velocity down through the oil is

$$F = F_v + F_s$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F = \pi\mu V_p L\left[\left\{\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)^2\right\} + \left\{\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{4D}{c}\right\}\right] \quad ...(13.82)$$

If V_p is negligible as compared to the average velocity of oil through the clearance space, then from Eq. 13.76

$$v = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy)$$

and using the same analysis as indicated above

$$p = \frac{3\mu V_p LD}{c^3}$$

and

$$\tau_0 = \frac{3}{2}\frac{\mu V_p D}{c^2}$$

Thus

$$F = F_v + F_s$$

$$= \pi\mu V_p L\left[\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)^2\right] \quad \ldots \quad (13.83)$$

When the shear force (or skin friction drag) on the piston wall is negligible as compared to the force due to pressure difference then

$$F = F_v = \frac{3}{4}\pi\mu V_p L\left(\frac{D}{c}\right)^3 \quad \ldots \quad (13.84)$$

Equation 13.82 gives the most general expression for the force resisting the downward motion of the piston and Eqs. 13.83 and 13.84 are only the simplified forms of Eq. 13.82. The criterion which controls the use of either of the simplifications given by the Eqs. 13.83 and 13.84 is the value of the (D/c). Thus larger is the value of (D/c) the smaller will be the effect of the piston velocity and the piston shear.

In the above experssions the downward force F on the piston includes the self-weight of the piston. However, since the piston is completely submereged in the oil, the net weight of the piston should be considered, which will be equal to the weight of the piston minus the weight of the oil displaced by it (or the buoyant force acting upwards on the piston). Further if the piston is moving downwards under its own weight only, then the value of F in the above equations will be equal to the net weight of the piston as indicated above.

13.13 MEASUREMENT OF VISCOSITY—VISCOMETERS

The various methods adopted for the measurement of viscosity of a fluid are based on the use of the following three principles:

(a) Newton's law of viscosity; (b) Hagen-Poiseuille equation; (c) Stokes' law.

(a) *Coaxial cylinder viscometer*: In this viscometer Newton's law of viscosity is employed to measure the viscosity of a fluid. In Newton's equation of viscosity, by measurement of the velocity gradient $(\partial v/$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

∂y) and the shear stress τ, the dynamic viscosity μ can be computed.

As shown in Fig. 13.14, this viscometer consists of two coaxial cylinders, one of radius R_1, placed inside the other of radius R_2, such that a very small space $(R_2 - R_1)$ is left in between the two. The space is filled with the liquid whose viscosity is to be determined. The inner cylinder is suspended by a torsion wire and it is held stationary while the outer cylinder can rotate at a known constant speed. When the outer cylinder rotates, the torque generated by such rotation is transmitted by the thin liquid film to the inner stationary cylinder, which causes rotation of the torsion wire. The rotation of the wire can be measured by means of a circular dial attached to the wire and a fixed pointer. From the previously obtained calibration curve between the torque and the rotation of the torsion wire, the torque exerted on the wire and hence on the inner cylinder, corresponding to the measured rotation of the wire can be known. By knowing the torque on the inner cylinder, shear stress τ can be computed.

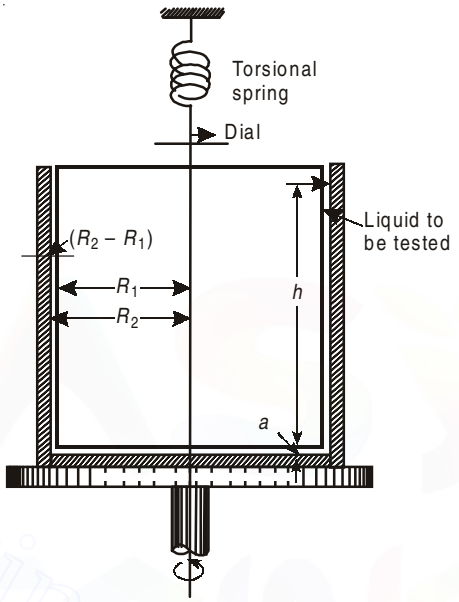


Figure 13.14 *Coaxial cylinder viscometer*

Let T be the torque exerted on the inner cylinder, obtained as indicated above and h be the depth of the liquid in the cylinder. Now if the clerance a between the two cylinders at their bottom is large, then the torque contribution from the bottom may be neglected. The torque$T$ will then be contributed only from the sides, being equal to T_1, and it may be expressed in terms of the shear stress τ as follows:

$$T = T_1 = (2\pi R_1 h)\tau R_1$$

$$= (2\pi R_1^2 h)\tau$$

or

$$\tau = \frac{T_1}{2\pi R_1^2 h}$$

The velocity gradient may be determined as mentioned below.

When the outer cylinder rotates at a speed N rev./min, the velocity of the liquid at the surface of the outer cylinder will be $(2\pi R_2 N/60)$. The inner cylinder being stationary the velocity of the liquid at the surface of the inner cylinder will be zero. Since the space between the cylinders in very small the velocity of the liquid in the space may be assumed to vary linearly from zero at the surface of the inner cylinder to a maximum at the surface of the outer cylinder. Hence the velocity gradient may be expressed as

$$\frac{\partial v}{\partial y} = \frac{2\pi R_2 N}{60(R_2 - R_1)}$$

By substituting the value of τ and $(\partial v/\partial y)$ in Newton's equation of viscosity and solving for μ, we obtain

$$\mu = \frac{15T_1(R_2 - R_1)}{\pi^2 R_1^2 R_2 h N} \quad \ldots \quad (13.85)$$

However, if the clearance a is so small that the torque exerted from the bottom is appreciable then the total torque T exerted on the inner cylinder will be contributed by both bottom as well as sides. Thus if T_2 is the torque contribution from the bottom, then by performing integration over whole area of the bottom of the inner cylinder, we obtain

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$T_2 = \left(\frac{\mu\pi^2}{a}\right)\frac{N}{60}R_1^4$$

The total torque T may be expressed as

$$T = T_1 + T_2 = \frac{\mu\pi^2 R_1^2 R_2 hN}{15(R_2 - R_1)} + \frac{\mu\pi^2 N R_1^4}{60a}$$

or
$$T = \frac{\mu\pi^2 N R_1^2}{15}\left(\frac{R_2 h}{R_2 - R_1} + \frac{R_1^2}{4a}\right) \quad \ldots \quad (13.86)$$

Since R_1, R_2, h and a are constants for a viscometer, the torque may be expressed as

$$T = C\mu N$$

in which C is defined as a constant for the viscometer. Solving for μ

$$\mu = \frac{T}{CN} \quad \ldots \quad (13.87)$$

Thus Eq. 13.86 or 13.87 may be used to determine μ since execpt μ all other quantities are known.

(b) *Capillary tube viscometer*: This viscometer involves use of the Hagen–Poiseuille equation. As shown in Fig. 13.15 it consists of a capillary tube of cross-sectional area a and length L attached horizontally very close to the bottom of a resevoir of area A, filled with liquid whose viscosity μ is to be determined. The tube is allowed to discharge freely in the atmosphere. As the liquid flows through the tube, in the absence of any inflow into the tank, the level of the liquid surface in the tank will be lowered.

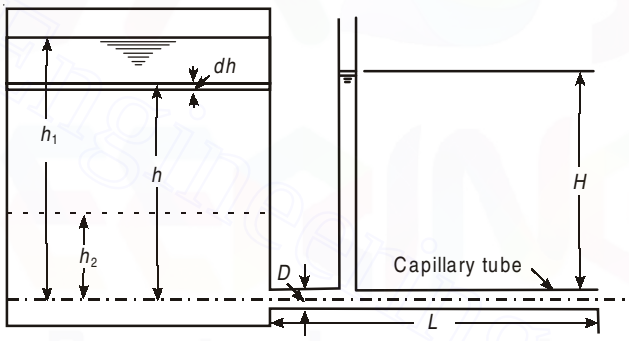


Figure 13.15 *Capillary tube viscometer*

Let at any instant the liquid surface be at a height h above the centre of the tube causing a disharge q and let the surface fall by a height dh in time dt. Then, we have

$$-A\,dh = q\,dt = aV\,dt$$

where V is the velocity of flow in the tube under the head h. Since the flow in the tube is laminar, it may be assumed that the head h is utilised completely to overcome the viscous resistance in the tube, (i.e., the velocity head possessed by the liquid leaving the tube is neglected). Then by Hagen–Poiseuille equation, we have

$$h = \frac{32\mu VL}{wD^2}; \quad \text{or} \quad V = \frac{whD^2}{32\mu L}$$

where D is the diameter of the tube and w is the specific weight of the liquid. By substituting for V, we get

$$-A\,dh = \frac{awD^2 h}{32\mu L}\,dt$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$dt = \left(-\frac{32\mu LA}{awD^2}\right)\frac{dh}{h}$$

The time t required to lower the liquid surface from a height h_1 to h_2 above the centre of the tube may be obtained by integrating both sides of the above expression. Thus, we have

$$\int dt = -\frac{32\mu LA}{awD^2}\int_{h_1}^{h_2}\frac{dh}{h}$$

or $$t = \frac{32\mu LA}{awD^2}\log_e\left(\frac{h_1}{h_2}\right)$$

$$\mu = \frac{tawD^2}{32AL[\log_e(h_1/h_2)]} \quad \ldots (13.88)$$

Alternatively by providing suitable arrangement for inflow into the tank, a constant head may be maintained over the capillary tube and the liquid is allowed to flow through the capillary tube under the constant head. Both the head H and discharge Q are measured and using Hagen–Poiseuille eqation the viscosity of the liquid may be determined as

$$\mu = \frac{\pi wHD^4}{128QL} \quad \ldots (13.89)$$

However, it is difficult to measure the viscosity by this type of viscometer. As such certain modified forms of the capillary tube viscometer have been developed for the measurement of viscosity of liquids. Two common viscometers of this type are Saybolt and Redwood viscometers, both of which involve the principle of vertical gravity flow through capillary tubes.

1. Saybolt Viscometer

It consists of a vertical cylindrical chamber surrounded by a constant temperature bath as shown in Fig. 13.16. A short capillary tube is attached vertically at the bottom of the cylindrical chamber. The diameter of the capillary tube varies from 1.75 mm. to 1.78 mm. and its length varies from 12.15 mm to 12.35 mm. The cylindrical chamber is completely filled with the liquid whose viscocity is to be determined and the outlet is plugged. The surrounding bath will maintain the liquid under test at a required constant temperature. The plug is then removed and the time in seconds is measured for the flow of 0.06 litre or 60 cm^3 of the liquid through the capillary tube under a falling head. The time so obtained is designated as Saybolt reading. This viscometer measures kinematic viscosity of the liquid. Assuming the flow through the tube to be laminar, Hagen–Poiseuille equation can be used. Rearranging the Hagen –Poiseuille equation for viscosity μ, we obtain

$$\mu = \frac{wh\pi D^4}{128QL} = \frac{\rho gh\pi D^4}{128QL}$$

in which h represents the head under which the liquid flows through the tube and Q is the volume of liquid flowing per unit time. If t is the time in seconds recorded for the flows of constant volume (viz., 60 cm^3) of liquid, then Q = (vol./t).Thus the above expression may be written as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{\mu}{\rho} = \upsilon = \frac{gh\pi D^4 t}{128(vol.)L}$$

In this expression although the head h varies during the test, its variation is over the same range for all liquids, since a constant volume of liquid is allowed to flow for each test. As such all the terms on the right hand side except t may be considerd as constant and may be grouped together to provide a constant of a particular viscometer. Let C_1 be the constant of the viscometer, then we get

$$\upsilon = C_1 t \qquad \ldots \quad (13.90)$$

Equation 13.90 shows that the kinematic viscosity varies directly as the time t. In this case since the capillary tube is quite short, steady laminar flow conditions will usually not exist in the tube. Thus a correction factor will have to be incorporated in Eq. 13.90 in order to compensate for the use of Hagen-Poisuille equation in the analysis and to obtain a correct value of kinematic viscosity of the liquid. The correction is of the form (C_2/t) hence the kinematic viscosity becomes

$$\upsilon = C_1 t + \frac{C_2}{t} \qquad \ldots \quad (13.91)$$

The approximate relationship between kinematic viscosity and time t for a Saybolt Universal viscometer is expressed by

$$\upsilon = 0.0022\, t - \frac{1.80}{t} \qquad \ldots \quad (13.92)$$

where υ is in stokes and t is in seconds.

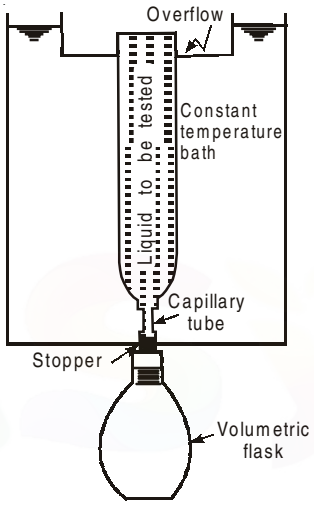


Figure 13.16 *Sybolt Viscometer*

2. Redwood Viscometer

It consists of vertical cylinder provided with an orifice in the centre of its base. The cylinder is surrounded by a water bath which can maintain the liquid to be tested at any required temperature by means of an immersed electric heater as shown in Fig. 13.17. The cylinder is 47.625 mm in diameter and 88.90 mm deep. The orifice is 1.70 mm diameter and 12 mm in length. The cylinder is filled upto a fixed height with the liquid whose viscosity is to be determined, and is heated by the water bath to the required temperature. The orifice is then opened and the time taken for 0.05 litre or 50 cm^3 of the flow is noted. The viscosity of the liquid is proportional to this time. Thus an expression similar to Eq. 13.90 can be obtained for this viscometer as well. It may, however, be pointed out that this viscometer does not provide a direct value of the viscosity of any liquid. It only facilitates a comparison of viscosities of different liquids. The exact value of viscosity of any liquid can, however, be obtained by comparison with the value of time t for liquids of known viscosity.

(c) *Falling sphere viscometer.* This method of measuring viscosity of a liquid is based on Stokes' Law. As illustrated in Fig. 13.18, it consists of a long vertical container either cylindrical or prismatic in shape. The container is filled with the liquid whose viscosity is to be determined and is surrounded by a constant temperature bath. A small spherical ball of steel is allowed to fall vertically through the liquid. The falling ball as it is dropped will at first accelerate, but the resistance to its motion increases

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

with its velocity until the resistance to its motion is counter-balanced by the pull of gravity on the ball. When this condition is reached the ball falls with a constant velocity. The time t taken by the ball to fall

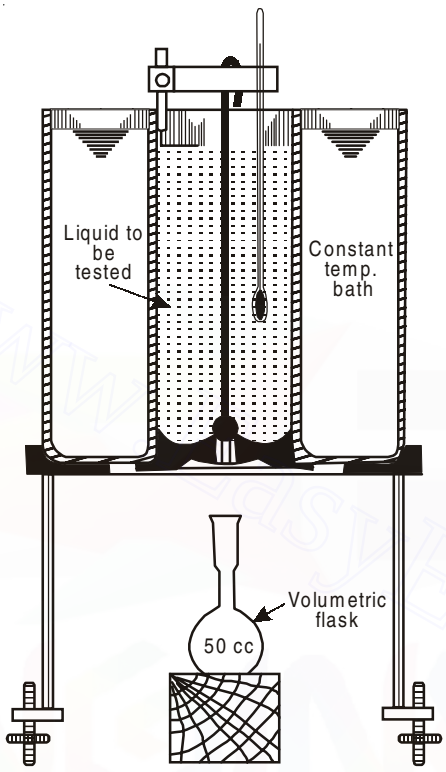


Figure 13.17 *Redwood viscometer*

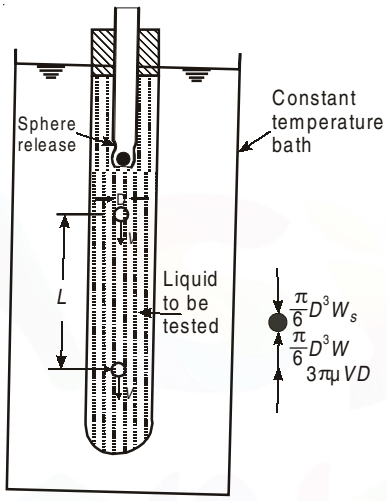


Figure 13.18 *Falling sphere viscometer*

through a known distance L, after a constant velocity has been attained by it, is noted. Then the fall velocity V is known, since $V = (L/t)$.

As indicated in Section 13.10 the viscosity μ may be determined using Eq. 13.62 as

$$\mu = \frac{D^2}{18V}(w_s - w) \qquad \ldots \quad (13.93)$$

where w_s and w are specific weights of the ball and the liquid respectively. This method will, however give accurate value of viscosity only if Stokes' Law is applicable.

ILLUSTRATIVE EXAMPLES

Example 13.1. *Lubricating oil of specific gravity 0.82 and dynamic viscosity 12.066 × 10^{-2} N.s/m^2 [1.23 ×10^{-2} kg (f)–s/m^2] is pumped at a rate of 0.02 m^3/s through a 0.15 m diameter 300 m long pipe. Calculate the pressure drop, average shear stress at the wall of the pipe and the power reuired to maintain flow (a) if the pipe is horizontal; (b) if the pipe is inclined at 15 degrees with the horizontal and the flow is (i) in the upward direction, (ii) in the downward direction. Also determine the slope of the pipe and the directon of flow so that the pressure gradient along the pipe is zero.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

$$V = \frac{0.02}{\frac{\pi}{4}(0.15)^2} = 1.132 \text{ m/s}$$

(a) For horizontal pipe from Hagen-Poiseuille equation, we have

$$(p_1 - p_2) = \frac{32\mu VL}{D^2}$$

SI units

$$(P_1 - P_2) = \frac{32 \times 12.066 \times 10^{-2} \times 1.132 \times 300}{(0.15)^2}$$

$$= 58\,277.17 \text{ N/m}^2$$

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= \frac{58277.17}{300} \times \frac{0.15}{2 \times 2} = 7.285 \text{ N/m}^2$$

Power required to maintain the flow is

$$P = Q\,(p_1 - p_2)$$

$$= 0.02 \times 58\,277.17 = 1166 \text{ W} = 1.166 \text{ kW}$$

Metric units

$$(p_1 - p_2) = \frac{32 \times 1.23 \times 10^{-2} \times 1.132 \times 300}{(0.15)^2}$$

$$= 5\,941 \text{ kg (f)/m}^2$$

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= \frac{5941}{300} \times \frac{0.15}{2 \times 2} = 0.743 \text{ kg(f)/m}^2$$

Power required to maintain the flow is

$$p = \frac{Q(p_1 - p_2)}{75}$$

$$= \frac{0.02 \times 5941}{75} = 1.584 \text{ h.p.}$$

(b) (i) For inclined pipe with flow in upward direction, we have

$$w(h_1 - h_2) = \frac{32\mu VL}{D^2}$$

SI units

$$(h_1 - h_2) = \frac{32 \times 12.066 \times 10^{-2} \times 1.132 \times 300}{(9810 \times 0.082) \times (0.15)^2} = 7.245 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\left(\frac{p_1}{w}+0\right)-\left(\frac{p_2}{w}+Z_2\right) = 7.245$$

Since $\frac{Z_2}{300} = \sin 15° = 0.2588$; $Z_2 = 77.64$ m

$\therefore$ $(p_1 - p_2) = (9\,810 \times 0.82)\,(7.245 + 77.64)$

$= 682\,832\ \text{N/m}^2 = 682.832\ \text{kN/m}^2$

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(9810\times0.82)\times7.245}{300}\times\frac{0.15}{2\times2} = 7.285\ \text{N/m}^2$$

Power required to maintain the flow is

$P = Q\,(p_1 - p_2)$

$= 0.02 \times 682\,832 = 13\,657\ \text{W} = 13.657\ \text{kW}$

Metric untis

$$(h_1 - h_2) = \frac{32\times1.23\times10^{-2}\times1.132\times300}{(1000\times0.82)\times(0.15)^2} = 7.245\ \text{m}$$

or $$\left(\frac{p_1}{w}+0\right)-\left(\frac{p_2}{w}+Z_2\right) = 7.245$$

Since $\frac{Z_2}{300} = \sin 15° = 0.2588$; $Z_2 = 77.64$ m

$\therefore$ $(p_1 - p_2) = (1000 \times 0.82)\,(7.245 + 77.64)$

$= 6.96 \times 104\ \text{kg (f)/m}^2$

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(1000\times0.82)\times7.245}{300}\times\frac{0.15}{2\times2}$$

$= 0.743\ \text{kg (f)/m}^2$

Power required to maintains the flow is

$$P = \frac{Q(p_1 - p_2)}{75}$$

$$= \frac{0.02\times(6.96\times10^4)}{75} = 18.56\ \text{h.p.}$$

(ii) For inclined pipe with flow in downward direction, we have

$$w\,(h_1 - h_2) = \frac{32\mu VL}{D^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SI units

$$(h_1 - h_2) = \frac{32 \times 12.066.10^{-2} \times 1.132 \times 300}{(9810 \times 0.82) \times (0.15)^2} = 7.245 \text{ m}$$

or $$\left(\frac{p_1}{w} + Z_1\right)\left(\frac{p_2}{w} + 0\right) = 7.245$$

Since $$\frac{Z_1}{300} = \sin 15° = 0.2588; Z_1 = 77.64 \text{ m}$$

$\therefore$ $$(p_1 - p_2) = (9810 \times 0.82)\ (7.245 - 77.64)$$
$$= -566\ 271 \text{N/m}^2 = -566.271 \text{ kN/m}^2$$

i.e., in this case the pressure increases in the direction of flow, or there is positive pressure gradient.

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(9810 \times 0.82) \times 7.245}{300} \times \frac{0.15}{2 \times 2} = 7.285 \text{ N/m}^2$$

In this case the resistance to flow is compensated by the excessive downward slope of the pipe and hence no external power is required to maintain the flow. Moreover, in this case the flow will have to be regulated by means of a regulating value to maintain the given flow rate.

For pressure gradient along the pipe to be zero, $p_1 = p_2$.

Then from the two above noted cases of inclination of the pipe we have either, $-Z_2 = 7.245$ m, or $Z_1 = 7.245$ m, from which it may be concluded that point 1 is higher than point 2, so that the flow is in the downward direction. The slope required to be provided for the pipe in this case is given by

$$\sin \theta = \frac{Z_1}{300} = \frac{7.245}{300} = 0.0242$$

$\therefore$ $$\theta = 1°23'$$

Example 13.2. *Water flows at a steady mean velocity of 1.5 m/s through a 50 mm diameter pipe slopping upwards at 45° to the horizontal. At a section some distance downstream of the inlet the pressure is 700 k Pa and at a section 30 m further along the pipe the pressure is 462 k Pa. Determine the average shear stress at the wall of the pipe and at a radius of 10 mm.*

Solution

Assuming datum to be passing through the lower point,we have

$$(h_1 - h_2) = \left(\frac{p_1}{w} + 0\right) - \left(\frac{p_2}{w} + Z_2\right)$$

$$= \left(\frac{700 \times 10^3}{9810}\right) - \left(\frac{462 \times 10^3}{9810} + \frac{30}{\sqrt{2}}\right)$$

$$= (71.356 - 68.308) = 3.048 \text{ m}$$

Average shear stress at the wall of the pipe is

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{9810 \times 3.048}{30} \times \frac{0.05}{2 \times 2}$$

$$= 12.459 \text{ N/m}^2 = 12.459 \text{ Pa}$$

Average shear stress at a radius of 10 mm is

$$\tau = \left(-\frac{\partial h}{\partial x}\right)\frac{r}{2}$$

$$= \frac{9810 \times 3.048}{30} \times \frac{0.01}{2}$$

$$= 4.983 \text{ N/m}^2 = 4.983 \text{ Pa}$$

Example 13.3. *For Laminar flow of an oil having dynamic viscosity μ = 1.766 Pa.s in a 0.3 m diameter pipe, the velocity distribution is parabolic with a maximum point velocity of 3 m/s at the centre of the pipe. Calculate the shearing stresses at the pipe wall and within the fluid 50 mm from the pipe wall.*

Solution

From Eq. 13.15, we have

$$V = \frac{1}{2} v_{max} = \frac{3}{2} = 1.5, \text{ m/s}$$

Also from Eq. 13.17, we have

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{p_1 - p_2}{L} = \frac{32 \mu V}{D^2}$$

Thus $$\left(-\frac{\partial p}{\partial x}\right) = \frac{32 \times 1.766 \times 1.5}{(0.3)^2} = 941.87 \text{ Pa/m}$$

The shear stress at the pipe wall is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(941.87 \times 0.3)}{2 \times 2} = 70.64 \text{ Pa}$$

The shear stress at 50 mm from the pipe wall is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= 941.87 \times \frac{(0.15 - 0.05)}{2} = 47.09 \text{ Pa}$$

Example 13.4. *Two parallel plates kept 0.1 m apart have laminar flow of oil between them with a maximum velocity of 1.5 m/s. Calculate the discharge per metre width, the shear stress at the plates, the difference in pressure in pascals between two points 20 m apart, the velocity gradient at the plates and velocity at 0.02 m from the plate. Take viscosity of oil to be 2.453 N.s/m².*

Solution

In this case the mean velocity of flow *V* is equal to two-thirds of the maximum velocity.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad V = \frac{2}{3}(1.5) = 1.0 \text{ m/s}$$

The discharge q per metre width the plate is given by

$$q = VB = (1.0 \times 0.1) = 0.1 \text{m}^3/\text{s per m}$$

From Eq. 13.38, we have

$$V = \frac{B^2}{12\mu}\left(-\frac{\partial p}{\partial x}\right)$$

$$\therefore \quad \left(-\frac{\partial p}{\partial x}\right) = \frac{12\mu V}{B^2}$$

$$= \frac{12 \times 2.453 \times 1.0}{(0.1)^2} = 2943.6 \text{ N/m}^2 \text{ per m}$$

The shear stress at the plates is given by Eq. 13.43 as

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2} = \frac{2943.6 \times 0.1}{2} = 147.18 \text{ N/m}^2$$

The pressure difference between the two points is given by Eq. 13.40 as

$$(p_1 - p_2) = \frac{12\mu VL}{B^2}$$

$$= \frac{12 \times 2.453 \times 1.0 \times 20}{(0.1)^2}$$

$$= 58872 \text{ N/m}^2 = 58.872 \text{kN/m}^2$$

The shear stress at plates is also given by

$$\tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=0}$$

As such the velocity gradient at the plates is given by

$$\left(\frac{\partial v}{\partial y}\right)_{y=0} = \frac{\tau_0}{\mu} = \frac{147.18}{2.453} = 60 \ s^{-1}$$

The velocity v at a distance of 0.02 m from the plate is given by Eq. 13.35 as

$$v = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)\left(By - y^2\right)$$

$$= \frac{1}{2 \times 2.453} \times (2943.6) \times \left[(0.1 \times 0.02) - (0.02)^2\right]$$

$$= 0.96 \text{ m/s}$$

Example 13.5. *Oil of viscosity 0.1 Pa.s and specific gravity 0.90, flows through a horizontal pipe of 25 mm diameter. If the pressure drop per metre length of the pipe is 12 k Pa, determine (a) the rate of flow in N/min; (b) the shear stress at the pipe wall; (c) the Reynolds number of the flow; and (d) the power required per 50 m length of pipe to maintain the flow.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

From Eq. 13.17, we have

$$V = \frac{(p_1 - p_2)D^2}{32\mu L} = \frac{12\times10^3\times(0.025)^2}{32\times0.1\times1} = 2.34 \text{ m/s}$$

(a) Rate of flow in N/min is given by

$$Q = \frac{\pi}{4}(0.025)^2\times2.34\times(9810\times0.90)\times60$$
$$= 608.5 \text{ N/min}$$

(b) From Eq. 13.4, we have

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} = (12\times10^3)\times\frac{0.025}{2\times2} = 75 \text{ N/m}^2$$

(c) Reynolds number

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(1000\times0.90)\times2.34\times0.025}{0.1} = 526.5$$

(d) From Eq. 13.24, we have

Power $P = Q(p_1 - p_2)$

$$= \frac{\pi}{4}(0.025)^2\times2.34\times(12\times10^3\times50) = 689.2 \text{ W}$$

Example 13.6. *A total of 12 litres per second of oil is pumped through two pipes in parallel, one 10 cm in diameter and the other 12 cm in diameter, both pipes being 1000 metres long. The specific gravity of the oil is 0.97 and the kinematic viscosity 9 cm^2 per second. Calculate the flow rate through each pipe and the horse-power of the pump.*

Solution

Assuming the flow in each pipe to be laminar, the head loss is given by Eq.13.18 as

$$h_f = \frac{32\mu VL}{wD^2} = \frac{128\mu QL}{w\pi D^4}$$

Let Q_1 and Q_2 be the flow rates through each of the two pipes, which being in parallel, we have

$$h_{f_1} = h_{f_2}$$

or
$$\frac{128\mu_1 Q_1 L_1}{w_1\pi D^2} = \frac{128\mu_2 Q_2 L_2}{w_2\pi D^2}$$

$\mu_1 = \mu_2 = (\rho\upsilon) = (0.97\times1\times9)$ g/cm.s = 8.73 g/cm.s;

$L_1 = L_2 = 1000$ m; $D_1 = 10$ cm = 0.10 m; $D_2 = 12$ cm = 0.12 m;

and $w_1 = w_2 = 0.97\times9810 \text{ N/m}^3 = 9515.7 \text{ N/m}^3$

By substituting, we get

$$\frac{128\times8.73\times Q_1\times1000}{9515.7\times\pi\times(0.10)^4} = \frac{128\times8.73\times Q_2\times1000}{9515.7\times\pi\times(0.12)^4}$$

or
$$\frac{Q_1}{(0.10)^4} = \frac{Q_2}{(0.12)^4}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{Q_1}{Q_2} = \left(\frac{0.10}{0.12}\right)^4 = 0.4823$$

or $$Q_1 = 0.4823\, Q_2$$

But $$Q_1 + Q_2 = 12\ l/\text{s}$$

$\therefore$ $$Q_1 = 3.904\ l/\text{s};\ \text{and}\ Q_2 = 8.096\ l/\text{s}$$

The Reynolds number is given by

$$\text{Re} = \frac{VD}{\upsilon}$$

For pipe of diameter 10 cm,

$$V = \frac{3.904\times10^3}{(\pi/4)(10)^2} = 49.71\ \text{cm/s}$$

$\therefore$ $$\text{Re} = \frac{49.71\times10}{9} = 55$$

Similarly for pipe of diameter 12 cm

$$V = \frac{8.096\times10^3}{(\pi/4)(12)^2} = 71.58\ \text{cm/s}$$

$\therefore$ $$\text{Re} = \frac{71.58\times12}{9} = 95$$

Thus the assumption of laminar flow in both the pipes is correct.

The head loss $$h_f = \frac{128\,\mu QL}{w\pi D^4}$$

$$= \frac{128\,\mu QL}{\rho g\pi D^4}$$

$$= \frac{128\,\upsilon QL}{g\pi D^4}$$

$\upsilon = 9\ \text{cm}^2/\text{s}$; $Q = 3.904 \times 10^3\ \text{cm}^3/\text{s}$; $L = 1000 \times 100 = 10^5$ cm;

$g = 981\ \text{cm/s}^2$; and $D = 10$ cm

Thus by substitution, we get

$$h_f = \frac{128\times9\times3.904\times10^3\times10^5}{981\times\pi\times(10)^4}$$

$$= 1.46 \times 10^4\ \text{cm} = 146\ \text{m}$$

Power of the pump is

$$P = \frac{wQh_f}{75}\ \text{metric horse-power}$$

$w = 0.97 \times 1000 = 970\ \text{kg(f)/m}^3$

$Q = 12\ l/\text{s} = 12 \times 10^{-3}\ \text{m}^3/\text{s}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad P = \frac{970 \times 12 \times 10^{-3} \times 146}{75}$$

$$= 22.66 \text{ metric horse-power}$$

$$= 22.66 \times 735.5 \text{ W}$$

$$= 16.7 \times 10^3 \text{ W}$$

$$= 16.7 \text{ kW}$$

Example 13.7. *Determine the depth d below the free surface at which the point velocity is equal to the average velocity of flow for a uniform laminar flow with a free surface.*

Solution

From Eqs. 13.52 and 13.54, equating the point velocity and the average velocity, we have

$$\frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy - \frac{y^2}{2}\right) = \frac{w}{3\mu}\left(-\frac{\partial z}{\partial x}\right)D^2$$

or $$6Dy - 3y^2 = 2D^2$$

or $$3y^2 - 6Dy + 2D^2 = 0$$

or $$y = 0.423D$$

Since the distance y in the above expression is measured from the bed of the channel, the required depth

$$d = D - 0.423D = 0.577D$$

Example 13.8. *A horizontal column of sand 0.6 m long and 75 mm in diameter has water flowing through it. If the pressure drop across the column of sand is 20.601 k Pa and the discharge is 4.55 litres/min, what is the value of transmission constant?*

Solution

From Eq. 13.60, we have

$$V = Ki$$

$$V = \frac{4.55 \times 10^6}{60 \times \frac{\pi}{4} \times (75)^2} = 17.165 \text{ mm/s}$$

$$i = \frac{h_f}{L} = \frac{20.601 \times 10^3}{9810 \times 0.6} = 3.5$$

$$K = \frac{V}{i} = \frac{17.165}{3.5} = 4.904 \text{ mm/s}$$

i.e., the transmission constant $K = 4.904$ mm/s

Example 13.9. *In a certain capillary tube viscometer the reservoir is of 100 mm diameter and a slender tube fixed horizontally at the bottom of the reservoir is 1 mm in diameter and 300 mm long. Determine the viscosity of a liquid of specific gravity 0.82 if the level in the reservoir is 0.1265 m at the beginning of the test and 0.12 m at the end of 2 minutes.*

Solution

From Eq. 13.88, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\mu = \frac{tawD^2}{32AL[\log_e(h_1/h_1)]}$$

Thus by substitution, we get

$$\mu = \frac{2\times 60\times\frac{\pi}{4}(0.001)^2\times 9810\times 0.82\times(0.001)^2}{32\times\frac{\pi}{4}(0.1)^2\times 0.3\times\left(\log_e\frac{0.1265}{0.12}\right)}$$

$$= 1.906\times 10^{-4}\ \text{N.s/m}^2$$

$$= 1.906\times 10^{-3}\ \text{poise}$$

Example 13.10. *0.02 litres of a liquid (specific gravity 0.9) flow in 40 seconds through a pipe 0.25 m long and 1.5 mm diameter emerging from the bottom of a tank holding the liquid maintained at a constant level of 0.35 m above the outlet end of the pipe discharging freely. Compute the dynamic viscosity of the liquid.*

Solution

From Eq. 13.89, we have

$$\mu = \frac{\pi wHD^4}{128QL}$$

$$w = (9\,810\times 0.9)\ \text{N/m}^3;\quad H = 0.35\ \text{m};\quad D = (1.5\times 10^{-3})\ \text{m}$$

$$Q = \frac{(0.02\times 10^{-3})}{40}\ \text{m}^3/\text{s};\quad L = 0.25\ \text{m}$$

Thus by substitution, we get

$$\mu = \frac{\pi\times(9810\times 0.9)\times 0.35\times(1.5\times 10^{-3})^4\times 40}{128\times(0.02\times 10^{-3})\times 0.25}$$

or $\mu = 3.072\times 10^{-3}\ \text{N.s/m}^2$

or $\mu = 3.072\times 10^{-2}\ \text{poise}$

Example 13.11. *A small glass sphere of specific gravity 2.65 has a diameter of 3 mm. If the sphere falls 0.6 m in 30 seconds through a liquid of specific gravity 0.85, what is the absolute viscosity of the liquid?*

Solution

From Eq. 13.93, we have

$$\mu = \frac{D^2}{18V}(w_s - w)$$

$$V = \frac{0.6}{30} = 0.02\ \text{m/s}$$

Thus by substitution, we have

$$\mu = \frac{(0.003)^2}{18\times(0.02)}\times 9810\ (2.65 - 0.85)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= 0.441 \text{ N.s/m}^2$

$= 0.0441$ poise

Example 13.12. *The space between two parallel plates distance h apart is filled with incompressible fluid of viscosity μ. The bottom plate moves at velocity $(-V_1)$ and the top plate moves at velocity $+V_2$. Obtain an expression for velocity distribution (v/V_2) for this case.*

Solution

From Eq. 13.34, we have

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2$$

In the present case the boundary conditions are as follows:

at $y = 0, v = -V_1$ and at $y = h, v = V_2$

Thus $C_2 = -V_1$

and $$C_1 = \frac{V_2}{h} + \frac{V_1}{h} - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{h}{2}$$

$\therefore$ By substitution, we get

$$v = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(hy - y^2) + \left(\frac{V_2 y}{h} + \frac{V_1 y}{h} - V_1\right)$$

or $$\frac{v}{V_2} = \frac{y}{h} - \frac{V_1}{V_2}\left(1 - \frac{y}{h}\right) + \left[\frac{h^2}{2\mu V_2}\left(-\frac{\partial p}{\partial x}\right)\frac{y}{h}\left(1 - \frac{y}{h}\right)\right]$$

Example 13.13. *A dash-pot has a cylinder 70 mm diameter in which slides a piston 80 mm long having a radial clearance of 1 mm. The cylinder is filled with oil of viscosity 1 poise. Determine the uniform velocity of the piston when the force acting on it is 180 N. What percentage error is involved if the shear force on the piston wall is neglected?*

Solution

From Eq. 13.82, we have

$$F = \pi \mu V_p L \left[\left\{\left(\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)\right)^2\right\} + \left\{\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{4D}{c}\right\}\right]$$

$$\frac{D}{c} = 68; \left(\frac{D}{c}\right)^2 = (68)^2 = 46.24 \times 10^2$$

$$\left(\frac{D}{c}\right)^3 = (68)^3 = 314.4 \times 10^3$$

$$\frac{3}{4}\left(\frac{D}{c}\right)^3 = \frac{3}{4} \times 314.4 \times 10^3 = 235.8 \times 10^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{3}{2}\left(\frac{D}{c}\right)^2 = \frac{3}{2} \times 46.24 \times 10^2 = 69.36 \times 10^2$$

$$4\,\frac{D}{c} = 4 \times 68 = 272$$

$$\left[\frac{3}{4}\left(\frac{d}{C}\right)^3 + \frac{3}{2}\left(\frac{d}{C}\right)^2\right] = 10^3(235.8 + 6.936) = 242.736 \times 10^3$$

$$\left[\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{4D}{c}\right] = 10^3(6.936 + 0.272) = 7.208 \times 10^3$$

$$1 \text{ poise} = 0.1 \text{ N.s/m}^2$$

Thus by substitution we get

$$180 = \pi \times 0.1 \times V_p \times 0.08 \times 10^3(242.736 + 7.208)$$

$$\therefore \quad V_p = \frac{180}{\pi \times 0.1 \times 0.08 \times 10^3 \times 249.944} = 2.865 \times 10^{-2} \text{ m/s}$$

If shear force at the piston wall is neglected, then $\left[\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{D}{c}\right]$ is negligible.

Hence $$V_p = \frac{180}{\pi \times 0.1 \times 0.08 \times 10^3 \times 242.736} = 2.951 \times 10^{-2} \text{ m/s}$$

$$\therefore \text{ Percentage error} = \left(\frac{2.951 \times 10^{-2} - 2.865 \times 10^{-2}}{2.865 \times 10^{-2}}\right) \times 100 = 3\%$$

Example 13.14. *The thrust of a shaft is taken by a collar bearing filled with a forced lubrication system which maintains a film of oil of constant thickness 0.3 mm between the surface of the collar and the bearing. The outer and inner diameters of the collar are 156 mm and 120 mm respectively. The coefficient of viscosity of the oil is 1.2 poise. Calculate the power lost in friction of the bearing when the shaft rotates at 500 revolutions per minute.*

Solution

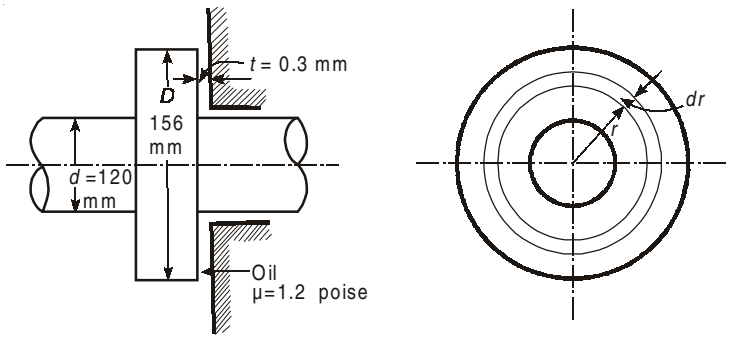


Figure Ex. 13.14

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The arrangement of the collar thrust bearing is shown in the Fig. Ex. 13.14. In this case the peripheral speed varies with the radius, hence considering an elemental ring of radius r and thickness dr, the shear stress at the ring.

$$\tau = \mu \frac{v}{t}$$

$$v = rw = \frac{2\pi Nr}{60} = \frac{2\pi \times 500r}{60} = \frac{50\pi r}{3} \text{ m/s}$$

$$\mu = 1.2 \text{ poise} = 0.12 \text{ N.s/m}^2$$

$$t = 0.3 \text{ mm} = 0.3 \times 10^{-3} \text{ m}$$

$$\therefore \quad \tau = 0.12 \times \frac{50\pi r}{3} \times \frac{1}{0.3 \times 10^{-3}} = 6667\, \pi\, r \text{ N/m}^2$$

Shear force on the ring $= \tau \times 2\pi r\, dr$

$= 13\,334\, \pi^2 r^2 dr$

Torque on the ring = Shear force × radius

$= 13\,334\, \pi^2 r^3 dr$

Total torque required to overcome the viscous resistance of oil

$$T = 13\,334\, \pi^2 \int_{0.06}^{0.078} r^3 dr = 0.791 \text{ Nm}$$

$$\therefore \quad \text{Power} = \frac{2\pi NT}{60} = \frac{2\pi \times 500 \times 0.791}{60} = 41.417 \text{ W}$$

Example 13.15. *A 0.15 m diameter journal runs in a bearing 0.3 m long. The lubricant used has a specific gravity of 0.855 and a kinematic viscosity 1.81 stokes. If the radial clearance may be assumed to be uniform and equal to 0.05 mm, determine the power required to overcome the viscous resistance of lubricant when the journal rotates at 300 r.p.m.*

Solution

Shear stress $\tau = \mu \frac{v}{C}$

where C is the clearance and v is peripheral velocity of the journal

$$C = 0.05 \times 10^{-3} \text{ m}; \; v = \frac{\pi DN}{60} = \frac{\pi \times 0.15 \times 300}{60} = 0.75\, \pi \text{ m/s}$$

$$\mu = \rho \upsilon = (0.855 \times 1000) \times 1.81 \times 10^{-4} = 0.155 \text{ N.s/m}^2$$

$$\therefore \quad \tau = 0.155 \times \frac{0.75\pi}{0.05 \times 10^{-3}} = 7304 \text{ N/m}^2$$

Shear force = Shear stress × wetted area

$= 7304 \times (\pi \times 0.15 \times 0.3)$

$= 1032.58$ N

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Torque to overcome viscous resistance of lubricant

$$T = \text{Shear force} \times \text{radius}$$

or $$T = 1032.58 \times \frac{0.15}{2} = 77.44 \text{ Nm}$$

$$\text{Power} = \frac{2\pi NT}{60}$$

$$= \frac{2\pi \times 300 \times 77.44}{60} = 2433 \text{ W} = 2.433 \text{ kW}$$

Example 13.16. *A flow of 420 litres/min of oil (specific gravity = 0.91, and viscosity = 1.24 poise) is pumped through a pipeline 75 mm diameter having a length of 62 m and whose outlet is 3 m higher than its inlet. Estimate the power required for the pump if its efficiency is 60%.*

Solution

Discharge $$Q = \frac{420 \times 10^{-3}}{60} = 7 \times 10^{-3} \text{ m}^3/\text{s}$$

Velocity $$V = \frac{7 \times 10^{-3}}{(\pi/4) \times (0.075)^2} = 1.584 \text{ m/s}$$

Reynolds number $$\text{Re} = \frac{\rho VD}{\mu}$$

$$\mu = 1.24 \text{ poise}$$

$$= (1.24 \times 0.1) \text{ N.s/m}^2$$

$$\therefore \quad \text{Re} = \frac{(0.91 \times 1000) \times 1.584 \times 0.075}{(1.24 \times 0.1)}$$

$$= 871.84$$

Since Re < 2000, the flow is laminar.

∴ Head loss due to friction

$$h_f = \frac{32\mu VL}{wD^2}$$

$$= \frac{32 \times (1.24 \times 0.1) \times 1.584 \times 62}{(9810 \times 0.91) \times (0.075)^2}$$

$$= 7.76 \text{ m}$$

Total head $$H = 3 + \frac{V^2}{2g} + h_f$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 3 + \frac{(1.584)^2}{2\times 9.81} + 7.76$$

$$= 10.888 \text{ m}$$

Power required for the pump

$$P = \frac{wQH}{\eta}$$

$$= \frac{(9810\times 0.91)\times 7\times 10^{-3}\times 10.888}{0.60}$$

$$= 1134 \text{ W} = 1.134 \text{ kW}$$

Example 13.17. *Derive an expression for the power lost in a footstep bearing shown in Fig. Ex. 13.17.*

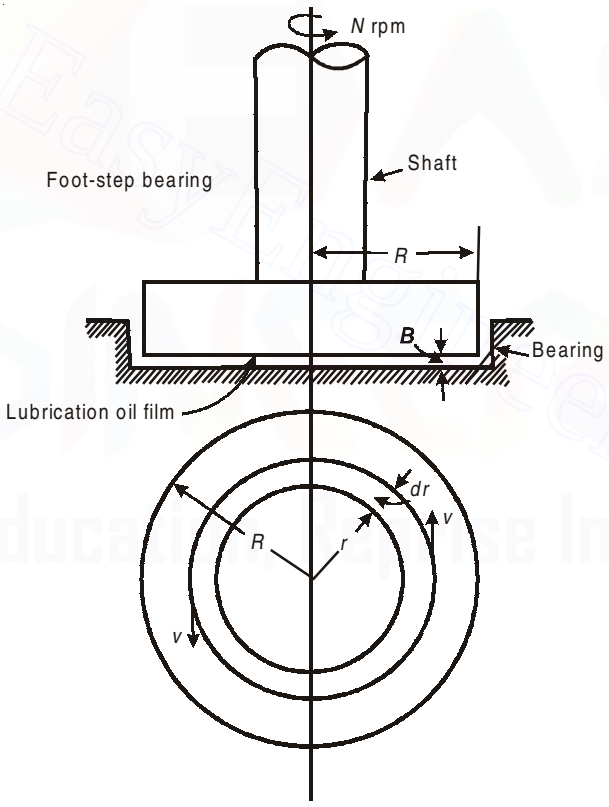


Figure Ex. 13.17

Solution

Since the lubrication film thickness B is generally very small, in this portion a linear velocity distribution may be assumed, or in other words plain (or simple) Couette flow may be considered for which from Eq. 13.45, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$v = V\frac{y}{B};\ \text{or}\ \frac{\partial v}{\partial y} = \frac{V}{B}$$

where V is the peripheral velocity of the shaft base at a radial distance r.

If the shaft is rotating at a speed of N r.p.m. then

$$V = \frac{2\pi Nr}{60}$$

Considering an elemental ring of radius r and thickness dr the shear stress at the ring.

$$\tau = \mu\frac{\partial v}{\partial y} = \mu\frac{V}{B} = \frac{2\pi Nr\mu}{60\times B}$$

Shear force on the ring $= \tau \times 2\pi r\, dr$

$$= \frac{4\pi^2 Nr^2\mu dr}{60\times B}$$

Torque on ring = Shear force × radius

$$= \frac{4\pi^2 Nr^3\mu dr}{60\times B}$$

Total torque required to overcome the viscous resistance of oil

$$T = \int_0^R \frac{4\pi^2 Nr^3\mu dr}{60\times B} = \frac{\pi^2 N\mu R^4}{60\times B}$$

$\therefore$ Power lost in the bearing $= \frac{2\pi NT}{60\times 1000}$ kW

$$= \frac{2\pi^3 N^2\mu R^4}{B\times 3600\times 1000}\ \text{kW}$$

SUMMARY OF MAIN POINTS

1. In laminar flow the fluid particles move along straight parallel paths in layers or laminae, such that the paths of the individual fluid particles do not cross those of the neighbouring particles. It occurs at low velocity so that forces due to viscosity predominate over the inertial forces and hence it is also known as *viscous flow*.
2. A flow in a pipe will be Laminar if the Reynolds number Re $\left(=\frac{\rho VD}{\mu}\right)$ is less than 2000.
3. For laminar flow of fluid through pipes shear stress distribution is given by

$$\tau = -\frac{\partial p}{\partial x}\frac{r}{2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where

τ = shear stress at a radial distance r from the pipe axis; and

$\frac{\partial p}{\partial x}$ = pressure gradient.

The shear stress τ varies linearly along the radius of the pipe. Thus at the pipe axis since $r = 0$, $\tau = 0$, and at the pipe surface since $r = R$, the shear stress τ_0 is given by

$$\tau_0 = -\frac{\partial p}{\partial x}\frac{R}{2}$$

4. For laminar flow of fluid through pipes the velocity distribution is given by

$$V = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)\left(R^2 - r^2\right)$$

The velocity distribution in laminar flow is parabolic with velocity $V = 0$ at the pipe surface since $r = R$, and at the axis of the pipe since $r = 0$, $V = V_{max}$ given by

$$V_{max} = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)R^2$$

The mean velocity of flow V, which is equal to (Q/A) or $(Q/\lambda R^2)$ is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^2$$

Thus

$$V = \frac{1}{2}v_{max}$$

The point where the local velocity is equal to the mean velocity is given by

$$r = 0.707R$$

where

R = radius of the pipe ; and

μ = dynamic viscosity of the folwing fluid.

5. For Laminar flow of fluid in a pipe the drop in pressure head h_f due to friction is given by Hagen–Poiseuille equation as

$$h_f = \frac{p_1 - p_2}{w} = \frac{32\mu VL}{wD^2}$$

or

$$h_f = \frac{64}{\text{Re}}\left(\frac{LV^2}{2gD}\right)$$

where

Re = Reynolds number = $\frac{\rho VD}{\mu}$; and

w = specific wieght of the flowing fluid.

The friction factor f for laminar flow in a pipe is given by

$$f = \frac{64}{\text{Re}}$$

The shear velocity $V_*\left(= \sqrt{\tau_0/\rho}\right)$

$$V_* = V\sqrt{\frac{f}{8}}$$

6. For laminar flow of fluid through inclined pipes the various expressions may be obtained by considering the piezometric head gradient $(\partial h/\partial x)$ in place of pressure gradient $(\partial p/\partial x)$.
7. Mean velocity of laminar flow through annulus of radius of the inner surface of the outer tube R_1 and radius of the outer surface of the inner tube R_2 is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)\left[R_1^2 + R_2^2 - \frac{\left(R_1^2 - R_2^2\right)}{\log_e\left(R_1/R_2\right)}\right]$$

8. For laminar flow of fluid between two parallel flat plates located at a distance B apart is given by

$$V = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)\left(By - y^2\right)$$

The maximum velocity v_{max} which occurs at the mid-point between the plates is obtained by substituting $y = \frac{B}{2}$, as

$$v_{max} = \frac{B^2}{8\mu}\left(-\frac{\partial p}{\partial x}\right)$$

The mean velocity of flow V is given by

$$V = \frac{B^2}{12\mu}\left(-\frac{\partial p}{\partial x}\right)$$

Thus

$$V = \frac{2}{3}v_{max.}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

9. For laminar flow of fluid between two parallel flat plates the drop in pressure head h_f due to friction is given by

$$h_f = \frac{p_1 - p_2}{w} = \frac{12\,\mu VL}{wB^2}$$

10. The distribution of shear stress at any section for laminar flow of fluid between two parallel flat plates is given by

$$\tau = \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right)$$

The shear stress varies linearly with the distance from the plate. It has the same maximum value at either plate i.e., at $y = 0$ or $y = B$ which is given by

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2}$$

The shear stress decreases linearly with the distance from the plate and it is equal to zero at the centre line between the plates.

11. The laminar flow of fluid between two parallel flat plates with one plate moving and the other being at rest is known as Couette flow for which the velocity distribution is given by

$$v = \frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left(By - y^2\right)$$

where

V = velocity of the moving plate.

When there is no pressure gradient in the direction of flow i.e., $\left(\frac{\partial p}{\partial x} = 0\right)$ it is known as *simple* (*or plain*) *Couette flow* for which the velocity distribution is linear given by

$$v = V\frac{y}{B}$$

12. For laminar flow of fluid in a very wide open channel with depth of flow D and bed slope $S_0(=-\partial z/\partial x)$ the velocity distribution is given by

$$v = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy - \frac{y^2}{2}\right)$$

The loss of head h_f between two sections of channel L distance apart is given by

$$h_f = \frac{3\mu VL}{wD^2}$$

13. For laminar flow of fluid through a porous media the velocity of flow of fluid V is given by Darcy's law as

$$V = Ki$$

where

i = hydraulic gradient ; and

K = coefficient of permeability

The loss of head for laminar flow of fluid through a porous media is given by

$$h_f = \frac{k\mu VL}{w\eta d_s^2}$$

where

η = effective ratio the area of pores A_p to the total area A i.e., $\eta = A_p/A$; and

d_s = grain diameter.

14. The resistance F_D experienced by a sphere of diameter D moving with a constant velocity V in a fluid of velocity μ is given by Stokes' law as

$$F_D = 3\pi\,\mu\,VD$$

15. For Journal Bearing velocity, shear force, torque and power absorbed are given as

$$\text{Velocity} \quad V = \frac{\pi DN}{60} = \frac{\partial v}{\partial y} = \frac{V}{t} = \frac{\pi DN}{60t}$$

$$\text{Shear stress} \quad \tau = \mu\frac{\partial v}{\partial y} = \frac{\mu\pi DN}{60t}$$

$$\text{Shear force} \quad F = \frac{\mu\pi^2 D^3 NL}{60t}$$

$$\text{Torque} \quad T = \frac{\mu\pi^2 D^3 NL}{2\times 60\,t}$$

$$\text{Power} \quad P = \frac{\mu\pi^3 D^3 NL}{60\times 60t}$$

where

L = Length of bearing;

D = diameter of shaft;

N = speed of shaft; and

t = clearance between shaft and bearing.

16. For Foot Step Bearing shear force, torque and power absorbed are given as

$$\text{Shear force } F = \frac{\mu}{15}\frac{\pi^2 N}{t}\frac{R^3}{3}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\text{Torque } T = \frac{\mu}{60t}\pi^2 NR^4$$

$$\text{Power } P = \frac{\mu\pi^3 N^2 R^4}{60 \times 30 \times t}$$

where
R = radius of the shaft; and
N = speed of the shaft.

17. For collar bearing torque and power absorbed are given as

$$\text{Torque } T = \frac{\mu}{60t}\pi^2 N\left(R_2^4 - R_1^4\right)$$

$$\text{Power } P = \frac{\mu\pi^3 N^2}{60 \times 30t}\left(R_2^4 - R_1^4\right)$$

where
R_1 = internal radius of the collar;
R_2 = external radius of the collar;
t = thickness of oil film; and
P = power in watts.

18. A dash-pot is a device used for damping vibrations of machines, which may be achieved by making use of a fluid of fairly high viscosity.

19. The coefficient of viscosity of a liquid may be determined experimentally by the following methods:

(i) Coaxial cylinder viscometer

$$\mu = \frac{60(R_2 - R_1)\alpha T}{\pi^2 R_1^2 N\left[4\alpha h R_2 + R_1^2(R_2 - R_1)\right]}$$

where
R_1 = radius of inner stationary cylinder;
R_2 = radius of outer rotating cylinder;
h = depth of liquid in the cylinder;
α = clearance between the two cylinders at the bottom;
N = speed of rotation; and
T = torque

(ii) Capillary tube viscometer

$$\mu = \frac{\pi w H D^4}{128 Q L}$$

where
w = specific weight of liquid;
H = head over capillary tube;
D = diameter of capillary tube;
L = length of capillary tube; and
Q = discharge through capillary tube.

(iii) Falling sphere viscometer

$$\mu = \frac{D^2}{18V}(w_s - w)$$

where
D = diameter of the spherical ball;
V = velocity of falling ball;
w_s = specific weight of ball; and
w = specific weight of the liquid.

PROBLEMS

13.1 Derive an expression for mean velocity for laminar flow (i) through a pipe; (ii) between parallel plates.

13.2 A horizontal pipe 50 mm diameter carrying glycerine has shear stress at the pipe boundary as 196.2 N/m² [20 kg (f)/m²]. Determine the pressure gradient, mean velocity and Reynolds number. Take for glycerine r = 1275.3 kg/m³ (130 msl/m³) and $\mu = 80.442 \times 10^{-2}$ N.s/m³ [8.20 $\times 10^{-2}$ kg(f)–s/m²].

[*Ans.* 15.696 kN/m² per m {1600 kg(f)/m² per m}; 1.52 m/s; 120.5]

13.3 A steady flow of oil (sp. gr. 0.85) occurs in a 0.1 m diameter pipe which has a downward slope of 1 in 100. If the discharge measured is 2 litres per second, determine the pressure drop between two points on the pipe 200 m apart. Take μ for oil as 1.2 poise.

[*Ans.* 2877 N/m³ {293.25 kg(f) /m²}]

13.4 Using Hagen-Poiseuille equation derive an expression for the head loss in a pipe of diameter D and length L in terms of Reynolds number and velocity head.

13.5 An oil having viscosity of 1.43 poise and specific gravity 0.9 flows through a pipe 25 mm diameter and 300 m long at 1/10 of the critical velocity for which Reynolds number is 2500. Find (a) the velocity of flow through the pipe; (b) the head in metres of oil across the pipe length required to maintain the flow; and (c) the power of the flow.

[*Ans.* (a) 1.59 m/s; (b) 396 m of oil; (c) 2.729 kW (3.71 h.p.)]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

13.6 Two parallel plates kept 75 mm apart have laminar flow of glycerine between them with a maximum velocity of 1 m/s. Calculate the discharge per metre width, the shear stress at the plates, the difference in pressure in pascals (or N/m^2) between two points 25 m apart, the velocity gradients at the plates and velocity at 15 mm from the plate. Take viscosity of glycerine as 8.35 poise.

[*Ans.* 0.05 m^3/s per m; 44.53 N/m^2 {4.54 kg(f)/m^2};29.69 kN/m^2 {0.303 kg(f)/cm^2}; 53.3 s^{-1} ; 0.64 m/s]

13.7 Two parallel plates are placed horizontally 10 mm apart. The bottom plate is fixed and the top plate is moved at a uniform speed of 0.25 m/s. The fluid between them has a dynamic viscosity μ equal to 1.472 N.s/m^2 [0.15 kg(f)-s/m^2]. Determine the pressure gradient which corresponds to the condition of zero discharge between the plates and the shearing stress at each plate. [**Hint.** For zero discharge

$$q = \int_0^B \left[\frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)\right]dy = 0$$

[*Ans.* 22.08 kN/m^2 per m {2.25 × 10^3 kg(f)/m^2 per m};– 73.6 N/m^2 {–7.5 kg(f)/m^2}; 147.2 N/m^2 {15 kg(f)/m^2}]

13.8 Lubricating oil of specific gravity 0.90 and viscosity 6.0 poise is pumped at the rate of 0.02 m^3/s through a 0.2 m diameter of the pipe 400 m long. Determine the slope of the pipe and the direction of flow so that the pressure gradient along the pipe is zero.

[*Ans.* 1 in 28.89; downward direction]

13.9 A horizontal column of sand 1 m long and 0.1 m in diameter has water flowing through it. If the discharge passing through it is 5 litres/min and the transmission constant of the material K is 4.5 mm/s, determine the pressure drop across the column.

[*Ans.* 23.131 kN/m^2 {2358 kg(f)/m^2}]

13.10 In a rotary viscometer the radii of the cylinders are respectively 50 mm and 50.5 mm, and the outer cylinder is rotated steadily at 30 rad/s. For a certain liquid the torque is 0.4503 N.m [0.0459 kg(f)-m] when the depth of the liquid is 0.05 m and the torque is 0.8103 N.m [0.0826 kg(f)-m] when the depth is 0.1 m. Estimate the viscosity of the liquid.

[*Ans.* 0.1513 N.s/m^2 {0.0154 kg(f)-s/m^2}]

13.11 A small lead sphere of specific gravity 11.5 has a terminal fall velocity of 7 mm/s in a fluid of specific gravity 1.25 and dynamic viscosity 7.848 poise. Determine the diameter of the sphere.

[*Ans.* 1 mm]

13.12 Show that when laminar flow occurs with mean velocity V between extensive stationary flat plates, the mean kinetic energy per unit weight of the fluid is (1.543 V^2/2g).

13.13 In a Saybolt universal viscometer, the time for the flow of 60 cm^3 of a liquid of specific weight 8.633 kN/m^3 [880 kg(f)/m^3] is 170 seconds. Calculate the absolute viscosity of the liquid.

[*Ans.* 3.198 × 10^{-2} N.s/m^2 {3.26 × 10^{-3} kg(f) – s/m^2}]

13.14 A force of 2.453 N [0.25 kg(f)] is required to drag a thin plate at a constant velocity of 0.3 m/s between two fixed plates when the space between these two fixed plates is filled with a liquid. If the area of the plate is 0.1 m^2, it travels equidistant from and parallel to the fixed plates, and the distance from the moving plate to the fixed plates is 3 mm, calculate the viscosity of the liquid.

[*Ans.* 1.227 × 10^{-2} N.s/m^2 {1.25 × 10^{-3} kg(f) – s/m^2}]

13.15 A uniform circular tube of bore radius a has a fixed coaxial cylindrical core of radius b. An incompressible viscous fluid flows through the annular passage under a pressure gradient $(-\partial p/\partial x)$. Determine the radius at which the shear stress in the stream is zero, given that the flow is laminar under steady state condition.

$$\left[\text{Ans. } \left\{\frac{a^2 - b^2}{2\log_e(a/b)}\right\}^{1/2}\right]$$

13.16 A horizontal circular tube of radius a has a fixed coaxial cylindrical rigid core of radius b. τ_a and τ_b are the shear stresses along the tube and core surfaces when a viscous liquid is flowing through the annulus. The flow is laminar and the rate of variation of pressure along the length of the passage is $(-\partial p/\partial x)$. Show that $a\tau_a - b\tau_b = \frac{1}{2}(a^2 - b^2)\left(\frac{\partial p}{\partial x}\right)$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Turbulent Flow in Pipes

Chapter 14

14.1 INTRODUCTION

Mostly the flow in pipes is turbulent. As stated earlier if Reynolds number is greater than 4000 the flow is turbulent. The velocity distribution in turbulent flow is relatively uniform and the velocity profile of turbulent flow is much flatter than the corresponding laminar flow parabola for the same mean velocity. It becomes even flatter with increasing Reynolds number as shown in Fig. 14.1 in which a comparison between velocity profiles for three different values of Reynolds number Re = 10^3, 10^4 and 10^6 is shown. In turbulent flow the fluid particles are in an extreme state of disorder, their movement is haphazard and large scale eddies are developed, which results in a complete mixing of the fluid. Such irregular motion of fluid particles in turbulent flow is on account of the fact that at any fixed point in space, the velocity and pressure do not remain constant with time, but fluctuate in irregular manner. In other words, in turbulent flow there are irregular velocity and pressure fluctuations of high frequency superimposed on the main flow. On account of these fluctuations the analytical treatment of turbulent flow is rendered extremely complicated. No complete theory has yet been developed for the analysis of turbulent flow and one has to depend at some point on experimentally derived information. However, in turbulent flow also the mean components of velocity and pressure may be considered, which may be assumed to have a constant value at any section if the average are taken over a sufficiently long interval of time.

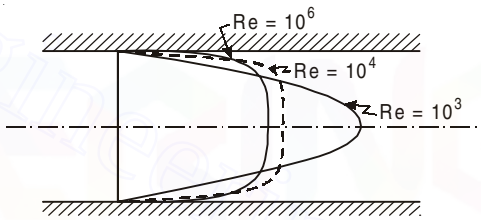


Figure 14.1 *Velocity distribution in laminar and turbulent flow*

In the case of turbulent flow the velocity fluctuations influence the mean motion in such a way that an additional shear (or frictional) resistance to flow is caused. This shear stress produced in turbulent flow is in addition to the viscous shear stress and it is termed as turbulent shear stress which may be evaluated as explained in the next section.

14.2 SHEAR STRESSES IN TURBULENT FLOW

As stated earlier, in turbulent flow velocity fluctuations cause a continuous interchange of fluid masses between the neighbouring layers, which is accompanied by a transfer of momentum. A large

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

number of lumps of fluid particles move from one fluid layer into the adjacent layers above and below and get merged in them. Since each fluid layer possesses a different velocity, such movement of fluid particles brings about a transfer of momentum from one layer to another. But the change of momentum is equivalent to the force in a particular direction. Hence such momentum transport due to fluctuations results in developing additional shear stresses of high magnitude between adjacent layers. In order to determine the magnitude of the turbulent shear stress a number of semiempirical theories have been developed, some of which are discussed below.

In analogy with the expression for the viscous shear. J. Boussinesq, a French mathematician in 1877 developed an expression for the turbulent shear stress which may be expressed as

$$\tau = \eta \frac{d\bar{v}}{dy} = \rho\varepsilon \frac{d\bar{v}}{dy} \qquad ...(14.1)$$

in which $\bar{v}$ denotes the average velocity at a distance y from boundary. The turbulent mixing coefficient η (Greek 'eta') introduced by Boussinesq which is analogous to the absolute viscosity μ, is often called 'apparent' or 'virtual' or 'eddy' viscosity. Similar to kinematic viscosity $\upsilon = \left(\frac{\mu}{\rho}\right)$, the 'apparent' or 'virtual' or 'eddy' kinematic viscosity ε (Greek 'epsilon') is also obtained by dividing eddy viscosity η, by the mass density of the fluid ρ, thus

$$\varepsilon = \frac{\eta}{\rho} \qquad ... (14.2)$$

When viscous action is also included the total shear stress may be expressed as

$$\tau = \mu \frac{d\bar{v}}{dy} + \eta \frac{d\bar{v}}{dy} = (\mu + \eta) \frac{d\bar{v}}{dy} \qquad ... (14.3)$$

The magnitude of η may vary from zero (if the flow is laminar) to several thousand times of μ . The value of η depends on the momentum carried by the migrating particles and thus on the density of the flowing fluid and the characteristics of flow. Further the kinematic eddy viscosity $\varepsilon = \left(\frac{\eta}{\rho}\right)$ may be considered to be independent of the properties of the fluid but depends on the characteristics of flow. Thus whereas μ is a fluid property and is a function of temperature of fluid alone, η and ε are mainly the functions of the characteristics of flow, and may be expected to vary from point to point in the flow. Since the values of η and ε cannot be predicted, the Boussinesq's hypothesis is, however, of limited use.

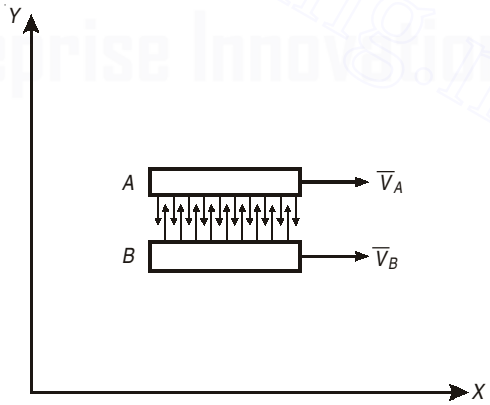


Figure 14.2 *Transfer of momentum in turbulent flow*

In 1886 Reynolds developed an expression for the turbulent shear stress which may be derived as follows. Consider two fluid layers A and B at a small distance apart as shown in Fig. 14.2, having the average velocities $\bar{V}_A$ and $\bar{V}_B$ respectively. It is assumed

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

that $\bar{V}_A > \bar{V}_B$ and the relative velocity of layer A with respect of layer B is $(\bar{V}_A - \bar{V}_B) = v_x$; where v_x is the fluctuating component of velocity in the x-direction due to turbulence. It then means that if layer B is assumed to be stationary then layer A may be assumed to be moving at a uniform velocity v_x. Further in the y-direction let the fluctuating component be v_y. Now if over a surface of area A perpendicular to the y-direction and separating two adjacent fluid layers the component v_y are uniformly distributed, then the mass of fluid transferred across that surface from one layer to the other per second is $(\rho A v_y)$. Since this mass of fluid has started moving with a relative velocity v_x, a transfer of momentum equal to $(\rho A v_y v_x)$ takes place, resulting in developing tangential forces on each of the layers. The corresponding turbulent shear stress exerted on the fluid layers is equal to

$$\tau = \frac{\text{Force}}{\text{Area}} = \frac{\rho A v_y v_x}{A} = (\rho v_y v_x) \qquad ...(14.4)$$

Since v_x and v_y are varying, the magnitude of τ will also vary. Hence usually the time average value of shear stress is considered. Thus by taking the time average on both sides of Eq.14.4 it becomes

$$\bar{\tau} = \overline{\rho v_y v_x} \qquad ...(14.5)$$

The stress represented by Eq. 14.5 is usually termed as Reynolds stress.

In 1925 L. Prandtl, a German Engineer made an important advance in this direction by presenting mixing length hypothesis, by means of which the turbulent shear stress can be expressed in terms of measurable quantities related to the average flow characteristics.

According to Prandtl, mixing length is that distance in the transverse direction which must be covered by a lump of fluid particles travelling with its original mean velocity in order to make the difference between its velocity and the velocity of the new layer equal to the mean transverse fluctuation in turbulent flow. Prandtl has further indicated that the velocity fluctuation in the x-direction, v_x may be related to the mixing length l by the following expression

$$v_x = l \frac{d\bar{v}}{dy} \qquad ...(14.6)$$

Further the fluctuation component v_y is of the same order of magnitude as v_x and can be written as

$$v_y = l \frac{d\bar{v}}{dy} \qquad ...(14.7)$$

and hence $\overline{v_x v_y}$ can be written as

$$\overline{v_x v_y} = l^2 \left(\frac{d\bar{v}}{dy}\right)^2 \qquad ...(14.8)$$

From Eqs. 14.5 and 14.8 the expression for turbulent shear stress becomes

$$\bar{\tau} = \rho l^2 \left(\frac{d\bar{v}}{dy}\right)^2 \qquad ...(14.9)$$

Again the total shear stress at any point is the sum of the viscous shear stress and turbulent shear stress and it may be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\bar{\tau} = \mu \frac{d\bar{v}}{dy} + \rho l^2 \left(\frac{d\bar{v}}{dy} \right)^2 \qquad \ldots(14.10)$$

However, the viscous shear stress is negligible as compared with the turbulent shear stress.

The superiority of Eq. 14.9 given by Prandtl lies in the fact that it is possible to make suitable assumptions regarding the variation of the mixing length. In several problems of fluid flow it is possible to establish a simple relation between l and a characteristic length of the flow and therefore Prandtl's Eq. 14.9 has been successfully applied to the various turbulent flow problems.

14.3 FORMATION OF BOUNDARY LAYER IN PIPES—ESTABLISHMENT OF FLOW IN PIPES

The concept of boundary layer formation along a flat plate, indicated in Chapter 12, may also be used in the case of flow of a fluid in a pipe. When a fluid flows through a pipe, close to the pipe wall the fluid is retarded, thus resulting in the formation of the boundary layer in the same manner as in the case of a plane boundary. However, the formation of boundary layer in a pipe is somewhat different from that on a flat plate. Whereas the boundary layer over a flat plate may expand indefinitely (provided the depth of flow and the length of the plate are infinite), the boundary layer in a pipe may attain a maximum thickness equal to the pipe radius. This is so because from the entrance section of the pipe the boundary layer gradually increases and at a certain section in the downstream when it attains a thickness equal to the radius of the pipe it cannot expand any more. Thus from this section onwards in the downstream direction the thickness of the boundary layer remains unchanged. With the increase in the thickness of the boundary layer from the entrance section, more and more fluid will be retarded in the downstream direction, and hence in order to maintain a constant discharge, the velocity in the central core will increase. As such, as shown in Fig. 14.3, in between the entrance section and the section AA, where the boundary layer attains thickness equal to the radius of the pipe, the velocity

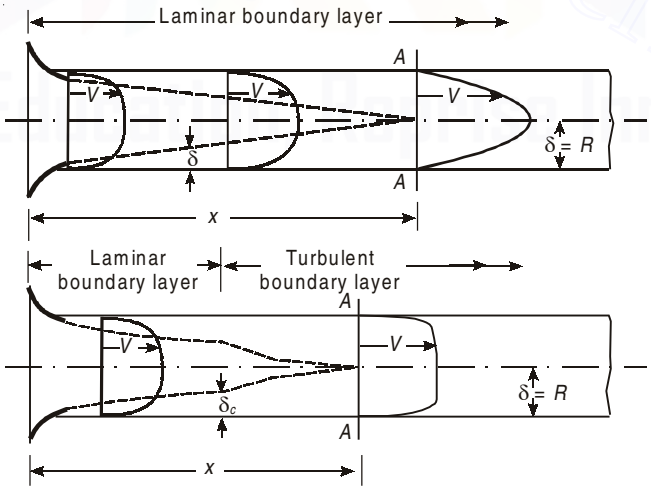


Figure 14.3 *Establishment of flow in a pipe*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

distributions will vary from section to section due to the variation in the thickness of the boundary layer, but the velocity distributions at the section *AA* and at any other section on the downstream will be exactly the same. Accordingly it is only beyond section *AA*, the flow in the pipe will be truly uniform and the flow is then said to be established. The length of the pipe from the entrance section upto the section *AA* is termed as the length required for the establishment of flow in the pipe. As described below different lengths are required for the establishment of laminar and turbulent flows in pipes.

For laminar flow in a pipe, laminar boundary layer will be developed for the entire length of the pipe and as shown in Fig. 14.3(a), at section *AA* thickness of the boundary layer will become equal to the radius of pipe. Since laminar flow has parabolic velocity distribution, at section *AA* and downstream of it the velocity distributions will be parabolic as shown in Fig. 14.3(a). The length of pipe *x*, from the entrance of the pipe upto section *AA*, is therefore the length required for the establishment of fully developed laminar flow in the pipe. Experiments have shown that for laminar flow (x/D) is a function of Reynolds number $\text{Re}\left(=\frac{\rho VD}{\mu}\right)$ where D is diameter of the pipe, ρ and μ are mass density and viscosity of the flowing fluid and V is the mean velocity of flow in the pipe. Rouse has given the following relationship between (x/D) and Re for the establishment of laminar flow in pipes.

$$\frac{x}{D} = 0.07\ \text{Re} \qquad \text{... (14.11)}$$

If the flow in a pipe is turbulent, even then as shown in Fig. 14.3 (b), for a small distance from the entrance section, laminar boundary layer will be formed, which will change to turbulent boundary layer before the thickness of the boundary layer becomes equal to the radius of the pipe. However, in some cases, if the pipe is rough and the intensity of turbulence of the incoming flow is high, from the entrance section itself turbulent boundary layer may be formed. The thickness of the boundary layer will increase in the downstream direction upto a certain section *AA* [see Fig. 14.3 (b)] where it will become equal to the radius of the pipe. As such from section *AA* onwards there will exist a fully developed turbulent flow in the pipe. Since a turbulent flow has logarithmic velocity distribution, it is much more uniform and hence the length of pipe *x*, required for the establishment of fully developed turbulent flow in a pipe is relatively less. Rouse has indicated that the distance required for the establishment of a fully developed turbulent flow in a pipe is given by the following expression

$$\frac{x}{D} = 50 \qquad \text{... (14.12)}$$

It may be noted that the various expressions derived for laminar flow in pipes in Chapter 13 and those for turbulent flow in pipes in this chapter are applicable only for the established flows.

14.4 HYDRODYNAMICALLY SMOOTH AND ROUGH BOUNDARIES

In general a boundary with irregularities of large average height k, on its surface is considered to be a rough boundary and the one with smaller k values is considered as a smooth boundary. However for a proper classification of smooth and rough boundaries, the flow and fluid characteristics are required to be considered in addition to the boundary characteristics.

Consider a boundary such that the average height k, of the irregularities projecting from its surface is much less than the thickness of the laminar sublayer δ', Fig. 14.4. As the flow outside the laminar

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

sublayer is turbulent, eddies of various sizes are present which try to penetrate through the laminar sublayer. But due to greater thickness of the laminar sublayer the eddies cannot reach the surface irregularities, and thus the boundary acts as a smooth boundary. Such a boundary is known as *hydrodynamically smooth boundary.*

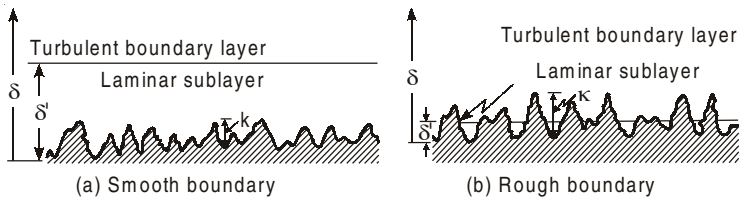


Figure 14.4 *Definition of smooth and rough boundaries*

With the increase in Reynolds number the thickness of the laminar sublayer decreases, and it can even become much smaller than the average height k, of surface irregularities. The irregularities will then project through the laminar sublayer and the laminar sublayer is completely destroyed. The eddies will thus come in contact with the surface irregularities and large amount of energy loss will take place such a boundary is known as *hydrodynamically rough boundary.*

As indicated later it has been found by Nikuradse's experiment that the boundary behaves as hydrodynamically smooth boundary when $(k//\delta')$ is less than 0.25 and as hydrodynamically rough when (k/δ') is greater than 6.0. The boundary is classified as boundary in transition for $0.25 < (k//\delta') < 6.0$.

14.5 VELOCITY DISTRIBUTION FOR TURBULENT FLOW IN PIPES

For turbulent flow in circular pipes Prandtl assumed the mixing length l to be linear function of the distance y from the pipe wall i.e.,

$$l \sim y$$

or
$$l = \kappa y \qquad ...(14.13)$$

in which κ (Greek 'kappa') is the constant of proportionality and is the so called 'Karman universal constant'. The Prandtl's hypothesis $l = \kappa y$, has been confirmed for the regions very close to the pipe boundary, by the experimental studies of Prandtl's student. J. Nikuradse, who used smooth pipes as well as pipes artificially roughened by cemented coatings of sand grains of uniform size. The analysis of Nikuradse's experimental results has shown that in the immediate neighbourhood of the pipe boundary the expression for the mixing length is approximately

$$l = 0.4y - 0.44\left(\frac{y^2}{R}\right) \qquad ...(14.14)$$

For small values of y, close to the pipe boundary, Eq. 14.14 reduces to

$$l = 0.4y \qquad ...(14.15)$$

By comparing Eqs 14.13 and 14.15 the value of Karman Universal constant κ is found to be 0.4.

Now by substituting for the mixing length from Eq. 14.13 in Eq. 14.9 the turbulent shear stress may be expressed as

$$\tau = \rho\kappa^2 y^2\left(\frac{dv}{dy}\right)^2 \qquad ...(14.16)$$

For the sake of convenience, the bars denoting the time average of the various quantities are omitted.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since turbulent shear stress also varies linearly with radius (Chapter 13) it can be expressed as

$$\tau = \tau_0\left(\frac{r}{R}\right) = \tau_0\left(\frac{R-y}{R}\right) = \tau_0\left(1-\frac{y}{R}\right) \quad ...(14.17)$$

where τ_0 represents the turbulent shear stress at the pipe boundary, i.e., at $y = 0$ or $r = R$.

However, for small values of y that is close to the pipe boundary the shear stress τ may be assumed to be constant, being approximately equal to τ_0 from Eq. 14.17. Thus Eq. 14.16 becomes

$$\tau_0 = \rho\kappa^2 y^2\left(\frac{dv}{dy}\right)^2 \quad ...(14.18)$$

Equation 14.18 can be rearranged to solve for (dv/dy) as

$$\left(\frac{dv}{dy}\right) = \frac{1}{\kappa y}\left(\frac{\tau_0}{\rho}\right)^{1/2} = \frac{V_*}{\kappa y} \quad ...(14.19)$$

since $(\tau_0/\rho)^{1/2} = V_*$, shear velocity or friction velocity (as explained in Chapter 13). For a given case of turbulent flow, V_* is constant and hence Eq. 14.19 may be integrated to yield the following relationship

$$v = \frac{V_*}{\kappa}\log_e y + C \quad ...(14.20)$$

Equation 14.20 indicates that in turbulent flow the velocity varies directly with the logarithm of the distance from the boundary or the velocity distribution in turbulent flow is logarithmic in nature. The constant of integration C may be determined from the boundary condition at the centre of the pipe:

$$v = v_{max} \text{ (maximum velocity) at } y = R$$

$$C = v_{max} - \frac{V_*}{\kappa}\log_e R$$

The use of boundary condition at the centre of the pipe is, however, not strictly agreeable to Prandtl's original assumption that Eq. 14.19 is applicable only to the flow close to the pipe boundary. But Nikuradse's experimental studies have indicated that Eq. 14.19 holds good for the entire cross-section of the pipe. Thus by substituting the expression for C in Eq. 14.20 and using the value of $\kappa = 0.4$, the expression for velocity distribution becomes

$$v = v_{max} + 2.5V_*\log_e\left(\frac{y}{R}\right) \quad ...(14.21)$$

Equation 14.21 is known as 'Prandtl's universal velocity distribution equation, for turbulent flow in pipes. In obtaining this equation no assumption has been made as to whether the pipe boundary is hydrodynamically smooth or rough. Hence Eq. 14.21 is applicable for both smooth and rough pipe boundaries. By dividing both sides of Eq. 14.21 by V_* and rearranging the terms, the equation may be expressed in a non-dimensional form:

$$\frac{v_{max} - v}{V_*} = 2.5\log_e\left(\frac{R}{y}\right) \quad ...(14.22)$$

Changing from natural (to the base e) to common (to the base ten) logarithms Eq. 14.22 may be written as

$$\frac{v_{max} - v}{V_*} = 5.75\log_{10}\left(\frac{R}{y}\right) \quad ...(14.23)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The difference between the maximum velocity v_{max} and the local velocity at any point v, that is $(v_{max} - v)$ is commonly known as 'velocity defect'.

It is obvious from Eq. 14.22 that $\left[\frac{(v_{max} - v)}{V_*}\right]$ is a function of (y/R) alone. Hence the velocity distribution curve given by Eq. 14.22 appears to be independent of the nature of the boundary. However, the experiments conducted by Fritsch have indicated that the velocity distribution curves for smooth and rough pipes are identical over a substantial portion of the pipe except for a narrow region in the vicinity of the boundary, where the nature of the pipe boundary affects the velocity distribution curve. It may therefore be stated that Prandtl's equation [Eq. 14.22] applies only to the turbulent flow in the central region of the pipe. Moreover different velocity distribution equations will have to be derived for the hydrodynamically smooth and rough boundaries as discribed below.

14.6 VELOCITY DISTRIBUTION FOR TURBULENT FLOW IN HYDRODYNAMICALLY SMOOTH AND ROUGH PIPES—KARMAN PRANDTL VELOCITY DISTRIBUTION EQUATION

As derived in the previous section the velocity distribution for turbulent flow in pipes is given by Eq. 14.20 as

$$v = \frac{V_*}{\kappa} \log_e y + C \qquad ...(14.20)$$

This velocity distribution for turbulent flow has a peculiarity that at the boundary, that is for $y = 0$, it gives a velocity v equal to $-\infty$ (minus infinity). Hence it is only at a certain finite distance above the boundary, say, $y = y'$, that the velocity will be equal to zero as shown in Fig. 14.5. Thus with this condition that at $y = y'$ the velocity $v = 0$, the constant C can be obtained as

$$C = \frac{V_*}{\kappa} \log_e y'$$

Introducing this value of the constant C in Eq. 14.20, it may be expressed as

$$v = \frac{V_*}{\kappa} \log_e \left(\frac{y}{y'}\right) \qquad ...(14.24)$$

The tendency of the logarithmic velocity distribution curve to indicate a zero velocity at a certain distance above boundary is, however, in disagreement with the physical fact that the velocity at the boundary is zero. As such Eq. 14.20 for logarithmic velocity distribution applies only to turbulent flow in the central region of the pipe, and it cannot be applied to the regions close to the pipe boundary.

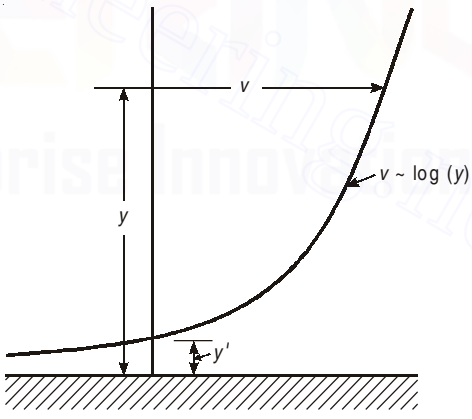


Figure 14.5. *Logarithmic velocity distribution curve for turbulent flow*

(a) Velocity Distribution in Smooth Pipes. In the vicinity of a smooth boundary there exists a laminar sublayer. The flow in the laminar sublayer being laminar has a parabolic velocity distribution.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As such the velocity which is zero at the pipe boundary increases parabolically in the zone of laminar motion, which extends upto certain distance from the boundary. Above the zone of laminar motion there exists a transition zone where the flow changes from laminar to turbulent. Beyond the transition zone the flow is turbulent having logarithmic velocity distribution. Since the change from parabolic to logarithmic distribution is gradual, the zone of laminar motion will extend well beyond the distance y', as shown in Fig. 14.6. However, in the absence of any specific line of demarcation between the different zones of flow near the boundary, the intersection of the parabolic and the logarithmic velocity distribution curves, as shown in Fig. 14.6 is arbitrarily chosen as nominal border line between the two types of flow. Moreover the same point of intersection between the two curves is arbitrarily taken as the limit of laminar sublayer. Since the thickness of the laminar sublayer δ' is generally very small, the parabolic velocity distribution in this region may be approximated by a straight line without appreciable error, as shown in Fig. 14.6.

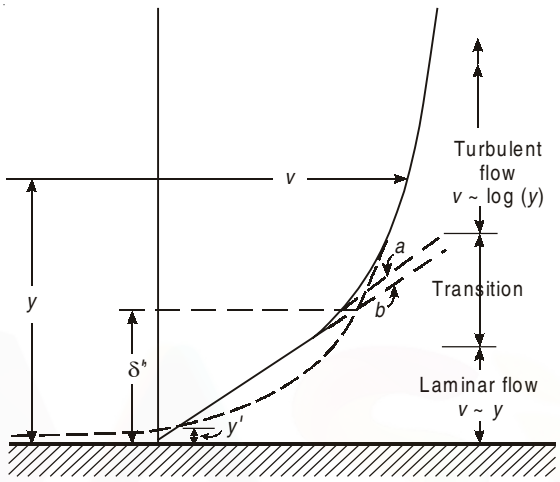


Figure 14.6 *Velocity distribution for turbulent flow near a smooth boundary*

In the zone of laminar sublayer, since the flow is laminar, the viscous stress predominate and the turbulent stresses tend to become zero. Therefore, in the laminar sublayer the shear stress $\tau = \mu(dv/dy)$.

For linear velocity distribution within the laminar sublayer, the velocity gradient (dv/dy) becomes (v/y). Furthermore, if it is assumed that in the laminar sublayer, that is upto $y = \delta'$, the shear stress τ, remains constant equal to τ_0, the shear stress at the pipe boundary, then

$$\tau = \tau_0 = \mu \frac{v}{y} \qquad ...(14.25)$$

By dividing both sides of Eq. 14.25 by mass density ρ and replacing the ratio (τ_0/ρ) by V_*^2 Eq. 14.25 may be rearranged in the following non-dimensional form:

$$\frac{v}{V_*} = \frac{\rho V_* y}{\mu}$$

or

$$\frac{v}{V_*} = \frac{V_* y}{\upsilon} \quad [\text{since } (\mu/\rho) = \upsilon] \qquad ...(14.26)$$

Equation 14.26 thus provides the velocity distribution within laminar sublayer, that is from $y = 0$ to $y = \delta'$. The term $(V_* y/v)$ on the right hand side of Eq. 14.26 is dimensionally equivalent to a form of Reynolds number.

Nikuradse's experimental studies of turbulent flow in smooth pipes have shown that for turbulent flow in smooth pipes of any size the value of the parameter $(V_* y/\upsilon)$ for $y = \delta'$ is approximately 11.6 and for $y = y'$ it is approximately 0.108. That means

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{V_* \delta'}{\upsilon} = 11.6$$

or $$\delta' = \frac{11.6\upsilon}{V_*} \quad \ldots (14.27)$$

and $$\frac{V_* y'}{\upsilon} = 0.108$$

or $$y' = \frac{0.108\upsilon}{V_*} = \frac{\delta'}{107} \quad \ldots (14.28)$$

Now by substituting the value of y' given by Eq. 14.28 in Eq. 14.24 the following expression is obtained

$$v = \frac{V_*}{\kappa} \log_e \left(\frac{V_* y}{0.108\upsilon} \right) \quad \ldots (14.29)$$

Changing from natural (to the base e) to common (to the base ten) logarithms and using the value of $\kappa = 0.4$, Eq. 14.29 may be written as

$$v = 5.75 V_* \log_{10} \left(\frac{V_* y}{0.108\upsilon} \right) \quad \ldots (14.30)$$

By dividing both sides of Eq. 14.30 by V_* it may be rearranged in the following form

$$\frac{v}{V_*} = 5.75\, V_* \log_{10} \left(\frac{V_* y}{\upsilon} \right) + 5.5 \quad \ldots (14.31)$$

Equation 14.31 represents the velocity distribution in the region of turbulent flow near hydrodynamically smooth boundaries and it is applicable for regions outside the laminar sublayer that is for y equal to or greater than δ'. It is known as Karman-Prandtl equation for the velocity distribution near hydrodynamically smooth boundaries. Though Eq. 14.31 has been derived to represent the velocity distribution near smooth boundaries only, but it has been found that the experimental data of Nikuradse on velocity measurements in hydrodynamically smooth pipes follow closely the trend of Eq. 14.31, even for the distances farther away from the pipe boundary. As such Eq. 14.31 is used to represent the velocity distribution in the entire region of turbulent flow in the pipe *viz.*, $y = \delta'$ to $y = R$. Therefore Eq. 14.31 may be considered as Karman–Prandtl equation for velocity distribution for turbulent flow in hydrodynamically smooth pipes.

The velocity distribution for turbulent flow in smooth pipe may also be represented by an exponential equation instead of the logarithmic equation. It has been found empirically that for $(V_* y/v)$ between 70 and 700. Equation 14.31 for turbulent flow in smooth pipes may be approximated by

$$\frac{v}{V_*} = 8.74 \left(\frac{V_* y}{\upsilon} \right)^{1/7} \quad \ldots (14.32)$$

Further at the centreline of the pipe i.e., for $y = R$ the velocity v will be equal to v_{max}. Then from Eq. 14.32

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{v_{max}}{V_*} = 8.74 \left(\frac{V_* R}{\upsilon}\right)^{1/7} \quad ...(14.33)$$

By dividing Eq. 14.32 by Eq. 14.33 the so called 'Blasius one-seventh power velocity distribution equation' is obtained as

$$\frac{v}{v_{max}} = \left(\frac{y}{R}\right)^{1/7} = \left(1 - \frac{r}{R}\right)^{1/7} \quad ...(14.34)$$

since $y = (R - r)$

However, in a general form the exponential equation for velocity distribution for turbulent flow in smooth pipes may be expressed as

$$\frac{v}{v_{max}} = \left(\frac{y}{R}\right)^{1/n} = \left(1 - \frac{r}{R}\right)^{1/n} \quad ...(14.35)$$

where exponent $(1/n)$ decreases with increasing Reynolds number, but tends to approach a constant numerical value asymptotically for very large Reynolds numbers.

(b) Velocity Distribution in Rough Pipes. The flow conditions near hydrodynamically rough boundaries are different from those near hydrodynamically smooth boundaries. This is so because in the case of rough boundaries the surface irregularities protrude well beyond the laminar sublayer, which is therefore completely destroyed, as shown in Fig. 14.7. As such velocity distribution in turbulent flow near hydrodynamically rough boundaries is considerably affected by the surface protrusions, Equation 14.24 applies equally to the turbulent flow in hydrodynamically rough pipes. But in order to obtain an equation which would represent the velocity distribution for turbulent flow near hydrodynamically rough boundaries, the distance y' must be evaluated in terms of the average height of the surface protrusions k. Again from the experiments by Nikuradse and others, using pipes artificially roughened by cemented coatings of sand grains of diameter k, it has been found that y' is directly proportional to k and $y' = (k/30)$. Introducing this value of y' in Eq. 14.24 and using $\kappa = 0.4$, the following expression is obtained

$$v = \frac{V_*}{0.4} \log_e \left(\frac{30y}{k}\right) \quad ...(14.36)$$

By converting to log base ten and dividing both sides of Eq. 14.36 by V_* it may be rearranged in the following non-dimensional form

$$\frac{v}{V_*} = 5.75 \log_{10} (y/k) + 8.5 \quad ...(14.37)$$

Equation 14.37 represents the velocity distribution for turbulent flow near hydrodynamically rough boundaries. It is known as Karman-Prandtl equation for the velocity distribution near hydrodynamically rough boundaries. Though Eq. 14.37 has been derived to represent the velocity distribution for turbulent flow near rough boundaries only, but on the basis of the Nikuradse's experimental data it has been found that it may be used to represent the velocity distribution in the complete region of turbulent flow in hydrodynamically rough pipes. Accordingly Eq. 14.37 may be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

considered as Karman-Prandtl equation for the velocity distribution for turbulent flow in hydrodynamically rough pipes.

(c) Criterion for Smooth and Rough Pipes. As stated in section 14.4, a pipe boundary will behave as hydrodynamically smooth or rough depending upon the relative magnitude of the average height of the surface protrusions k, and the thickness of the laminar sublayer δ' that is (k/δ') must be the parameter which will determine whether a pipe boundary is hydrodynamically smooth or rough. By substituting the value of δ' from Eq. 14.27 the value of (k/δ') becomes

$$\frac{k}{\delta'} = \frac{V_* k}{\upsilon}\frac{1}{11.6} \qquad ...(14.38)$$

Equation 14.38 thus indicates that the parameter (k/δ') is a function of $(V_* k/\upsilon)$. Thus it may be stated that a pipe boundary may be characterized as hydrodynamically smooth or rough depending upon the value of the parameter $(V_* k/\upsilon)$. On the basis of Nikuradse's experiments the following criterion for hydrodynamically smooth and rough pipes has been given.

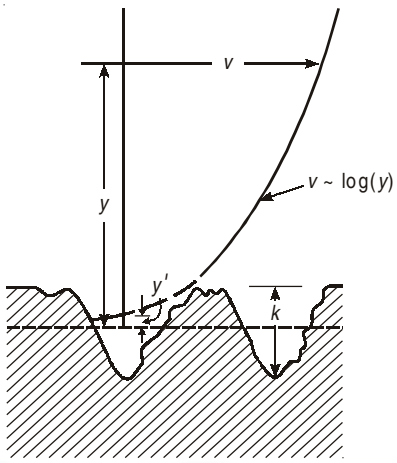


Figure 14.7 *Velocity distribution for turbulent flow near a rough boundary*

For hydrodynamically smooth: $\left(\frac{V_* k}{\upsilon}\right) \le 3$; or $\left(\frac{k}{\delta'}\right) \le 0.25$

For transition: $3 < \left(\frac{V_* k}{\upsilon}\right) < 70$; or $0.25 < \left(\frac{k}{\delta'}\right) < 6.0$

For hydrodynamically rough: $\left(\frac{V_* k}{\upsilon}\right) \ge 70$; or $\left(\frac{k}{\delta'}\right) \ge 6.0$.

14.7 VELOCITY DISTRIBUTION EQUATION FOR TURBULENT FLOW IN TERMS OF MEAN VELOCITY, FOR SMOOTH AND ROUGH PIPES

Equations 14.31 and 14.37 represent the velocity distribution equations for turbulent flow in smooth and rough pipes respectively. From these equations the velocity distribution in terms of the mean velocity of flow V, may be obtained for both smooth and rough pipes as described below.

Consider an elementary ring of thickness dr at a radial distance r from the pipe centre or at a distance y from the pipe boundary as shown in Fig. 14.8. Thus $y = (R - r)$; where R is the radius of the pipe.

Using Eq. 14.31, the mean velocity V, for turbulent flow in smooth pipes is given as follows :

$$V = \frac{Q}{\pi R^2} = \frac{1}{\pi R^2}\int_A (v)dA$$

$$= \frac{1}{\pi R^2}\int_0^R v(2\pi r)dr$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$V = \frac{1}{\pi R^2}\int_0^R 2pr\ V^*\left[5.75\log_{10}\frac{V_*(R-r)}{\upsilon}+5.5\right]dr$$

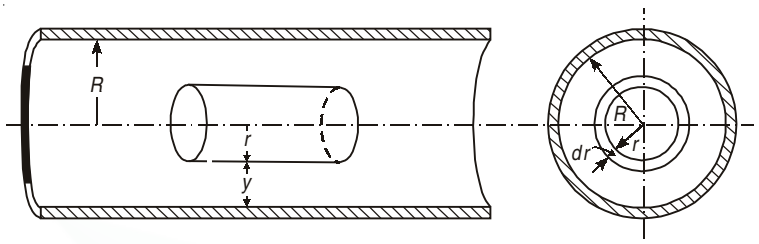


Figure 14.8 *Elementary ring of thickness dr at radial distance r from pipe centre*

On integrating and simplifying the above expression the following relation is obtained for mean velocity V, for turbulent flow in smooth pipe:

$$\frac{V}{V_*} = 5.75\log_{10}\left(\frac{V_*R}{\upsilon}\right)+1.75 \qquad ...(14.39)$$

Similarly using Eq. 14.37, the mean velocity V, for turbulent flow in rough pipes is given by

$$V = \frac{1}{\pi R^2}\int_0^R \pi r\ V_*\left[5.75\log_{10}\frac{(R-r)}{k}+8.5\right]dr$$

Integration of the above expression and subsequent simplification will give the following relation for mean velocity V, for turbulent flow in a rough pipe:

$$\frac{V}{V_*} = 5.75\log_{10}\left(\frac{R}{k}\right)+4.75 \qquad ...(14.40)$$

By subtracting Eq. 14.39 from 14.31 for smooth pipes or Eq. 14.40 from 14.37 for rough pipes the following identical relationship is obtained in both the cases:

$$\frac{v-V}{V_*} = 5.75\log_{10}\left(\frac{y}{R}\right)+3.75 \qquad ...(14.41)$$

Thus when referred to mean velocity of flow, the Karman-Prandtl expressions for velocity distribution in smooth and rough pipes become identical. Therefore, when Eq. 14.41 is used to plot a velocity distribution curve, the pipe wall roughness k need not be known. Further at $y = R$ the velocity v is equal to v_{max}, Eq. 14.41 becomes

$$\frac{v_{max}-V}{V_*} = 3.75 \qquad ...(14.41\ a)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

14.8 RESISTANCE TO FLOW OF FLUID IN SMOOTH AND ROUGH PIPES

When a fluid flows through a pipe frictional resistance is offered to the motion of the fluid which results in the dissipation of certain amount of energy, thereby causing a reduction of pressure intensity in the direction of flow. As derived earlier the loss of head, due to friction in pipes, may be expressed by Darcy-Weisbach equation. But the loss of head due to friction in the case of flow through pipes may be correctly predicted if the friction factor f can be evaluated accurately. By dimensional analysis* it can be shown that the friction factor f depends upon the Reynolds number (VD/v) and the ratio (k/D).

Thus

$$f = \phi\left[\left(\frac{VD}{\upsilon}\right), \left(\frac{k}{D}\right)\right] \qquad ...(14.42)$$

where k is the average height of the pipe wall roughness protrusions, D is the diameter of the pipe and υ is the kinematic viscosity of the fluid flowing in the pipe. The term (k/D) is commonly known as relative roughness. Sometimes (k/D) is replaced by (R/k) where R is the radius of the pipe. The term (R/k) is known as relative smoothness. Equation 14.42 is a general equation which is applicable for laminar as well as turbulent flow in pipes. For laminar flow as well as for turbulent flow with smooth boundary the relative roughness (k/D) is unimportant. As such for both these cases the friction factor f will be a function of only the Reynolds number. On the other hand for a rough boundary, since viscosity of fluid no longer plays a significant role, Reynolds number is unimportant. Hence in the case of turbulent flow with rough boundary the friction factor f will be a function of only the relative roughness (k/D).

(a) Variation of Friction Factor f for Laminar Flow. As derived in Chapter 13, the friction factor f for laminar flow in pipe is given as

$$f = \frac{64}{\text{Re}} \qquad ...(14.43)$$

The above equation shows that for laminar flow the friction factor f varies inversely with Reynolds number and it is independent of the relative roughness (k/D). It has been verified experimentally that upto Reynolds number equal to 2000, which is the limit for the flow to be laminar in circular pipes, the experimental points closely follows the trend of Eq. 14.43 as shown in Fig. 14.9. Hence for the laminar flow in circular pipes the friction factor f may be computed by using Eq. 14.43.

(b) Variation of Friction Factor f for Turbulent Flow. As stated earlier the flow of fluid in circular pipes is turbulent if the Reynolds number is greater than 4000. As such for the Reynolds number more than 2000 but less than 4000 there exists a zone of transition from laminar to turbulent flow. In the transition zone, however, there exists no specific relationship between the friction factor f and the Reynolds number Re.

In the fully developed turbulent flow, as stated earlier, the friction factor f depends on either Re or (k/D) or both, depending on whether the boundary is hydrodynamically smooth or rough or it is in transition. As such in the following paragraphs the relationships for the friction factor f for the different boundaries have been developed.

(c) Variation of Friction Factor for Turbulent Flow in Smooth Pipes. For turbulent flow in smooth pipes the friction factor depends only on the Reynolds number and it is independent of the relative

*See chapter 17.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

roughness (k/D). This is so because in case of hydrodynamically smooth pipes the roughness protrusions are submerged in the laminar sublayer and hence these will have no effect on the friction factor.

In 1911 Blasius studied the experimental data of Saph and Schoder and developed the following empirical equation for turbulent flow with smooth boundaries

$$f = \frac{0.316}{(Re)^{1/4}} \qquad \ldots (14.44)$$

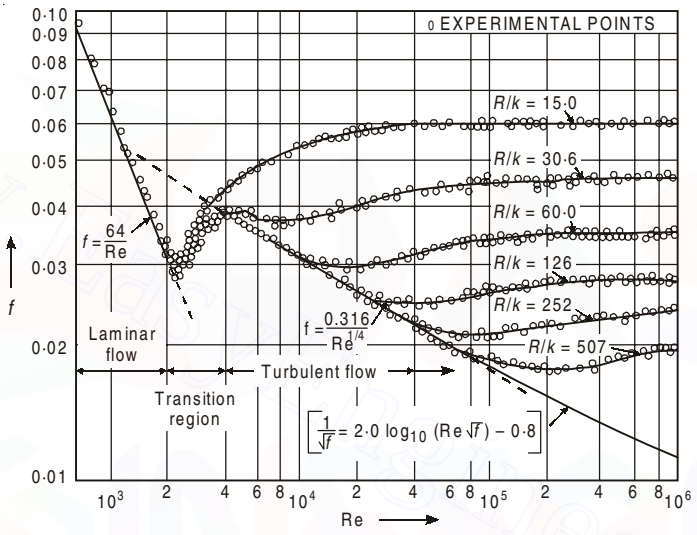


Figure 14.9 *Variation of friction factor with Reynolds number for artificially roughened pipes*

Equation 14.44 shows that the coefficient f varies inversely with the one-fourth power of the Reynolds number. Equation 14.44 has been plotted as shown in Fig. 14.9, using log f and log Re as coordinates. On the same plot Nikuradse's experimental data for smooth pipes is also plotted. It is thus observed that the experimental points follow closely the trend of Eq. 14.44 for Reynolds number varying from 4×10^3 to 10^5. For Reynolds number greater than 10^5, there is an apparent deviation of the experimental points from the straight line represented by Eq. 14.44. It was shown by Nikuradse on the basis of his experimental data that for Reynolds number greater than 10^5 the values of friction factor f given by Blasius equation were lower than the values obtained experimentally. It may thus be concluded that the Blasius equation is applicable for hydrodynamically smooth pipes for Reynolds number from 4×10^3 to 10^5.

The friction factor f for Reynolds numbers greater than 10^5 may be obtained from the expression derived from the logarithmic law of velocity distribution for smooth pipes. For turbulent flow in smooth pipes the mean velocity V of flow has been expressed by Eq. 14.39 as

$$\frac{V}{V_*} = 5.75 \log_{10}\left(\frac{V_* R}{\upsilon}\right) + 1.75 \qquad \ldots (14.39)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $V_* = V\sqrt{\frac{f}{8}}$, so by substituting the value of V_* in Eq. 14.39, it becomes

$$\frac{V}{V(f/8)^{1/2}} = 5.75 \log_{10}\left(\frac{V(f/8)^{1/2}R}{\upsilon}\right) + 1.75$$

which may be simplified and rearranged in the following form

$$\frac{1}{\sqrt{f}} = 2.03 \log_{10}\left(\frac{VD}{\upsilon}\sqrt{f}\right) - 0.91$$

or
$$\frac{1}{\sqrt{f}} = 2.03 \log_{10}\left(\mathrm{Re}\sqrt{f}\right) - 0.91 \quad ...(14.45)$$

However, on the basis of Nikuradse's experimental data for turbulent flow in smooth pipes, it has been indicated that the experimental results, instead of following the trend of Eq. 14.45, follow closely the trend of the following equation:

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\mathrm{Re}\sqrt{f}\right) - 0.8 \quad ...(14.46)$$

A comparison of Eq. 14.45 and 14.46 reveals that there is only a slight difference in the values of the numerical constants of the two equations. Equation 14.46 is known as 'Karman-Prandtl resistance equation' for turbulent flow in smooth pipes. From Nikuradse's experimental measurements equation 14.46 has been found to be valid from Reynolds number 5×10^4 to a Reynolds number as high as 4×10^7. Therefore Eq. 14.46 may be used to predict the value of f, for any arbitrary large Reynolds number of flow beyond the range of available experimental data. However Eq. 14.46 can be solved for f only by trial and error method, because f appears on both sides of this equation. As such the following empirical relationship given by Nikuradse, which fits the experimental data very well, can be used alternatively to obtain the value of f directly.

$$f = 0.0032 + \frac{0.221}{(\mathrm{Re})^{0.237}} \quad ...(14.47)$$

(d) Variation of Friction Factor for Turbulent Flow in Rough Pipes. For turbulent flow in rough pipes the friction factor depends only on the relative roughness and it becomes independent of the Reynolds number. An expression for the friction factors for rough pipes may also be obtained on the same line as for smooth pipes. For turbulent flow in rough pipes the mean velocity V of flow has been expressed by Eq. 14.40 as

$$\frac{V}{V_*} = 5.75 \log_{10}\left(\frac{R}{k}\right) + 4.75 \quad ...(14.40)$$

Again by substituting the value of V_* in Eq. 14.40 it becomes

$$\frac{V}{V(f/8)^{1/2}} = 5.75 \log_{10}\left(\frac{R}{k}\right) + 4.75$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which may be simplified as

$$\frac{1}{\sqrt{f}} = 2.03 \log_{10}\left(\frac{R}{k}\right) + 1.68 \qquad \ldots(14.48)$$

Again Nikuradse's experimental data on turbulent flow in rough pipes has shown that the experimental results instead of following the trend of Eq. 14.48, follow closely the trend of the following equation

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74 \qquad \ldots(14.49)$$

A comparison of Eqs.14.48 and 14.49 indicated that there is only a slight difference in the value of the numerical constants of the two equations. Equation 14.49 is known as 'Karman-Prandtl resistance equation for turbulent flow in rough pipes'. It is indicated from Eq. 14.49 that the friction factor for a rough pipe depends only on the parameter (R/k), and is therefore independent of the Reynolds number. In other words for any given value of the relative pipe wall smoothness (R/k) the friction factor f has a constant value, which is indicated by the horizontal portion of various (R/k) curves in Fig. 14.9.

In Fig. 14.9 are shown the curves representing the variation of friction factor with Reynolds number for artificially roughened pipes based on the results obtained by Nikuradse. The plot of this type with f versus Re for different values of (R/k) on a log-log graph is called 'Stanton diagram' because Thomas E. Stanton was the first to present the experimental data on friction factor in this form.

A study of Fig. 14.9 reveals that the various curves for different values of (R/k) appear to bifurcate, at different points, from the curve for the smooth boundary condition, which may be considered to represent a curve corresponding to (k/R) equal to zero. Therefore it may be stated that in the turbulent flow there is a range of Reynolds number over which a pipe of given relative roughness behave in the same way as a smooth pipe. The rough pipe can then be said to be hydrodynamically smooth in this range and f depends on Re alone as indicated by Eq. 14.44 and 14.46. With further increase in Reynolds number since δ' becomes smaller than k, the boundary can no longer act as smooth boundary. Hence at a certain value of Reynolds number [whose magnitude increases as (R/k) increases], the resistance curve for a rough pipe deviates from that for a smooth pipe and after passing through a transitory phase, it finally becomes horizontal. The horizontal lines, which closely follow the trend of Eq. 14.49, correspond to the pipe boundary being hydrodynamically rough, and show that in this region of turbulent flow, f is independent of Re but depends only on the parameter (R/k). Figure 14.9 indicates that greater is the value of the parameter (R/k), larger is the Reynolds number at which the pipe which originally behaves as a smooth pipe begins to follow the pattern of a rough pipe.

(e) Criteria for Hydrodynamically Smooth and Rough Boundaries. The nature of the boundary *viz.*, smooth, rough or in transition depends on the value of (k/δ'). From Eq. 14.27

$$\delta' = \frac{11.6\upsilon}{V_*}$$

in which $V_* = V\sqrt{(f/8)}$

$$\therefore \quad \delta' = \frac{11.6\upsilon}{V(f/8)^{1/2}} = \frac{11.6\upsilon(\sqrt{8})}{V\sqrt{f}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{\delta'}{R} = \frac{11.6\upsilon(\sqrt{8})}{VR\sqrt{f}} = \frac{11.6\sqrt{8}(2\upsilon)}{VD\sqrt{f}} = \frac{65.6}{\text{Re}\sqrt{f}} \quad \ldots (14.50)$$

Equation 14.50 indicates that the ratio (δ'/R) is inversely proportional to the quantity (Re $\sqrt{f}$).

Further from Eq. 14.50 the value of (k/δ') may be obtained as

$$\frac{k}{\delta'} = \frac{\text{Re}\sqrt{f}}{65.6(R/k)} \quad \ldots (14.51)$$

From Eq. 14.51 it follows that in addition to (k/δ'), $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is another parameter the value of which may also define the nature of the boundary. The criteria for a pipe to behave as hydrodynamically smooth or rough pipe may be established on the basis of Nikuradse's experimental data as indicated below.

Equations 14.46 and 14.49 may be rearranged in the following form:

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}(R/k) = 2.0 \log_{10}\left(\frac{\text{Re}\sqrt{f}}{R/k}\right) - 0.8$$

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}(R/k) = 1.74$$

from which it follows that when the pipe is smooth $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ is a function of $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ and when it is rough $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ attains a constant value of 1.74. In other words by plotting $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ *v/s* $\log_{10}\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ a unique curve would be obtained which would define the nature of the boundary. Thus on the basis of Nikuradse's experimental data a plot of $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ *v/s* $\log_{10}\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is made as shown in Fig. 14.10 from which it is observed that as long as the parameter $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is less than 17 the pipe will behave as hydrodynamically smooth pipe and when $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ becomes greater than 400, then the pipe will behave as hydrodynamically

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

rough pipe. For $17 < \left(\frac{\text{Re}\sqrt{f}}{Rk}\right) < 400$ the pipe boundary will behave as in transition. From Eq. 14.51 the corresponding values of (k/δ') representing the boundary in transition are $0.25 < (k/\delta') < 6$. Therefore a pipe will behave as hydrodynamically smooth pipe if (k/δ') is less than 0.25 and it will behave as hydrodynamically rough pipe when (k/δ') is greater than 6.0.

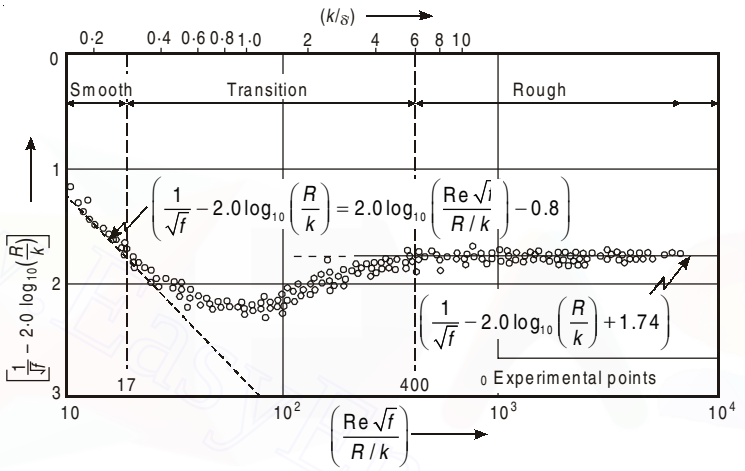


Figure 14.10 *Plot of* $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ v/s $\log_{10}\left(\frac{Re\sqrt{f}}{R/k}\right)$ *for artificially roughened pipes*

(f) Variation of Frictional Factor for Commercial Pipes. Although Nikuradse's experimental data on turbulent flow in smooth and artificially roughened pipes have been used to verify the various theoretically developed relationships of turbulent pipe flow, yet Nikuradse's experimental curves of Fig. 14.9 cannot be used directly to evaluate the friction factor f for commercial pipes. This is so because the actual wall roughness pattern of commercial pipes is very much different from the uniform sand grain roughness used by Nikuradse. However, for commercial pipes it is possible to evaluate the surface roughness of such pipes in terms of a uniform sand grain diameter k, which is then called '*equivalent sand grain roughness*'. In other words for any given pipe it is possible to find a uniform sand grain diameter such that, when the pipe is coated with these sand grains, it will give the same limiting value (i.e., for rough condition) of friction factor f as that for the commercial pipe in its natural form. In order to find the equivalent sand grain roughness k, for any commercial pipe, a series of experiments are conducted on the pipe at sufficiently large Reynolds numbers and the limiting value of the friction factor f is determined from Darcy-Weisbach equation. This value of f is then substituted in Eq. 14.49 and it is solved for (R/k) from which the value of equivalent sand grain roughness k for the pipe may be computed.

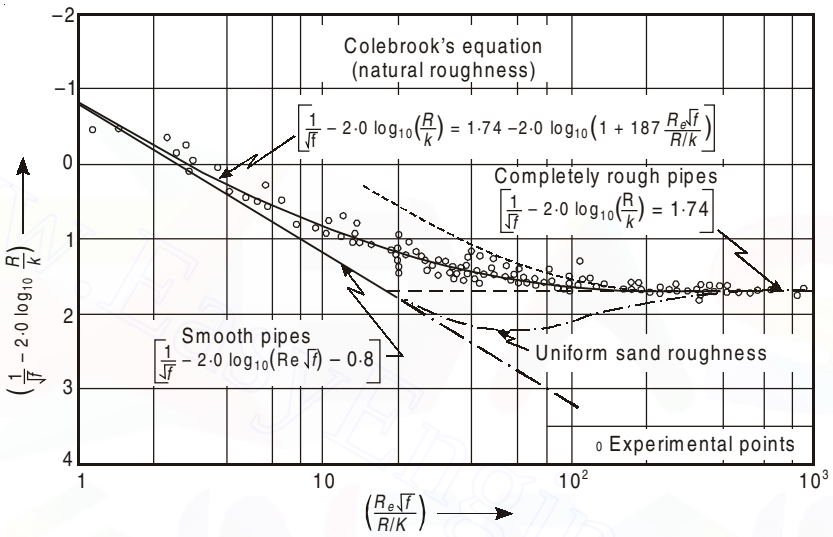
In the transitional region of the boundary where the value of f depends on both Re and k/D, the artificially roughened pipes would produce results totally different from those for commercial pipes. This difference can be seen from Fig. 14.11 in which test results from both Nikuradse's artificially roughened pipes and commercial pipes are plotted. The plotting of the experimental results for the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

commercial pipes as shown in Fig. 14.11 was carried out by C.F. Colebrook, and it is seen that the experimental data for commercial pipes in the transition region deviate from Nikuradse's data on turbulent flow in artificially roughened pipes. The coordinates of Fig. 14.11 are chosen to be

$$\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}\left(\frac{R}{k}\right)\right] \text{ and } \log_{10}\left(\frac{\mathrm{Re}\sqrt{f}}{R/k}\right)$$



Fligure14.11 *Plot of* $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ *versus* $\log_{10}\left(\frac{Re\sqrt{f}}{R/k}\right)$ *for commercial pipes in transition region*

so that for smooth pipes in accordance with Eq. 14.46 a straight line with a slope of 2 is obtained and for rough pipes in accordance with Eq. 14.49 a horizontal line is obtained. The experimental points for commercial pipes in the transition region are seen to follow closely the trend of a single curve which is asymptotic to the plots corresponding to the equations for both the smooth and the rough pipes. Colebrook and White developed an empirical equation for this curve which represents the variation of the friction factor *f* for commercial pipes. It is of the form

$$\frac{1}{\sqrt{f}} - 2.0\log_{10}\left(\frac{R}{k}\right) = 1.74 - 2.0\log_{10}\left(1 + 18.7\frac{R/k}{\mathrm{Re}\sqrt{f}}\right) \quad \ldots (14.52)$$

Equation 14.52 may therefore be used to predict the friction factor *f* for commercial pipes.

L.F. Moody has plotted Eq. 14.52 as shown in Fig. 14.12 which is in the form of *f* versus Re curves for various values of (*R*/*k*). Figure 14.12 is commonly known as 'Moody's diagram' which is essentially the same as Nikuradse's plot (Fig. 14.9) except for the transition regions. For any turbulent pipe flow problem the value of friction factor *f* can therefore be determined from Moody's diagram (Fig. 14.12) if the numerical values of *R*/*k* for the pipe and Re of flow are known.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

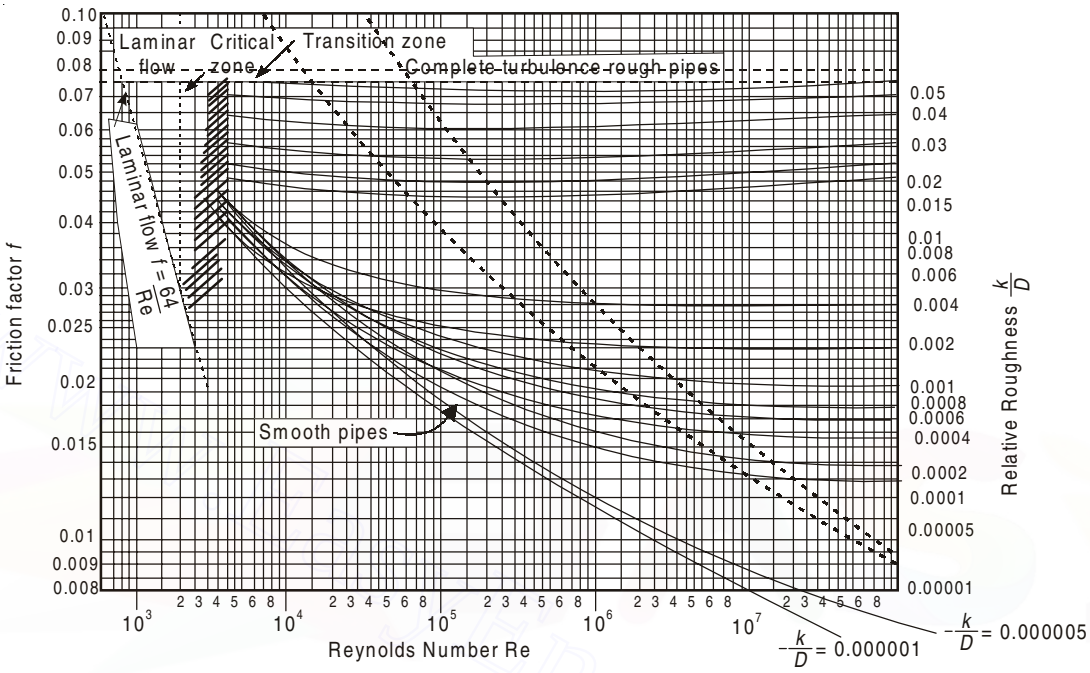


Figure 14.12 *Moody's diagram for friction factor for commercial pipes*

The values of k which may be adopted for the pipes of some of the common materials are given below in Table 14.1.

TABLE 14.1

Pipe Materials	*k in mm*
1. Glass, Brass, Copper, Lead	Smooth
2. Steel, Wrought iron	0.045
3. Asphalted cast iron	0.120
4. Galvanized iron	0.150
5. Cast iron	0.260
6. Concrete	0.30 to 3.0
7. Rivetted steel	0.90 to 9.0

The value of equivalent sand grain roughness k given in Table 14.1 correspond to material in new and clean condition. As the pipes become older the roughness increases due to corrosion. Colebrook and White have shown that the boundary roughness may increase with time approximately in accordance with the following expression

$$k = k_0 + \alpha t \qquad \ldots(14.53)$$

in which k_0 is the equivalent sand grain roughness of the new pipe material, k is the equivalent sand grain roughness at any time t, and α is the time rate of increase of roughness. Resistance measurements taken on the same pipe at two different times will enable the evaluation of the constant α which may be used to estimate the value of k at any future time.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

14.9 TYPES OF PROBLEMS IN PIPELINE DESIGNS

In practice there are three types of problems which occur most frequently in the design of pipelines as mentioned below:

(1) Determination of the head lost due to friction h_f from the known values of discharge Q, length of pipe L, pipe diameter D, roughness k, and kinematic viscosity υ of the flowing fluid.

(2) Determination of discharge Q from the known values of head loss due to friction h_f, pipe diameter D, length of pipe L, roughness k, and kinematic viscosity υ of the flowing fluid.

(3) Determination of the diameter D of a pipe from the known values of discharge Q, head loss due to friction h_f, length of pipe L, roughness k and kinematic viscosity υ of the flowing fluid.

In the first case the Reynolds number Re and the relative roughness k/D can be calculated from the given data. For these values of Re and (k/D), the value of friction factor f can be found from Moody's diagram (Fig. 14.12) or from the relevant resistance equations.

In the second case the relative roughness (k/D) is known and the parameter $(\mathrm{Re}\sqrt{f})$ can be computed from the following expression

$$\mathrm{Re}\sqrt{f} = \frac{VD}{\upsilon}\left(\frac{2gh_fD}{V^2L}\right)^{1/2} = \frac{D^{3/2}}{\upsilon}\left(\frac{2gh_f}{L}\right)^{1/2} \quad \ldots(14.54)$$

From these values of (k/D) and $(\mathrm{Re}\sqrt{f})$, the value of friction factor f can be found from Moody's diagram (Fig. 14.12) or from the relevant resistance equation.

In the third case, since the diameter of the pipe is unknown, the relative roughness (k/D), Reynolds number and friction factor f are also unknown. As such only trial and error solution is possible. The procedure will be to first assume a suitable value of f and compute the value of diameter D from Darcy-Weisbach equation. This will give a trial value of D. With this trial value of D, values of Re and (k/D) can be computed to find a new value of f from Moody's diagram (Fig. 14.12). The new values of f is used for the next trial and the procedure is repeated until the value of f does not change.

The illustrative examples below indicate the above discussed methods for solving the different pipeline problems.

14.10 FRICTION IN NON-CIRCULAR CONDUITS

Mostly the closed conduits (or pipes) used in engineering practice are of circular cross-section. However, sometimes non-circular conduits, such as rectangular air ducts are also employed in engineering installations, for which the friction loss is required to be estimated. Experiments have shown that the expressions developed for turbulent flow in circular conduits may be applied to non-circular conduits of many shapes, if in place of the diameter D of the circular conduit an 'equivalent diameter' is used, such that the hydraulic mean depth m is the same. For a circular conduit flowing full $m = (D/4)$ and so the equivalent diameter must equal 4 m, (i.e., D must be replaced by four times the hydraulic mean depth). The concept of equivalent diameter is, however, not applicable to laminar flow in non-circular conduits and also to non-circular conduits whose cross-sections deviate considerably from the circular section even if the flow is turbulent.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ILLUSTRATIVE EXAMPLES

Example 14.1. *If mixing length l is given by*

$$l = \frac{\kappa(dv/dy)}{(d^2v/dy^2)}$$

and $\tau = \tau_0\left(l - \frac{y}{R}\right)$, *derive the expression for velocity distribution for a fully developed turbulent flow in a circular pipe of radius R.*

Solution

From Eq. 14.9, we have

$$\tau = \rho l^2 \left(\frac{dv}{dy}\right)^2$$

For small values of y, we have

$$\tau = \tau_0\left(1 - \frac{y}{R}\right) \approx \tau_0$$

Thus
$$\tau_0 = \rho l^2 \left(\frac{dv}{dy}\right)^2$$

Introducing the given value of l, we get

$$\tau_0 = \frac{\rho\kappa(dv/dy)^4}{(d^2v/dy^2)^2}$$

Dividing both sides by ρ and taking square root of both sides, we get

$$\sqrt{\frac{\tau_0}{\rho}} = \pm\frac{\kappa(dv/dy)^2}{(d^2v/dy^2)}$$

or
$$V_* = \pm\frac{\kappa(dv/dy)^2}{(d^2v/dy^2)}$$

or
$$\frac{d^2v}{dy^2} = \pm\frac{\kappa}{V_*}\left(\frac{dv}{dy}\right)^2$$

Since (d^2v/dy^2) will be negative, only negative sign is taken on the R.H.S. Thus

$$\frac{d^2v}{dy^2} = -\frac{\kappa}{V_*}\left(\frac{dv}{dy}\right)^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Let $$\frac{dv}{dy} = z; \quad \therefore \quad \frac{d^2v}{dy^2} = \frac{dz}{dy}$$

Thus $$\frac{dz}{dy} = -\frac{\kappa}{V_*} z^2$$

or $$\frac{dz}{z^2} = -\frac{\kappa}{V_*} dy$$

Integrating both sides of the above equation, we get

$$-\frac{1}{z} = -\frac{\kappa}{V_*} y$$

or $$z = \frac{V_*}{\kappa} \times \frac{1}{y}$$

or $$\frac{dv}{dy} = \frac{V_*}{x} \times \frac{1}{y}$$

or $$dv = \frac{V_*}{\kappa} \times \frac{1}{y} dy$$

Integrating both sides of the above equation, we get

$$v = \frac{V_*}{\kappa} \log_e y + C$$

The constant of integration C may be determined from the boundary condition at the centre of the pipe :

at $y = R$

$v = v_{max}$ (maximum velocity)

Thus $$C = v_{max} - \frac{V_*}{\kappa} \log_e R$$

Introducing this value of C, the expression for velocity distribution for a fully developed turbulent flow in a circular pipe becomes

$$v = v_{max} + \frac{V_*}{\kappa} \log_e (y/R)$$

or $$\frac{v - v_{max}}{V_*} = \frac{1}{\kappa} \log_e (y/R)$$

Taking $\kappa = 0.4$ and changing from natural to common logarithms the above equation becomes

$$\frac{v - v_{max}}{V_*} = 5.75 \log_{10} (y/R)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 14.2. *At a point in a turbulent flow field the instantaneous values of u and v velocity components measured at an interval of 0.05 seconds are listed below.*

u (mm/s)	+105	+110	+84	+89	+102	+94	+111	+101	+87	+ 95	+89
v (mm/s)	–3	–16	+11	+25	–6	–20	–20	+4	+21	–2	+6

Determine $\bar{u}$, $\bar{v}$, $\overline{u'v'}$ *and local value of Reynolds' shear stress. Take* $\bar{\rho} = 1.23 kg/m^3$.

Solution

The computations are performed in tabular form as shown below.

n	*u*(*mm/s*)	$u' = (u - \bar{u})$	*v* (*mm/s*)	$v' = (v - \bar{v})$	$u'v'$
1	+105	+8	–3	–3	–24
2	+110	+13	–16	–16	–208
3	+84	–13	+11	+11	–143
4	+89	–8	+25	+25	–200
5	+102	+5	–6	–6	–30
6	+94	–3	–20	–20	+60
7	+111	+14	–20	–20	–280
8	+101	+4	+4	+4	+16
9	+87	–10	+21	+21	–210
10	+95	–2	–2	–2	+4
11	+89	–8	+6	+6	–48
	$\Sigma u = 1067$		$\Sigma v = 0$		$\Sigma(u'v') = -1063$
	$\therefore \bar{u} = \frac{1067}{11}$		$\bar{v} = 0$		$\overline{u'v'} = -\frac{1063}{11}$
	= 97 mm/s				= – 96.64

Reynold's shear stress

$$\tau = \overline{\rho u'v'}$$

$$= (1.23 \times 96.64) = 118.87 \text{ N/m}^2$$

Example 14.3. *For a turbulent flow in pipes determine the value of y at which the point velocity is equal to the mean velocity of flow.*

Solution

From Eq. 14.41

$$\frac{v - V}{V_*} = 5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75$$

for $v = V$

$$5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75 = 0$$

or $$\left(\frac{y}{R}\right) = 0.223 \quad \text{i.e., } y = 0.223\, R.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 14.4. *For turbulent flow in pipes (smooth as well as rough) show that*

$$\frac{v_{max}}{V} = 1.33\sqrt{f} + 1$$

Solution

For turbulent flow in pipes from Eq. 14.41

$$\frac{v - V}{V_*} = 5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75$$

which is applicable to both smooth and rough pipes.

For $y = R, v = v_{max}$

Thus

$$\frac{v_{max} - V}{V_*} = 3.75$$

But $V_* = V\sqrt{(f/8)}$

Hence

$$\frac{v_{max} - V}{V\sqrt{(f/8)}} = 3.75$$

or $$\frac{v_{max}}{V} = 3.75\sqrt{(f/8)} = 1.33\sqrt{f} + 1$$

which is the required expression.

Note. It may however be stated that Nikuradse's experimental data instead of following the trend of above derived equation, closely follow the trend of the following equation

$$\frac{v_{max}}{V} = 1.43\sqrt{f} + 1.$$

Example 14.5. *A smooth brass pipeline 75 mm in diameter and 900 m long carries water at the rate of 7 litres per second. If the kinematic viscosity of water is 0.0195 stokes, calculate the loss of head, wall shearing stress, centre line velocity, shear stress and velocity at 25 mm from the centre line and the thickness of the laminar sublayer. Take $\rho = 1000$ kg/m^3 (10^2 msl/m^3).*

Solution

Mean velocity $$V = \frac{Q}{A} = \frac{7 \times 10^{-3}}{\frac{\pi}{4}(0.075)^2} = 1.584 \text{ m/s}$$

Reynolds number $$\frac{VD}{\upsilon} = \frac{1.584 \times 0.075}{0.0195 \times 10^{-4}} = 6.09 \times 10^4$$

Hence the flow is turbulent. For this value of Reynolds number of flow in smooth pipes, the value of friction factor f is given by Blasius equation as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$f = \frac{0.316}{(\text{Re})^{1/4}} = \frac{0.316}{(6.09\times10^4)^{1/4}} = 0.0201$$

From Darcy-Weishbach equation the head loss is given as

$$h_f = \frac{fLV^2}{2gD} = \frac{0.0201\times900\times(1.584)^2}{2\times9.81\times0.075} = 31 \text{ m}$$

Wall shearing stress is given as

SI units

$$\tau_0 = \frac{\rho V^2 f}{8} = \frac{1000\times(1.584)^2\times0.0201}{8} = 6.304 \text{ N/m}^2$$

Metric units

$$\tau_0 = \frac{\rho V^2 f}{8} = \frac{102\times(1.584)^2\times0.0201}{8} = 0.643 \text{ kg(f)/m}^2$$

Shear velocity, $V_* = V\sqrt{\frac{f}{8}} = 1.584\sqrt{\frac{0.0201}{8}} = 0.079\,4 \text{ m/s}$

For turbulent flow in smooth pipes from Eq. 14.31, we have

$$\frac{v}{V_*} = 5.75\log_{10}\left(\frac{V_* y}{\upsilon}\right) + 5.5$$

For $y = \frac{D}{2} = \frac{0.075}{2} = 0.0375$ m, $v = v_{\max}$ (i.e., centre line velocity)

Thus by substitution, we get

$$\frac{v_{\max}}{0.0794} = 5.75\log_{10}\left(\frac{0.0794\times0.0375}{0.0195\times10^{-4}}\right) + 5.5$$

or $v_{\max} = 1.89$ m/s

From Eq. 14.17, we have

$$\tau = \tau_0 \frac{r}{R}$$

Thus shear stress at $r = 0.025$ m is obtained as

SI units

$$\tau = 6.304\left(\frac{0.025}{0.0375}\right) = 4.203 \text{ N/m}^2$$

Metric units

$$\tau = 0.643\left(\frac{0.025}{0.0375}\right) = 0.429 \text{ kg (f)/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again from Eq. 14.31, for

$$y = (0.0375 - 0.025) = 0.0125 \text{ m}$$

$$\frac{v}{0.0794} = 5.75 \log_{10}\left(\frac{0.0794 \times 0.0125}{0.0195 \times 10^{-4}}\right) + 5.5$$

or $$v = 1.672 \text{ m/s}$$

Thickness of laminar sublayer is given by Eq. 14.27 as

$$\delta' = \frac{11.6\upsilon}{V_*} = \frac{11.6 \times 0.0195 \times 10^{-4}}{0.0794} = 2.85\ 10^{-4} \text{ m}$$

Example 14.6. *Assuming the following velocity distribution in a circular pipe,*

$$v = v_{max}\left(1 - \frac{r}{R}\right)^{1/7}$$

where v_{max} = maximum velocity, R = radius of pipe, v = velocity at a radius r from the centre of pipe, calculate:

(i) The ratio of the mean and the maximum velocity ;

(ii) The radius at which the actual velocity equals the mean velocity.

Solution

Consider an elementry ring of thickness dr at a radial distance r from the centre of the pipe as shown in Fig. 14.8.

The discharge dQ through the elementry ring is given by

$$dQ = v(2\pi r)dr$$

$\therefore$ Total discharge $$Q = \int_0^R dQ$$

$$= \int_0^R v(2\pi r)dr$$

$$= \int_0^R v_{\max}\left(1 - \frac{r}{R}\right)^{1/7} \times 2\pi r\, dr$$

$$= 2\pi v_{\max} \int_0^R r\left(1 - \frac{r}{R}\right)^{1/7} dr$$

$$= \frac{49}{60} \times \left(\pi R^2\right) \times v_{\max}$$

Mean velocity of flow $$V = \frac{Q}{\pi R^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V = \frac{49}{60} v_{max}$$

or $$\frac{V}{v_{max}} = \frac{49}{60} = 0.817$$

For $$v = V$$

$$V = v_{max}\left(1 - \frac{r}{R}\right)^{1/7}$$

or $$\frac{V}{v_{max}} = \left(1 - \frac{r}{R}\right)^{1/7} = \frac{49}{60}$$

$$\therefore \quad r = 0.7579\, R$$

i.e., actual velocity is equal to mean velocity at a radius of 0.7579 R from the centre of the pipe.

Alternataively if y is the distance from the pipe boundary, then since $y = (R - r)$ and hence

$$y = (1 - 0.7579)\, R = 0.2421R$$

i.e., actual velocity is equal to mean velocity at a distance of 0.2421 R from the pipe.

Example 14.7. *Mean point velocities measured with the help of a pitot tube at mid-point and quarter point of a 0.2 m diameter pipe were found to be 1.50 m/s and 1.35 m/s respectively. If the flow in the pipe is turbulent, determine the discharge, friction factor and average height of roughness projections.*

Solution

From Eq. 14.23, we have

$$\frac{v_{max} - v}{V_*} = 5.75 \log_{10}\left(\frac{R}{y}\right)$$

$v_{max} = 1.50$ m/s; $v = 1.35$ m/s ; $R = 0.1$ m; and

$y = (R - r) = (0.1 - 0.05) = 0.05$ m

Thus by substitution, we get

$$\frac{1.50 - 1.35}{V_*} = 5.75 \log_{10}\left(\frac{0.1}{0.05}\right)$$

$$\therefore \quad V_* = 0.0867 \text{ m/s}$$

From Eq. 14.41 (a), we have

$$\frac{V_{max} - V}{V_*} = 3.75$$

By substituting the given values, we get

$$\frac{1.50 - V}{0.0867} = 3.75 \text{ ; or } V = 1.175 \text{ m/s}$$

Thus discharge $$Q = \frac{\pi D^2}{4} \times V$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{\pi(0.2)^2}{4} 1.175 = 0.037 \text{ m3/s}$$

Also $V_* = V\sqrt{f/8}$

or $0.0867 = 1.175\sqrt{f/8}$

$\therefore \quad f = 0.0436$

From Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

or $$\frac{1}{\sqrt{0.0436}} = 2.0 \log_{10}\left(\frac{0.1}{k}\right) + 1.74$$

$\therefore \quad k = 2.99\ 10^{-3}$ m = 2.99 mm

Example 14.8. *A rough pipeline of diameter 0.1 m carries water at 20°C at the rate of 50 l/s. If the average height of the protrusions on the pipe surface is 0.15 mm, calculate the friction factor, maximum velocity, shear stress at the pipe surface and the shear velocity. For water at 20°C take* $\upsilon = 1.0 \times 10^{-6}$ *m²/s and* $\rho = 1000$ *kg/m³ (102 msl/m³).*

Solution

Mean velocity $V = \frac{Q}{A} = \frac{50\times10^{-3}}{\frac{\pi}{4}(0.1)^2} = 6.37$ m/s

Reynolds number $\text{Re} = \frac{VD}{\upsilon} = \frac{6.37\times0.1}{1.0\times10^{-6}} = 6.37\times 10^5$

Hence the flow is turbulent in the rough pipe.

For rough turbulent flow from Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

$R = 0.05$ m; and $k = 0.15\ 10^{-3}$ m

Thus by substitution, we get

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{0.05}{0.15\times10^{-3}}\right) + 1.74$$

or $f = 0.0217$

Shear stress at the pipe surface is given by $\tau_0 = \frac{\rho V^2 f}{8}$

In SI units

$$\tau_0 = \frac{1000\times(6.37)^2\times0.0217}{8} = 110.065 \text{ N/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In Metric units

$$\tau_0 = \frac{102 \times (6.37)^2 \times 0.0217}{8} = 11.227 \text{ kg(f)/m}^2$$

Shear velocity $$V_* = \sqrt{\frac{\tau_0}{\rho}} = \sqrt{\frac{110.065}{1000}} = 0.332 \text{ m/s}$$

Also $$V_* = \sqrt{\frac{11.227}{102}} = 0.332 \text{ m/s}$$

From Eq. 14.37 the velocity distribution in rough pipe is given as

$$\frac{v}{V_*} = 5.75 \log_{10}\left(\frac{y}{k}\right) + 8.5$$

For $y = R$, $v = v_{max}$

Hence by substitution, we get

$$\frac{v_{max}}{0.332} = 5.75 \log_{10}\left(\frac{0.05}{0.15 \times 10^{-3}}\right) + 8.5$$

$\therefore$ $$v_{max} = 7.64 \text{ m/s}$$

Example 14.9. *A liquid of specific gravity 0.88 and absolute viscosity 6.533 10^{-4} N.s/m^2 flows through a pipe of diameter 0.15 m at the rate of 60 litres per second. If the loss of head in 100 m length of pipe is 4.56 m, determine whether the pipe is rough or smooth.*

Solution

Mean velocity $$V = \frac{Q}{A} = \frac{60 \times 10^{-3}}{\frac{\pi}{4}(0.15)^2} = 3.40 \text{ m/s}$$

Reynolds number $$\text{Re} = \frac{\rho VD}{\mu}$$

$$= \frac{0.88 \times 1000 \times 3.4 \times 0.15}{6.533 \times 10^{-4}} = 6.87 \times 10^5$$

Hence the flow is turbulent.

According to Darcy-Weisbach equation

$$h_f = \frac{fLV^2}{2gD}$$

By substituting the given values, we get

$$4.56 = \frac{f \times 100 \times (3.40)^2}{2 \times 9.81 \times 0.15}$$

or $$f = 0.0116$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Let the pipe be assumed as hydrodynamically rough, then from Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

By substitution, we have

$$\frac{1}{\sqrt{0.0116}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

or $$(R/k) = 5921$$

Thus $$\left[\frac{\text{Re}\sqrt{f}}{R/k}\right] = \frac{6.87 \times 10^5 \times (0.0116)^{1/2}}{5921} = 12.5 \approx 13$$

Since the parameter $\left[\frac{\text{Re}\sqrt{f}}{R/k}\right]$ is less than 17, the assumption made for the pipe to be rough is not correct. Hence the pipe is hydrodynamically smooth.

Example 14.10. *Find the rate of flow of water at 40°C through a cast iron pipe of 100 mm diameter, if the loss of head is 1.05 m per 100 m length of the pipe. The average height of the roughness projection on the pipe wall is 0.25 mm and υ for water at 40°C is 7.5 × 10^{-3} stokes.*

Solution

$$\frac{R}{k} = \frac{50}{0.25} = 200$$

From Eq. 14.54, we have

$$\text{Re}\sqrt{f} = \frac{D^{3/2}}{\upsilon}\sqrt{\frac{2gh_f}{L}}$$

$$\text{Re}\sqrt{f} = \frac{(0.1)^{3/2}}{7.5 \times 10^{-3} \times 10^{-4}}\left[\frac{2 \times 9.81 \times 1.05}{100}\right]^{1/2}$$

$$= 1.914 \times 10^4$$

$$\therefore \quad \left[\frac{\text{Re}\sqrt{f}}{R/k}\right] = \frac{1.914 \times 10^4}{200} = 95.7$$

Since the parameter is more than 17 but less than 400, the boundary acts in transition. Thus from Eq. 14.52, we have

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}\left(\frac{R}{k}\right) = 1.74 - 2.0 \log_{10}\left(1 + 18.7\frac{R/k}{\text{Re}\sqrt{f}}\right)$$

or $$\frac{1}{\sqrt{f}} - 2.0 \log_{10}(200) = 1.74 - 2.0 \log_{10}\left(1 + \frac{18.7 \times 200}{1.914 \times 10^4}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $f = 0.0261$

Now by substituting in Darcy-Weisbach equation, we have

$$1.05 = \frac{0.0261 \times 100 \times V^2}{2 \times 9.81 \times 0.1}$$

or $V = 0.888 \text{ m/s}$

$\therefore$ Discharge $Q = \frac{\pi D^2}{4} \times V$

$$= \frac{\pi \times (0.1)^2}{4} \times 0.888$$

$$= 0.007 \text{ m}^3/\text{s} = 7 \text{ } l/\text{s}$$

Example 14.11. *A pipe of diameter 0.1m carries water at 15°C. If sand grains 0.75 mm in diameter are cemented on the inner surface of the pipe for test purposes, at what velocity of water will (a) the surface roughness just begins to disturb the laminar sublayer, and (b) the pipe wall behaves as rough boundary. Kinematic viscosity of water at 15°C is 1.14 × 10^{-2} stokes.*

Solution

$$\frac{R}{k} = \frac{0.05}{0.75 \times 10^{-3}} = 66.67$$

(a) For smooth pipes from Blasius equation, we have

$$f = \frac{0.316}{\text{Re}^{1/4}}$$

and in the limiting condition the parameter $\left[\frac{\text{Re}\sqrt{f}}{R/k}\right]$ is equal to 17,

or $\left(\text{Re}\sqrt{f}\right) = (17 \times 66.67)$

By substituting the value of f from the Blasius equation, we get

$$(\text{Re})^{7/8} = \frac{17 \times 66.67}{\sqrt{0.316}} = 2016$$

$\therefore$ $\text{Re} = 5978$

Thus the velocity at which the surface roughness will just begin to disturb the laminar sublayer is

$$V = \frac{Re\upsilon}{D} = \frac{5978 \times 1.14 \times 10^{-2} \times 10^{-4}}{0.1} = 0.068 \text{ m/s}$$

(b) For rough pipes from Eq. 14.49, we have

$$\sqrt{f} = \frac{1}{2\log_{10}\left(\frac{R}{k}\right) + 1.74}$$

or $\sqrt{f} = \frac{1}{2\log_{10}(66.67) + 1.74} = 0.1856$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In the limiting condition the parameter $\left[\frac{\text{Re}\sqrt{f}}{R/k}\right] = 400$

or $$\text{Re} = \frac{400 \times 66.67}{0.1856} = 1.44 \times 10^5$$

Hence the minimum velocity at which the pipe wall will behave as rough boundary is

$$V = \frac{\text{Re}\,\upsilon}{D} = \frac{1.44 \times 10^5 \times 1.14 \times 10^{-2} \times 10^{-4}}{0.1} = 1.642 \text{ m/s}$$

Example 14.12. *Determine the diameter of a cast iron pipe which is required to carry water at 10°C at the rate of 250 litres per second, if the loss of head is not to exceed 2 m per 100 m length of pipe. The average height of the pipe wall projection is 0.36 mm and kinematic viscosity of water at 10°C is 0.013 stokes.*

Solution

Discharge $Q = 250\ \ 10^{-3} = 0.25 \text{ m}^3/\text{s}$

Assume $f = 0.018$

From Darcy-Weisbach equation, we have

$$h_f = \frac{fLV^2}{2gD} = \frac{8fLQ^2}{\pi^2 g D^5}$$

or $$D^5 = \frac{8fLQ^2}{\pi^2 g h_f} = \frac{8 \times 0.018 \times 100 \times (0.25)^2}{\pi^2 \times 9.81 \times 2} = 465 \times 10^{-5}$$

or $$D = 0.342 \text{ m}$$

Mean velocity $$V = \frac{Q}{A} = \frac{0.25}{\frac{\pi}{4}(0.342)^2} = 2.721 \text{ m/s}$$

Reynolds number $$\text{Re} = \frac{VD}{\upsilon} = \frac{2.721 \times 0.342}{0.013 \times 10^{-4}} = 7.16 \times 10^5$$

Relative roughness $$\frac{k}{D} = \frac{0.36 \times 10^{-3}}{0.342} = 0.001\ 05$$

From Fig 14.12 the value of f corresponding to these values of Re and (k/D) is obtained as $f = 0.02$.

Computing a new value of D based on $f = 0.02$

$$D^5 = \frac{8fLQ^2}{\pi^2 g h_f} = \frac{8 \times 0.02 \times 100 \times (0.25)^2}{\pi^2 \times 9.81 \times 2} = 516 \times 10^{-5}$$

or $$D = 0.349 \text{ m}$$

Thus $$V = \frac{0.25}{\frac{\pi}{4}(0.349)^2} = 2.613 \text{ m/s}$$

$$\text{Re} = \frac{VD}{\upsilon} = \frac{2.613 \times 0.349}{0.013 \times 10^{-4}} = 7.01 \times 10^5$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$\frac{k}{D} = \frac{0.36\times10^{-3}}{0.349} = 0.00103$$

Again from Fig. 14.12 for these values of Re and (k/D), $f = 0.0198$ which agrees with the assumed value of f.

Hence $$D = 0.349 \text{ m}$$

Note. The next larger standard size of pipe should be selected.

Example 14.13. *Design the diameter of a steel pipe to carry oil having kinematic viscosity of 0.20 stokes with a velocity of 1.5 m/s, if Darcy's friction factor over almost the whole turbulent range is given by*

$$f = 0.0055\left[1+\left(20000\frac{\varepsilon}{D}+\frac{10^6}{Re}\right)^{0.33}\right]$$

in which ε *= equivalent sand grain roughness, D = diameter of pipe, Re = Reynolds number.*

The head loss is to be limited to 0.4 m per 100 metres of pipe. The equivalent sand grain roughness for steel may be assumed as 0.045 mm.

Solution

If D is the diameter of the pipe, then

$$\text{Re} = \frac{VD}{\upsilon} = \frac{1.5\times D}{0.20\times10^{-4}} = 7.5\times10^4\, D$$

$$\varepsilon = 0.045 \text{ mm} = 0.045\times10^{-3} \text{ m}$$

Substituting these values in the given expression for f it becomes

$$f = 0.0055\left[1+\left(\frac{0.9}{D}+\frac{13.33}{D}\right)^{0.33}\right]$$

or $$f = 0.0055+\frac{0.0132}{D^{0.33}} \quad \text{... (i)}$$

Further according to Darcy-Weisbach equation

$$h_f = \frac{fLV^2}{2gD}$$

or $$\frac{0.4}{100} = \frac{f\times(1.5)^2}{2\times9.81\times D}$$

or $$f = 0.0349\, D \quad \text{... (ii)}$$

Solving Eqs (i) and (ii) by trial the pipe diameter D may be obtained. However, the pipe diameter which may be adopted should be such that corresponding to this value of the diameter, the value of f obtained by Eq. (i) is slightly less than that obtained by Eq. (ii), so that the head loss will be less than the given limiting value. Accordingly the value of the pipe diameter which may be adopted is

$$D = 0.625 \text{ m}$$

However, if the pipe of diameter 0.625 m is not available, the next larger standard size of the pipe may be adopted, which would have head loss less than the given limiting value.

Example 14.14. *A pipeline of diameter 0.3 m carries oil at the rate of 540 l/s. If the specific gravity of the oil is 0.80 and its kinematic viscosity is equal to 0.023 stokes, determine (i) the maximum permissible height of the*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

protrusions upto which the pipe acts as smooth pipe, (ii) the height of the protrusions beyond which the pipe would become rough.

Solution

Mean velocity, $V = \frac{Q}{A} = \frac{540 \times 10^{-3}}{(\pi/4)(0.3)^2} = 7.64 \text{ m/s}$

Reynolds number $\text{Re} = \frac{VD}{\upsilon} = \frac{7.64 \times 0.3}{0.023 \times 10^{-4}} = 9.97 \times 10^5$

(i) For turbulent flow in smooth pipes, from Eq. 14.46, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10} \left(\text{Re}\sqrt{f}\right) - 0.8$$

or $$\frac{1}{\sqrt{f}} = 2.0 \log_{10} \left(9.97 \times 10^5 \sqrt{f}\right) - 0.8$$

or $$\frac{1}{\sqrt{f}} = 2.0 \log_{10} \sqrt{f} + 11.2$$

Solving by trail, we obtain

$$f = 0.0116$$

Alternatively for turblent flow in smooth pipes, from Eq. 14.47, we have

$$f = 0.0032 + \frac{0.221}{(\text{Re})^{0.237}}$$

or $$f = 0.0032 + \frac{0.221}{(9.97 \times 10^5)^{0.237}}$$

or $$f = 0.01157 \approx 0.0116$$

Thus $$\left(\text{Re}\sqrt{f}\right) = (9.97 \times 10^5) \times \sqrt{0.0116}$$
$$= 1.074 \times 10^5$$

The pipe will act as smooth pipe as long as

$$\left(\frac{\text{Re}\sqrt{f}}{R/k}\right) < 17$$

or $$\frac{R}{k} > \frac{1.074 \times 10^5}{17}$$

or $$k < \frac{17 \times 0.15}{1.074 \times 10^5}$$

or $$k < 2.37 \times 10^{-5} \text{ m}$$

or $$k < 0.0237 \text{ mm}$$

Hence the maximum permissible height of the surface protrusions = 0.0237 mm.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(ii) The pipe will act as rough when

$$\left(\frac{\text{Re}\sqrt{f}}{R/k}\right) \geq 400$$

or $$\sqrt{f} \geq \frac{400(R/k)}{9.97\times10^5}$$

For rough turbulent flow from Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}(R/k) + 1.74$$

or $$\frac{9.97\times10^5}{400(R/k)} = 2.0 \log_{10}(R/k) + 1.74$$

Solving by trail $$\frac{R}{k} = 363.3$$

$$\therefore \quad k = \frac{0.15}{363.3}$$

$$= 4.13 \times 10^{-4} \text{ m} = 0.413 \text{ mm}$$

Hence the pipe will behave as rough when the height of the surface protrusions is equal to or greater than 0.413 mm.

Example 14.15. *Water flows in a 80 mm pipe at Reynold's number 80 000. The pipe is estimated to have an equivalent sand grain roughness of size 0.16 mm. Determine the head loss expected in 500 m length of the pipe. How much head loss would be expected if this pipe were smooth? Take kinematic viscosity of water as 10^{-6} m^2/s. The following explicit equation for friction factor may be used.*

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{k}{D} + \frac{21.25}{Re^{0.9}}\right)$$

in which k is the equivalent sand grain roughness, D is the pipe diameter and Re is the pipe Reynold's number.

Solution

If V is the velocity of flow, D is the pipe diameter and υ is kinematic viscosity of water, then Reynolds' number

$$\text{Re} = \frac{VD}{\upsilon}$$

$$\text{Re} = 80\,000;\ D = 80 \text{ mm} = 0.08 \text{ m; and}$$

$$\upsilon = 10^{-6} \text{ m}^2/\text{s}$$

Thus by substitution, we get

$$8000 = \frac{V\times0.08}{10^{-6}}$$

$$\therefore \quad V = 1 \text{ m/s}$$

From Darcy-Weisbach equation, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h_f = \frac{fLV^2}{2gD}$$

L = 500 m; and f is given by

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{k}{D} + \frac{21.25}{\text{Re}^{0.9}}\right)$$

k = 0.16 mm; D = 80 mm; and Re = 80 000

Thus by substitution, we get

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left[\frac{0.16}{80} + \frac{21.25}{(80000)^{0.9}}\right]$$

or $$\frac{1}{\sqrt{f}} = 6.2396$$

$\therefore$ $$f = 0.0257$$

Thus

$$h_f = \frac{0.0257 \times 500 \times (1)^2}{2 \times 9.81 \times 0.08} = 8.19 \text{ m}$$

If the pipe were smooth, $k = 0$ and hence the equation for friction factor becomes

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{21.25}{\text{Re}^{0.9}}\right)$$

Thus by substitution, we get

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left[\frac{21.25}{(80000)^{0.9}}\right]$$

or $$\frac{1}{\sqrt{f}} = 7.311$$

$\therefore$ $$f = 0.0187$$

Thus in this case, we have

$$h_f = \frac{0.0187 \times 500 \times (1)^2}{2 \times 9.81 \times 0.08} = 5.96 \text{ m}$$

Thus when the pipe is smooth the head loss is reduced by about 27%.

Example 14.16. *Starting from the velocity distribution in pipes given below*

$$\frac{v_{max} - v}{V_*} = 5.75 \log_{10}\left(\frac{R}{y}\right)$$

obtain the relationship for the ratio of maximum velocity to average velocity in terms of Darcy-Weisbach friction factor and show that this ratio increases with the friction factor.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

From the given velocity distribution, we have

$$v = v_{max} - 5.75 \quad V_* \log_{10}\left(\frac{R}{y}\right)$$

Changing from common (to the base 10) to normal (to the base e) logarithms, above equation becomes

$$v = v_{max} - \frac{5.75}{2.30} V_* \log_e\left(\frac{R}{y}\right)$$

or
$$v = v_{max} - 2.5 \quad V_* \log_e\left(\frac{R}{y}\right)$$

or
$$v = v_{max} + 2.5 \quad V_* \log_e\left(\frac{y}{R}\right)$$

Consider an elementary ring of thickness dr at a radial distance r from the pipe centre or at a distance y from the pipe boundary (see Fig. 14.8). Thus $y = (R–r)$ and the above equation becomes

$$v = v_{max} + 2.5 V_* \log_e\left(\frac{R-r}{R}\right)$$

The average or mean velocity V is given as follows

$$V = \frac{Q}{\pi R^2} = \frac{1}{\pi R^2}\int_A (v)dA$$

or
$$V = \frac{1}{\pi R^2}\int_0^R v(2\pi r)dr$$

or
$$V = \frac{1}{\pi R^2}\int_0^R 2\pi r\left[v_{max} + 2.5 V_* \log_e\left(\frac{R-r}{R}\right)\right]dr$$

or
$$V = \frac{2}{R^2}\int_0^R r\left[v_{max} + 2.5 V_* \log_e\left(\frac{R-r}{R}\right)\right]dr$$

or
$$V = \frac{2}{R^2}\left[\frac{v_{max} R^2}{2} + 2.5 V_* \int_0^R r \log_e\left(\frac{R-r}{R}\right)dr\right]$$

or
$$V = v_{max} + \frac{5V_*}{R^2}\int_0^R r \log_e\left(\frac{R-r}{R}\right)dr$$

or
$$V = v_{max} + \frac{5V_*}{R^2}\left[\left\{\frac{r^2}{2}\log_e\left(\frac{R-r}{R}\right)\right\}_0^R - \int_0^R \frac{-(r^2)}{2(R-r)}dr\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$V = v_{max} + \frac{5V_*}{R^2}\left[\frac{R^2}{2}\log_e(0) + \frac{1}{2}\int_0^R \frac{r^2}{(R-r)}dr\right]$$

or $$V = v_{max} + \frac{5V_*}{R^2}\left[\frac{R^2}{2}\log_e(0) + \frac{1}{2}\int_0^R \left\{-r - R + \frac{R^2}{(R-r)}\right\}dr\right]$$

or $$V = v_{max} + \frac{5V_*}{R^2}\left[\frac{R^2}{2}\log_e(0) + \frac{1}{2}\left(-\frac{R^2}{2} - R^2\right) - \frac{R^2}{2}\log_e(0)\right]$$

or $$V = v_{max} - 3.75\, V_*$$

Since $$V_* = V\sqrt{\frac{f}{8}}, \text{ we get}$$

$$V = v_{max} - 3.75\, V_*\sqrt{\frac{f}{8}}$$

or $$v_{max} = V\left(1 + 3.75\sqrt{\frac{f}{8}}\right)$$

or $$\frac{v_{max}}{V} = 1 + 3.75\sqrt{\frac{f}{8}} \quad \text{(It is same as Eq. 14.41(a))}$$

which is the required expression for the ratio (v_{max}/V). It shows that the ratio (v_{max}/V) increases as the friction factor f increases.

Example 14.17. *After 10 years of service, an asphalted cast iron water main 0.45 m in diameter is found to require 30% more power to deliver 240 litres per second for which it was originally designed. Determine the corresponding magnitude of the rate of roughness increase α. Take υ for water as 0.014 stokes.*

Solution

The rate of flow remaining constant, a 30% increase in power corresponds to a 30% increase in the value of friction factor f. Thus if f_{10} represents the value of f after 10 years then

$$f_{10} = 1.3\, f_0$$

where f_0 is the value of f for the new pipe at the time of its installation.

Discharge $$Q = 240 \times 10^{-3} = 0.24 \text{ m}^3/\text{s}$$

Mean velocity $$V = \frac{Q}{A} = \frac{0.24}{(\pi/4)(0.45)^2} = 1.51 \text{ m/s}$$

Reynolds number $$\text{Re} = \frac{VD}{\upsilon} = \frac{1.51 \times 0.45}{0.014 \times 10^{-4}} = 4.85 \times 10^5$$

From Table 14.1 the value of k for a new asphalted cast iron pipe is $k = k_0 = 0.12$ mm

Thus relative roughness k/D, at the time of installation of the pipe is

$$\left(\frac{k}{D}\right)_0 = \frac{0.12}{0.45 \times 10^3} = 0.000\,27$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From Fig. 14.12 for Re = 4.85×10^5 and $(k/D) = 0.00027$, the value of the friction factor $f_0 = 0.0165$

Thus $f_{10} = (1.3 \times 0.0165) = 0.0215$

Again from Fig. 14.12 for Re = 4.85×10^5 and $f_{10} = 0.0215$, the value of relative roughness $(k/D)_{10} = 0.00125$.

Now if k_{10} represents the value of k after 10 years then

$$k_{10} = (0.00125 \times 0.45)$$
$$= 0.5625 \times 10^{-3}\ \text{m} = 0.5625\ \text{mm}$$

and from Eq. 14.51 we have

$$\alpha = \frac{k_{10} - k_0}{t}$$

$$= \frac{0.5625 - 0.12}{10} = 0.04425\ \text{mm per year.}$$

SUMMARY OF MAIN POINTS

1. The flow in pipes is turbulent if Reynolds number is more than 4000.
2. The velocity distribution in turbulent flow is relatively uniform and the velocity profile of turbulent flow is much flatter than the corresponding laminar flow parabola for the same mean velocity.
3. Shear stress in turbulent flow is sum of shear stress due to viscosity and shear stress due to turbulence, i.e.,

$$\tau = \mu \frac{dv}{dy} + \eta \frac{dv}{dy}$$

where

μ = absolute viscosity; and

η = eddy viscocity;

4. Turbulent shear stress by Reynolds is given as

$$\tau = \overline{\rho v_y v_x}$$

where

v_x and v_y = the fluctuating components of velocity in the x and y directions respectively.

5. According to Prandtl the shear stress in turbulent flow is given by

$$\tau = \rho l^2 \left(\frac{dv}{dy}\right)^2$$

where

l = mixing length.

6. According to Prandtl, mixing length is that distance in the transverse direction which must be covered by a lump of fluid particles travelling with its original mean velocity in order to make the difference between its velocity and the velocity of the new layer equal to the mean transverse fluctuation in turbulent flow.
7. When a fluid flows through a pipe, close to the pipe wall a boundary layer is formed which may attain a maximum thickness equal to the radius of the pipe at a certain section of the pipe at which the flow is considered to have been established. For laminar flow in a pipe laminar boundary layer will be developed for the entire length of the pipe. According to Rouse the length of the pipe x from the entrance of the pipe upto the section where the flow is established is given as

$$\frac{x}{D} = 0.07\text{Re}$$

where

D = diameter of the pipe ; and

Re = Reynolds number $\left(= \frac{\rho VD}{\mu}\right)$

According to Rouse the distance required for the establishment of a fully developed turbulent flow is given as

$$\frac{x}{D} = 50$$

8. The boundary may be classified as

(i) Hydrodynamically smooth boundary

(ii) Hydrodynamically rough boundary

A boundary is known as hydrodynamically smooth boundary when (k/δ') is less than 0.25

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and it is known as hydrodynamically rough boundary when (k/δ') is greater than 6.0, where k is the average height of the irregularities projecting from the boundary surface, and δ' is thickness of the laminar sublayer.

The boundary is classified as boundary in transition when $0.25 < (k/\delta') < 6.0$

9. The velocity distribution for turbulent flow in hydrodynamically smooth pipes is given by

$$\frac{v}{V_*} = 5.75 \log_{10}\left(\frac{V_* y}{\nu}\right) + 5.5$$

where

V_* = shear or friction velocity $= \left(= \sqrt{(\tau_0/\rho)}\right)$;

and

ν = kinematic viscosity of the fluid.

10. The velocity distribution fort urbulent flow in hydrodynamically rough pipes is given by

$$\frac{v}{V_*} = 5.75 \log_{10}(y/k) + 8.5$$

where

k = average height of irregularities projecting from the boundary surface.

11. The mean velocity V for turbulent flow in hydrodynamically smooth pipes of radius R is given by

$$\frac{V}{V_*} = 5.75 \log_{10}\left(\frac{V_* R}{\nu}\right) + 1.75$$

12. The mean velocity V for turbulent flow in hydrodynamically rough pipes of radius R is given by

$$\frac{V}{V_*} = 5.75 \log_{10}\left(\frac{R}{k}\right) + 4.75$$

13. The velocity distribution identical for both hydrodynamically smooth and hydrodynamically rough pipes is given as

$$\frac{v - V}{V_*} = 5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75$$

For $y = R$, i.e., at pipe axis $v = v_{max}$, thus

$$\frac{v_{max} - V}{V_*} = 3.75$$

14. The loss of head due to friction for turbulent flow in pipes is given by Darcy-Weisbach equation as

$$h_f = \frac{fLV^2}{2gD}$$

15. For hydrodynamically smooth pipes the friction factor f is given by

(i) Blasius formula

$$f = \frac{0.316}{(\mathrm{Re})^{1/4}} \text{ .....for } 4 \times 10^3 < \mathrm{Re} < 10^5$$

(ii) Karman-Prandlt resistance equation

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\mathrm{Re}\sqrt{f}\right) - 0.8 \text{ .....for Re} > 10^5$$

(iii) Nikuradse's empirical equation

$$f = 0.0032 + \frac{0.021}{(\mathrm{Re})^{0.237}} \text{ .....for Re} > 10^5$$

16. For hydrodynamically rough pipes the friction factor f is given by

(i) Karman-Prandtl resistance equation

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

17. For commercial pipes the friction factor f is given by

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}\left(\frac{R}{k}\right) = 1.74 - 2.0 \log_{10}\left(1 + 18.7 \frac{R/k}{\mathrm{Re}\sqrt{f}}\right)$$

18. The pipe behave as hydrodynamically smooth pipe when the parameter $\left(\frac{\mathrm{Re}\sqrt{f}}{R/k}\right)$ is less than 17 and the pipe behaves as hydrodynamically rough when the parameter $\left(\frac{\mathrm{Re}\sqrt{f}}{R/k}\right)$ is more than 400.

For $17 < \left(\frac{\mathrm{Re}\sqrt{f}}{R/k}\right) < 400$ the pipe boundary behaves as in transition. This is another criterion for the classification of the boundary surface of pipes.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

PROBLEMS

14.1 Define 'shear velocity' or 'friction velocity' for turbulent flow in circular pipes.

14.2 Show that for laminar flow in circular pipes the friction factor is inversely proportional to the Reynolds number.

14.3 What do you understand by hydrodynamically smooth and rough pipes?

14.4 How is head loss in commercial pipes determined?

14.5 Describe Nikuradse's experiments on the resistance of artificially roughened pipes. Discuss the characteristic features of the results obtained.

14.6 With the help of Prandtl's pitot tube installed at mid point and quarter point of a 300 mm diameter pipe, mean point velocities at these points were observed as 1.5 m/s and 1.35 m/s respectively. Assuming the flow to be rough turbulent, determine the discharge, friction factor and relative roughness.

[*Ans.* 83 *l*/s; 0.0436; (*k*/*D*) = (1/66.93)]

14.7 A smooth pipeline 0.1 m in diameter and 1000 m long carries water at the rate of 7.5 litres per second. If the kinematic viscosity of water is 0.02 stokes, calculate the head lost, wall shearing stress, centreline velocity, shear stress and velocity at 40 mm from the centreline, and the thickness of the laminar sublayer.

[*Ans.* 9.95 m; 2.44 N/m^2 {0.25 kg (f)/m^2}; 1.15 m/s; 1.95 N/m^2 {0.2 kg (f)/m^2}; 0.95 m/s; 0.47 mm]

14.8 Rough turbulent flow occurs in a pipe 0.2 m diameter conveying water. If at y = 20 mm, v = 2.0 m/s and $(\partial v/\partial y)$ = 10.5 s^{-1}, determine k_0, t_0, f and V. Assume ρ for water as 1000 kg/m^3 (102 msl/m^3).

[*Ans.* 0.12 mm; 8.82 N/m^2 {0.90 kg (f)/m^2}; 0.017; 2.025 m/s]

14.9 A pipe of diameter 0.3 m is to convey water at 40°C at the rate of 200 litres per second. If the power required to maintain the flow in 100 m length is 61.8 kW (84 h.p.), calculate the value of k, V_{max}, t_0 and V^*. Take v for water at 40°C as 0.0075 stokes.

[*Ans.* 0.57 mm; 3.4 m/s; 23.18 N/m^2 {2.363 kg (f)/m^2} ; 0.152 m/s]

14.10 If a pipe 3 m in diameter is to carry a flow of 7.5 m^3/s of water with a minimum expenditure of energy, determine the permissible height of the roughness projection on the pipe wall. υ for water is 0.015 × 10^{-4} m^2 s. [*Ans.* 0.12 mm]

14.11 A smooth pipe of diameter 0.25 m and length 3000 m carries oil at the rate of 50 litres per second. The oil has specific gravity of 0.85 and viscosity of 0.044 N.s/m^2. If an increase in temperature causes the viscosity to decrease to 0.025 N.s/m^2 and if the power input remains the same, determine the change in the rate of flow. [*Ans.* Increase at the rate of 2.67 *l*/s]

14.12 The oil is pumped through a pipe of diameter 0.4 m at the rate of 500 litres per second to a distance of 10 km. Determine the power input required for the pump if the pump efficiency is 80 percent. Take ρ = 804.42 kg/m^3 (82 msl/m^3) and μ= 3.924× 10^{-2} N.s/m^2 [4 × 10^{-3} kg (f)$^{-s}$/m^2] for the oil and k for the pipe as 0.12 mm.

[*Ans.* 2.34 ×10^3 kW (3.18 × 10^3 h.p.)]

14.13 A 300 mm diameter pipeline carries water at 20°C with a mean velocity of 7.5 m/s. The pipeline is new with no surface irregularities at the beginning, but it was found that the surface irregularities grow at the rate of 0.075 mm per year. Find the number of years after which the surface irregularities will affect the flow. Take υ = 0.01 stokes. [*Ans.* 2.67 years]

14.14 After 15 years of service a steel water main 0.6 m in diameter is found to require 40% more power to deliver the 300 litres per second, for which it was originally designed. Determine the corresponding magnitude of the rate of roughness increase α. Take υ = 0.015 stokes.

[*Ans.* 0.033 mm per year]

14.15 A smooth wrought iron pipe 0.2 m in diameter carries crude oil at a velocity of 2.5 m/s. What is the loss of head per 100 metres? What is the power required to maintain the flow? Assume kinematic viscosity for oil 0.40 stokes and sp. gr. 0.90. [*Ans.* 4.78 m; 3.315 kW (4.505 h.p.)]

14.16 Water with kinematic viscosity 0.01 stokes flows in a 200 mm dia. pipe with absolute roughness 0.04 mm with a mean velocity of 1 m/s and the observed head loss is 0.6 m in 100 m length of pipe. If a glycerine-water mixture of kinematic viscosity 0.02 stokes flows in a 100 mm dia pipe of roughness 0.02 mm with a velocity of 4 m/s, how much head loss per 100 m of pipe should be expected? [*Ans.* 19.2 m]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Flow in Open Channels

Chapter 15

15.1 INTRODUCTION

A channel may be defined as a passage through which water flows under atmospheric pressure. As such in channels the flow of water takes place with a free surface which is subjected to atmospheric pressure. Figure 15.1 shows a comparison between the flow through a pipe and a channel. In the case of flow through a pipe there is no free surface as in the case of flow through a channel. This is so because the water flowing in a pipe is under pressure, which at any section of a pipe is indicated by the level of water column in a piezometric tube installed on the pipe at the section. The water levels in

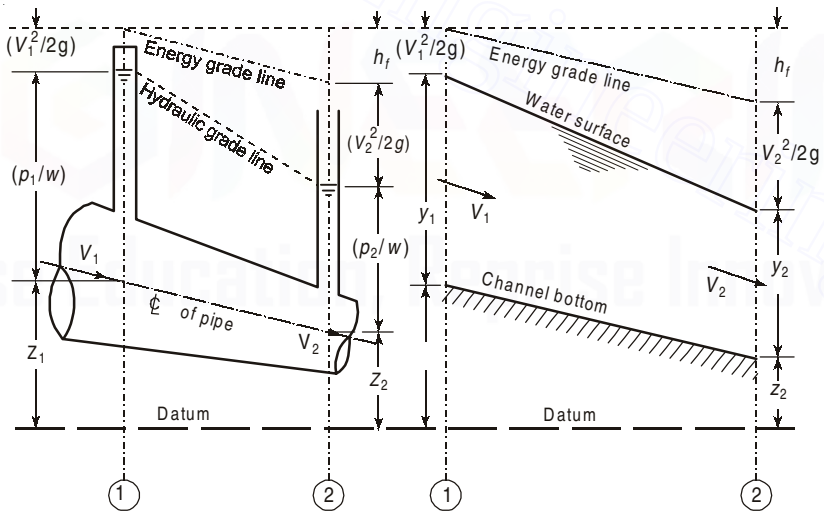


Figure 15.1 *Comparison between pipe flow and channel flow*

the piezometric tubes installed at different sections of a pipe indicate the hydraulic grade line. On the other hand in the case of channel flow the water surface itself is the hydraulic grade line. However, the total energy lines in both the cases lie at a distance of $(V^2/2g)$ above the hydraulic grade line at every section, where V is the mean velocity of flow at the section.

Since the pressure on the top surface of water in a channel is constant, no pressure difference can be built up between any two sections along the channel as in the case of pipes. Moreover, as the water

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

flows in a channel, it will always be subjected to some resistance. As such in order to overcome the resistance and to cause the flow of water in a channel, it is constructed with its bottom slopping towards the direction of flow, so that a component of the weight of the flowing water in the direction of flow is developed which causes the flow of water in the channel.

The channels may be classified according to different considerations as described below. On the basis of the cross-sectional form of the channel it may be classified as *natural channel* or *artificial channel*. A natural channel is the one which has irregular sections of varying shapes, which is developed in a natural way. The examples of natural channels are rivers, streams etc. On the other hand an artificial channel is the one which is built artificially for carrying water for various purposes. Obviously artificial channels have their cross-sections with regular geometrical shapes, which usually remain same throughout the length of the channel. The artificial channels may be further classified according to the shape of the cross-section as, *rectangular channel, trapezoidal channel*, *triangular channel*, *parabolic channel,* and *circular channel*.

The channels may also be classified as *open channels* and *closed channels.* The channels without any cover at the top are known as open channels. On the other hand the channels having cover at the top are known as closed channels. However, the closed channels will always be running partly full of water, in order that the flow may be a channel flow with atmospheric pressure prevailing over its entire top surface. Some of the common examples of closed channels are closed conduits or pipes flowing partly full of water, underground drains, tunnels etc., not running full of water.

A channel having the same shape of various sections along its length and laid on a constant bottom slope is known as *prismatic channel*, otherwise the channel is *non-prismatic*. In this chapter only prismatic channels have been dealt with.

15.2 TYPES OF FLOW IN CHANNLES

The flow in channels can be classified into following types depending upon the change in the depth of flow with respect to space and time.

(a) Steady Flow and Unsteady Flow. Flow in a channel is said to be steady if the flow characteristics at any point do not change with time that is $(\partial V/\partial t)= 0$, $(\partial y/\partial t) = 0$ etc. However, in the case of prismatic channels the conditions of steady flow may be obtained if only the depth of flow does not change with time, that is $(\partial y/\partial t) = 0$. On the other hand if any of the flow characteristics changes with time the flow is unsteady. Most of the open channel problems involve the study of flow under steady conditions and therefore in this chapter only steady flow has been considered.

(b) Uniform and Non-uniform (or Varied) Flow. Flow in a channel is said to be uniform if the depth, slope, cross-section and velocity remain constant over a given length of the channel. Obviously, a uniform flow can occur only in a prismatic channel in which the flow will be uniform if only the depth of flow y is same at every section of the channel, that is $(\partial y/\partial s) = 0$.

Flow in channels is termed as non-uniform or varied if the depth of flow y, changes from section to section, along the length of the channel, that is $(\partial y/\partial s)$ is not equal to zero. Varied flow may be further classified as rapidly varied flow (R.V.F.) and gradually varied flow (G.V.F.). If the depth of flow changes abruptly over a comparatively short distance, the flow is characterized as a rapidly varied flow. Typical examples of rapidly varied flow are hydraulic jump and hydraulic drop. In a gradually varied flow the change in the depth of flow takes place gradually in a long reach of the channel.

(c) Laminar Flow and Turbulent Flow. Just as in pipes, the flow in channels may also be characterized as laminar, turbulent or in a transitional state, depending on the relative effect of viscous

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and inertia forces and alike pipes, Reynolds number Re is a measure of this effect in channel flow also. However, the Reynolds number for flow in channel is commonly defined as $\text{Re} = \left(\frac{\rho VR}{\mu}\right)$ where V is the mean velocity of flow, R is the hydraulic radius (or hydraulic mean depth) of the channel section and ρ and μ are respectively the mass density and absolute viscosity of water. On the basis of the experimental data it has been found that upto Re equal to 500 to 600 the flow in channels may be considered to be laminar and for Re greater than 2000 the flow in channels is turbulent. Thus for Re in between 500 to 2000 the flow in channels may be considered to be in transitional state. Since most of the channel flows are turbulent, in this chapter only the turbulent flow has been dealt with.

(d) Subcritical Flow, Critical Flow and Supercritical Flow. As stated earlier gravity is a predominant force in the case of channel flow. As such depending on the relative effect of gravity and inertia forces the channel flow may be designated as subcritical, critical or supercritical. The ratio of the inertia and the gravity forces is another dimensionless parameter called Froude number Fr which is defined as $Fr = \frac{V}{\sqrt{gD}}$; where V is the mean velocity of flow, g is acceleration due to gravity and D is hydraulic depth of channel section (as defined in Section 15.3). As indicated later, when $Fr = 1$, that is $V = \sqrt{gD}$, the channel flow is said to be in a critical state. If $Fr < 1$, or $V < \sqrt{gD}$, the flow is described as subcritical or tranquil or streaming. If $Fr > 1$, or $V > \sqrt{gD}$, the flow is said to be supercritical or rapid or shooting or torrential.

15.3 GEOMETRICAL PROPERTIES OF CHANNEL SECTION

The geometrical properties of a channel section can be defined entirely by the geometry of the section and the depth of flow. Some of the geometrical properties of basic importance are defined below.

The depth of flow y, is the vertical distance of the lowest point of a channel section from the free surface.

The top width T is the width of the channel section at the free surface.

The wetted area A (or water area or area of flow section) is the cross-sectional area of the flow normal to the direction of flow.

The wetted perimeter P is the length of the channel boundary in contact with the flowing water at any section.

The hydraulic radius R (or hydraulic mean depth) is the ratio of the wetted area to its wetted perimeter,

or $$R = \left(\frac{A}{P}\right)$$

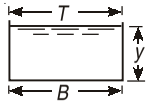
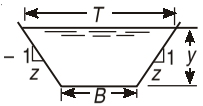
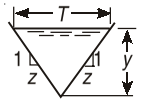
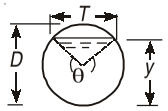
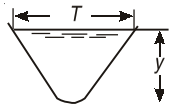
The hydraulic depth D is the ratio of the wetted area to the top width T,

or $$D = \left(\frac{A}{T}\right)$$

The section factor for critical flow computation Z is the product of the wetted area and the square root of the hydraulic depth,

Downloaded From : www.EasyEngineering.net

TABLE 15.1 *Geometric Elements of Channel Section*

	Section	Area A	Wetted perimeter P	Hydraulic radius R	Top width T	Hydraulic depth D	Section factor Z
Rectangle		By	$B + 2y$	$\frac{By}{B+2y}$	B	y	$B\,(y)^{1.5}$
Trapezoid		$(B + zy)\,y$	$B+2y\sqrt{z^2+1}$	$\frac{(B+zy)y}{B+2\sqrt{z^2+1}}$	$B + 2zy$	$\frac{(B+zy)y}{B+2zy}$	$\frac{[(B+zy)y]^{1.5}}{\sqrt{B+2zy}}$
Triangle		zy^2	$2y\sqrt{z^2+1}$	$\frac{zy}{2\sqrt{z^2+1}}$	$2zy$	$\frac{1}{2}y$	$\frac{\sqrt{2}}{2}z(y)^{2.5}$
Circle		$\frac{1}{8}(\theta-\sin\theta)D^2$	$\frac{1}{2}\theta D$	$\frac{1}{4}\left(1-\frac{\sin\theta}{\theta}\right)D$	$D\left(\sin\frac{\theta}{2}\right)$, or $2\sqrt{y(D-y)}$	$\frac{1}{8}\left(\frac{\theta-\sin\theta}{\sin\frac{\theta}{2}}\right)$ D	$\frac{\sqrt{2}}{32}\frac{(\theta-\sin)^{1.5}}{\left(\sin\frac{\theta}{2}\right)^{0.5}}D^{2.5}$
Parabola		$\frac{2}{3}Ty$	$T+\frac{8}{3}\frac{y^2}{T}$	$\frac{2T^2y}{3T^2+8y^2}$	$\frac{3A}{2y}$	$\frac{2}{3}y$	$\frac{2}{9}\sqrt{6}Ty^{1.5}$

Downloaded From : www.EasyEngineering.net
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$Z = A\sqrt{D} = A\sqrt{\left(\frac{A}{T}\right)} = \left(\frac{A^3}{T}\right)^{1/2}$$

The section factor for uniform flow computation $AR^{2/3}$ is the product of the wetted area and the hydraulic radius to two-thirds power.

Table 15.1 provides a list of the expressions for the geometrical properties of five commonly used channel sections.

15.4 VELOCITY DISTRIBUTION IN A CHANNEL SECTION

The velocity of flow at any channel section is not uniformly distributed. The non-uniform distribution of velocity in an open channel is due to the presence of a free surface and the frictional resistance along the channel boundary. The velocity distribution in a channel is measured either with the help of a pitot tube or a current meter, described later. The general patterns for velocity distribution as represented by lines of equal velocity, in some of the common channel sections are illustrated in Fig. 15.2. A velocity distribution curve along a vertical line of the channel section is also shown in Fig. 15.2. In a straight reach of a channel maximum velocity usually occurs below the free surface at a distance of 0.05 to 0.15 of the depth of flow. The velocity distribution in a channel section depends on the various factors such as the shape of the section, the roughness of the channel and the presence of bends in the channel alignment.

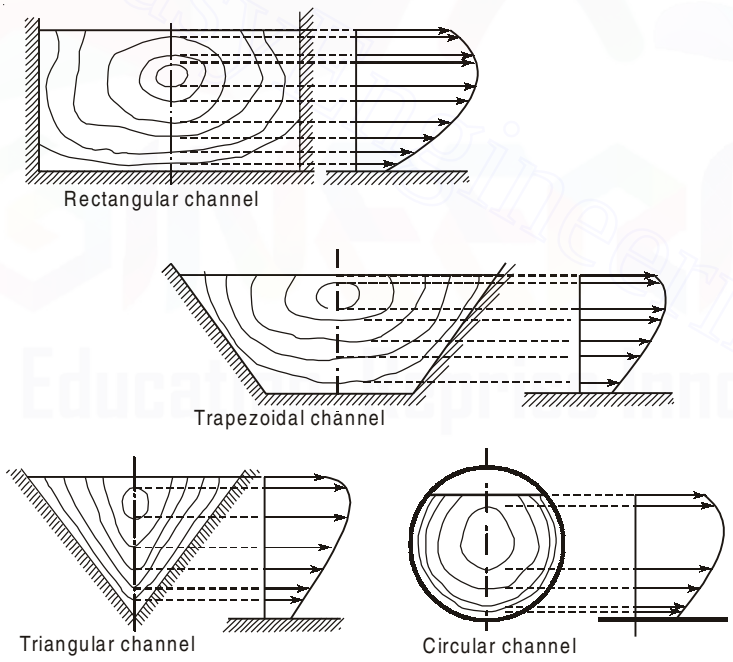


Figure 15.2 *Typical patterns of velocity distribution in various channel sections*

The mean velocity of flow in a channel section can be computed from the vertical velocity distribution curve obtained by actual measurements. It is observed that the velocity at 0.6 depth from the free surface is very close to the mean velocity of flow in the vertical section. A still better approximation for the mean velocity of flow is obtained by taking the average of the velocities measured at 0.2 depth and 0.8 depth from the free surface.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

On account of non-uniform distribution of velocities over a channel section, correction factors will have to be applied while computing the kinetic energy and the momentum. The kinetic energy correction factor α (Greek 'alpha') is also known as Coriolis coefficient in honour of G. Coriolis who first proposed it. The momentum correction factor β (Greek 'beeta') is also called Boussinesq coefficient after J. Boussinesq who first proposed it. As mentioned in previous chapters the values of these correction factors can be obtained if the actual velocity profile at a section is known. Experimental data indicate that for turbulent flow in fairly straight prismatic channels, the value of α varies from 1.03 to 1.36 and that of β from about 1.01 to 1.12. However, for the sake of simplicity, both α and β are assumed to be unity in the present analysis.

15.5 UNIFORM FLOW IN CHANNLES

When water flows in an open channel resistance is offered to it, which results in causing a loss of energy. The resistance encountered by the flowing water is generally counteracted by the components of gravity forces acting on the body of the water in the direction of motion [Fig. 15.3 (b)]. A uniform flow

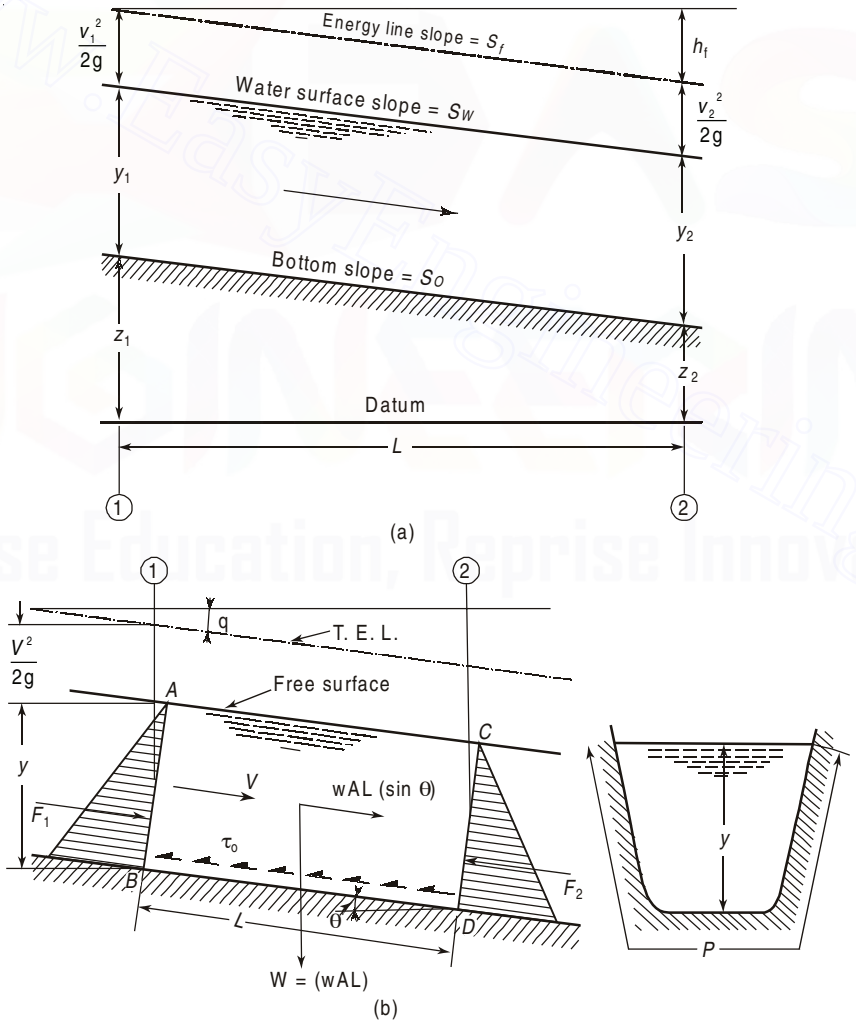


Figure 15.3 (*a*) *Uniform flow in open channel* (*b*) *Forces on a segment of channel having uniform flow*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

will be developed if the resistance is balanced by the gravity forces. The magnitude of the resistance, when other physical factors of the channel are kept unchanged, depends on the velocity of flow. When water enters the channel, the velocity and hence the resistance are smaller than the gravity forces, which results in an accelerating flow in the upstream reach of the channel. The velocity and the resistance increase gradually until a balance between the resistance and gravity forces is reached. From this point onwards the flow becomes uniform. Several *uniform-flow formulae* have been developed which correlate the mean velocity of uniform flow in open channels with the hydraulic radius, energy line slope and a factor of flow resistance. The most widely used uniform-flow formulae are the Chezy and Manning formulae which are described in the following paragraphs.

The main features of uniform flow in a channel can be summarized as follows:

(1) The depth of flow, wetted area, velocity of flow and discharge are constant at every section along the channel reach.

(2) The total energy line, water surface and the channel bottom are all parallel, that is, their slopes are all equal, or $S_f = S_w = S_0 = S$, where S_f is energy line slope, S_w is water surface slope and S_0 is channel bottom slope.

Figure 15.3 (a) shows a short reach of prismatic channel having uniform flow. Applying Bernoulli's equation between sections 1 and 2 at a distance L apart, we obtain

$$y_1 + \frac{V_1^2}{2g} + Z_1 = y_2 + \frac{V_2^2}{2g} + Z_2 + h_f$$

where h_f is the loss of energy between the two sections. Since the flow is uniform, $y_1 = y_2$; $V_1 = V_2$. Thus from the above expression

$$Z_1 - Z_2 = h_f$$

or
$$\frac{Z_1 - Z_2}{L} = S_0 = \frac{h_f}{L} = S_f = S_w.$$

The depth of a uniform flow is called the *normal depth* and it is generally represented by y_n.

The fundamental equation for uniform flow in channels may be derived by applying Newton's second law of motion. In uniform flow since the velocity of flow does not change along the length of the channel, there is no acceleration. Hence the sum of the components of all the external forces in the direction of flow must be equal to zero. Consider a short reach of channel of length L in which uniform flow occurs, as shown in Fig. 15.3 (b). The forces acting on the free-body of water $ABCD$ in the direction of flow are as follows:

(1) The forces of hydrostatic water pressure F_1 and F_2 acting on the two ends of the freebody. As the depths of water at these two sections are the same, the forces F_1 and F_2 are equal and hence they cancel each other.

(2) The component of weight of the water in the direction of flow, which is ($wAL \sin \theta$), where w is specific weight of water, A is the wetted cross-sectional area of channel and θ is the angle of inclination of the channel bottom with the horizontal.

(3) The resistance to the flow is exerted by the wetted surface of the channel. If P is the wetted perimeter of the channel and τ_0 is the average shear stress at the channel boundary, the total resistance to flow will be ($PL\tau_0$).

According to Newton's second law of motion, in the direction of flow

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$wAL \sin\theta - PL\tau_0 = 0$$

or
$$\tau_0 = w\frac{A}{P}\sin\theta = wRS_0 \qquad ...(15.1)$$

where R is the hydraulic radius and S_0 ($= \sin\theta = S$) is the slope of the channel bottom which for small values of θ may be considered equal to $\tan\theta$, since $\sin\theta \simeq \tan\theta$.

In Chapter 13 the shear stress τ_0 has been related to mass density ρ and average velocity V by the equation $\tau_0 = \frac{f}{8}\rho V^2$. Thus equating the two expressions for τ_0, we get

$$wRS = \frac{f}{8}\rho V^2$$

or
$$V = \sqrt{\frac{8w}{\rho f}}\sqrt{RS} = \sqrt{\frac{8g}{f}}\sqrt{RS}$$

Since $(w/\rho) = g$

The above expression is generally written as

$$V = C\sqrt{RS} \qquad ...(15.2)$$

in which
$$C = \sqrt{\left(\frac{8g}{f}\right)}$$

Equation 15.2 is known as Chezy's formula named after the French Engineer Antoine Chezy who developed this formula in 1775. The term C is known as Chezy's coefficient. It may, however, be observed that the Chezy's coefficient C varies inversely as the square root of the Darcy-Weisbach resistance coefficient f. Further it may be noted that Chezy's coefficient is not a dimensionless coefficient but it has the dimensions $[L^{1/2}\ T^{-1}]$. Although Chezy's equation is quite simple the selection of a correct value of C is rather difficult. As such many empirical formulae have been developed to determine the value of Chezy's C. Some of the important formulae developed for this purpose are as noted below.

(a) The Ganguillet-Kutter Formula. On the basis of a series of flow measurements taken in channels of various types, two Swiss engineers Ganguillet and Kutter proposed an empirical formula in 1869, expressing the value of Chezy's C in terms of the slope S, hydraulic radius R and a roughness coefficient n. In SI or metric system the formula is

$$C = \frac{23 + \frac{0.00155}{S} + \frac{1}{n}}{1 + \left(23 + \frac{0.00155}{S}\right)\frac{n}{\sqrt{R}}} \qquad ...(15.3)$$

The roughness coefficient n is known as Kutter's n. The value of n varies widely depending upon the channel surface and its condition. Some typical values of n for different types of surfaces commonly encountered in practice are given in Table 15.3.

(b) The Bazin Formula. In 1897, the French hydraulician H. Bazin proposed another empirical formula for Chezy's C. According to this formula Chezy's C is expressed in terms of hydraulic radius R and another roughness, factor m, known as Bazin's roughness coefficient. In metric system Bazin's formula may be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$C = \frac{157.6}{1.81 + \frac{m}{\sqrt{R}}} \qquad \ldots(15.4)$$

Some of the values of m proposed by Bazin are given in Table 15.2.

TABLE 15.2 *Value of Bazin's m for Various Surface Materials*

Type of Channel Boundary Surfacem	
Very smooth cement, or planed wood	0.11
Concrete or brick or unplaned wood	0.21
Ashlar, rubble masonry or poor brick work	0.83
Earth channels in very good condition	1.54
Earth channels in ordinary condition	2.36
Earth channels in rough condition	3.17

(c) The Manning Formula. In 1889 an Irish engineer Robert Manning presented a formula according to which the mean velocity V of uniform flow in a channel is expressed in terms of a coefficient of roughness n, called Manning's n, hydraulic radius R and channel bottom slope S. The Manning's formula expressed in SI or metric units is written as

$$V = \frac{1}{n} R^{2/3} S^{1/2} \qquad \ldots(15.5)$$

Owing to its simplicity of form and to the satisfactory result it yields in practice, the Manning formula has now becomes the most widely used of all the empirical formula for the computation of uniform flow in open channels.

If the Manning formula is compared with the Chezy formula it can be seen that

$$C = \frac{1}{n} R^{1/6} \qquad \ldots(15.6)$$

The Eq. 15.6 provides an important relationship between Chezy's C and Manning's n.

The values of Manning's n are found to be approximately equal to those of Kutter's n for the normal ranges of slope and hydraulic radius. Some of the typical values which hold good for both Kutter's n and Manning's n are given in Table 15.3.

TABLE 15.3 *Average Values of Manning's Roughness Coefficient n*

Type of channel boundary surface	*Value of n*
Very smooth surface such as glass, plastic or brass	0.010
Very smooth concrete and planed timber	0.011
Smooth concrete	0.012
Ordinary concrete lining	0.013
Glazed brick work	0.013
Good wood	0.014
Vitrified clay	0.014
Brick surface lined with cement mortar	0.015
Cement concrete finish	0.015
Unfinished cement surface	0.017

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 15.3 Contd.

Type of channel boundary surface	*Value of n*
Earth channel in best condition	0.017
Neatly excavated rock	0.020
Straight unlined earth canals in good condition	0.020
Rubble masonry	0.020
Corrugated metal surface	0.020
Rivers and earth canals in fair condition	0.025
Earth channel with gravel bottom	0.025
Earth channel with dense weed	0.035
Mountain stream with rock beds and rivers with variable sections and some vegetation along banks	0.045

In addition to the above mentioned uniform-flow formulae, a few other formulae for uniform flow in channels have been derived on the basis of theoretical velocity distribution across the channel section. For turbulent flow in channels the velocity distribution can be shown to be approximately logarithmic. Thus using the Prandtl-von Karman universal velocity-distribution law viz.,

$$[(v/V_*) = 2.5 \log_e y + C],$$

Keulegan has derived the following equations for the mean velocity of uniform turbulent flow in channels.

For smooth channels

$$(V/V_*) = 3.25 + 5.75 \log_{10}\left(\frac{RV_*}{\upsilon}\right) \quad \ldots(15.7)$$

For rough channels

$$(V/V_*) = 6.25 + 5.75 \log_{10}(R/k) \quad \ldots(15.8)$$

where V is mean velocity of flow, V_* is shear velocity, R is hydraulic radius, υ is kinematic viscosity and k is average height of roughness protrusions.

From Eq. 15.1, $V_* = \sqrt{(\tau_0/\rho)} = \sqrt{gRS}$;

and from Chezy's formula $V = C\sqrt{RS}$

Thus $$(V/V_*) = \left(C/\sqrt{g}\right) \quad \ldots(15.9)$$

Substituting Eq. 15.9 and the Reynolds number Re = (VR/υ) in Eqs 15.7 and 15.8, the following expressions for Chezy's C are obtained.

For smooth channels

$$C = 18 \log_{10}\left(\frac{11.5Re}{C}\right) \quad \ldots (15.10)$$

For rough channels

$$C = 18 \log_{10}\left(\frac{12.2R}{k}\right) \quad \ldots (15.11)$$

However, some of the later investigators have indicated that the values of the additive constants in Keulegan's Eqs 15.7 and 15.8 vary with Froude number.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For rough channels, by eliminating Chezy's C from Eqs 15.6 and 15.11, Manning's n may be expressed as follows.

Thus $$n = \phi(R/k)k^{1/6} \quad \ldots (15.12)$$

where $$\phi(R/k) = \frac{(R/k)^{1/6}}{18\log(12.2R/k)} \quad \ldots(15.13)$$

For a wide range of (R/k), the variation in the values of $\phi(R/k)$ is small and hence as an approximation, $\phi(R/k)$ may be assumed to be constant. On the basis of actual obervations, Strickler has given a formula for n, which when compared with Eq. 15.12, gives an average value of $\phi(R/k) = 0.0476$ or $(1/21)$. Thus according to Strickler formula

$$n = \frac{k^{1/6}}{21} \quad \ldots(15.14)$$

in which k is in metres. It may however be stated that the roughness height used by Strickler is the median sieve size of the material (or d_{50} which represents such a size that fifty per cent of the material is finer than d_{50}). One of the later investigators has however shown that the value of the constant in the Strickler formula should be 24 instead of 21, in which case Eq. 15.14 becomes

$$n = \frac{k^{1/6}}{24} \quad \ldots(15.14\ a)$$

Equation 15.14 leads to the conclusion that Manning's n varies with one-sixth power of the roughness height and hence a slight error in the value of roughness height does not have much effect on the value of Manning's n computed from Eq. 15.14. However, it is extremely difficult to predict the correct value of n or C, since each of these parameters is highly variable and depends on several factors.

15.6 MOST ECONOMICAL OR MOST EFFICIENT SECTION OF CHANNEL

A channels section is considered to be the most economical or most efficient when it can pass a maximum discharge for given cross-sectional area, resistance coefficient and bottom slope. From the equation of continuity it is evident that for area of cross-section being constant, discharge is maximum when velocity of flow is maximum. From Chezy formula and Manning formula it can be seen that for a given value of slope and surface roughness the velocity of flow will be maximum if hydraulic radius R is maximum. Further the area being constant hydraulic radius is maximum if the wetted perimeter is minimum. As such this condition is utilized to determine the dimensions of economical sections of different forms of channels as discussed below.

(a) Most Economical or Most Efficient Rectangular Channel Section. For a rectangular channel section of bottom width B and depth of flow y (Figure15.4), following expressions for wetted area A and wetted perimeter P can be written

$$A = By \quad \ldots(i)$$

$$P = B + 2y \quad \ldots(ii)$$

From Eq. (i)

$$B = \frac{A}{y}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the value of B in Eq. (ii) it becomes

$$P = \frac{A}{y} + 2y \qquad ...(iii)$$

Assuming the area A to be constant, Eq. (iii) can be differentiated with respect to y and equated to zero for obtaining the condition for minimum P.

Thus $$\frac{dP}{dy} = -\frac{A}{y^2} + 2 = 0$$

or $$A = 2y^2 = By$$

$\therefore$ $$B = 2y$$

or $$y = \frac{B}{2} \qquad ...(15.15)$$

Also hydraulic radius $$R = \frac{A}{P} = \frac{By}{B + 2y}$$

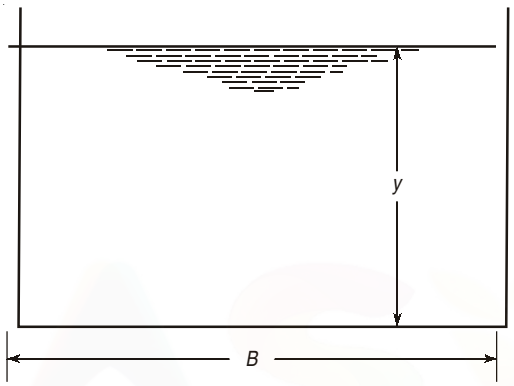


Figure 15.4 *Rectangular channel section*

Substituting the value of

$$B = 2y$$

$$R = \frac{2y^2}{2y + 2y} = \frac{y}{2} \qquad ... (15.16)$$

Thus it can be seen that a rectangular channel section will be most economical or most efficient when either the depth of flow is equal to half the bottom width, or hydraulic radius is equal to half the depth of flow.

(b) Most Economical or Most Efficient Trapezoidal Channel Section. For a trapezoidal channel section of bottom width B, depth of flow y and side slope z horizontal to 1 vertical (Fig. 15.5), following expression for wetted area A and wetted perimeter P can be written.

$$A = (B + zy)\, y \qquad ... (i)$$

$$P = B + 2y\sqrt{1 + z^2} \qquad ... (ii)$$

From Eq. (i)

$$B = \frac{A}{y} - zy$$

Substituting the value of B in Eq. (ii) it becomes

$$P = \frac{A}{y} - zy + 2y\sqrt{1 + z^2} \qquad ... (iii)$$

Assuming area A and side slope z to be constant, Eq. (iii) can be differentiated with respect to y and equated to zero for obtaining the condition for minimum P.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $$\frac{dP}{dy} = -\frac{A}{y^2} - z + 2\sqrt{1+z^2} = 0$$

or $$\frac{A}{y^2} + z = 2\sqrt{1+z^2}$$

Substituting the value of A from Eq. (i) the above equation becomes

$$\frac{(B+zy)y}{y^2} + z = 2\sqrt{1+z^2}$$

or $$\frac{B+2zy}{2} = y\sqrt{1+z^2} \quad \ldots (15.17)$$

Equation 15.17 is the required condition for a trapezoidal channel section to be most economical or most efficient. It shows that for a trapezoidal channel section to be most economical or most efficient half the top width must be equal to one of the sloping sides of the channel.

The hydraulic radius R can be expressed as

$$R = \frac{A}{P}$$

$$= \frac{(B+zy)y}{B+2y\sqrt{1+z^2}}$$

Substituting the values of B from Eq. 15.17 in the above expression, we get

$$R = \frac{(2y\sqrt{1+z^2} - 2zy + zy)y}{2y\sqrt{1+z^2} - 2zy + 2y\sqrt{1+z^2}}$$

or $$R = \frac{y}{2} \quad \ldots (15.18)$$

Thus it can be seen that for a trapezoidal channel section to be most economical or most efficient the hydraulic radius must be equal to half the depth of flow.

Further let O be the centre of top width of the trapezoidal channel section. Drop perpendicular OA from O to the slopping side of the channel, (Fig. 15.5.) If θ is the angle made by the slopping side with the horizontal, then from the right-angled triangle OAC, $OA = OC \sin\theta$.

But $$OC = \left(\frac{B+2zy}{2}\right)$$

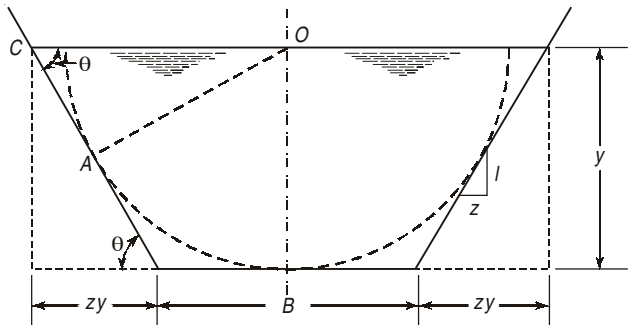


Figure 15.5 *Trapezoidal channel section*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$\sin\theta = \frac{y}{y\sqrt{1+z^2}}$$

$$\therefore \quad OA = \frac{B+2zy}{2} \times \frac{y}{y\sqrt{1+z^2}}$$

$$= \frac{B+2zy}{2\sqrt{1+z^2}}$$

Now substituting for $(B + 2zy)$ from Eq. 15.17, we obtain for most economical or most efficient trapezoidal channel section

$$OA = y \qquad \ldots (15.19)$$

That is, if a semi-circle is drawn with O as centre and radius equal to the depth of flow y, the three sides of a most economical or most efficient trapezoidal channel section, viz., the bottom and the two slopping sides will be tangential to the semi-circle.

Thus Eqs. 15.17, 15.18 and 15.19 represent the three conditions, either of which must be satisfied for a trapezodial channel section to be most economical or most efficient.

The above derived conditions are based on the assumption that side slope z is constant. As such these conditions will not provide a most economical or most efficient trapezoidal channel section if either the bottom width or the depth of flow has to be limited due to physical considerations. However, it is possible to derive the conditions which should be satisfied for a trapezoidal channel section to be most economical or most efficient when either the bottom width or the depth of flow has to be kept constant. By adopting the same procedure as indicated above, the following two conditions can be obtained which may be used for the design of a most economical or most efficient trapezoidal channel section.

(1) For the bottom width to be kept constant

$$\frac{B}{y} = \frac{1-z^2}{z} \qquad \ldots (15.20)$$

or $$A = \frac{y^2}{z} \qquad \ldots (15.21)$$

(2) For the depth of flow to be kept constant

$$z = \frac{1}{\sqrt{3}}; \text{ or } \theta = 60^\circ \qquad \ldots (15.22)$$

If Eq. 15.22 is introduced in Eqs. 15.17 and 15.20, then in both the cases, we get

$$\frac{B}{y} = \frac{2}{\sqrt{3}}$$

It may, therefore, be stated that the most economical or most efficient trapezoidal channel section, which would satisfy all the above noted three conditions, may be obtained by adopting side slope $z = \left(1/\sqrt{3}\right)$ in the design of the channel section. This fact is indicated in Example 15.2.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(c) Most Economical or Most Efficient Triangular Channel Section. For a triangular channel section, if θ is the angle of inclination of each of the slopping sides with the vertical and y is the depth of flow (Fig.15.6), the following expression for the wetted area A and the wetted perimeter P can be written

$$A = y^2 \tan\theta$$

or
$$y = \sqrt{A/\tan\theta} \quad \text{... (i)}$$

$$P = (2y)\sec\theta \quad \text{... (ii)}$$

Substituting the value of y from Eq. (i) in Eq. (ii) it becomes

$$P = \frac{2\sqrt{A}}{\sqrt{\tan\theta}}(\sec\theta) \quad \text{... (iii)}$$

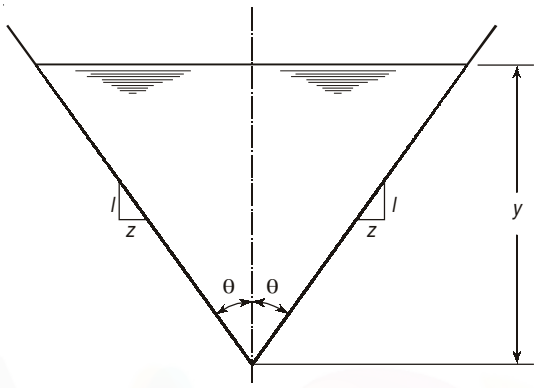


Figure 15.6 *Triangular channel section*

Assuming the area A to be constant, Eq. (iii) can be differentiated with respect to θ and equated to zero for obtaining the condition for minimum P.

Thus
$$\frac{dP}{d\theta} = 2\sqrt{A}\left[\frac{\sec\theta\tan\theta}{\sqrt{\tan\theta}} - \frac{\sec^3\theta}{2(\tan\theta)^{3/2}}\right] = 0$$

or $\sec\theta\,(2\tan^2\theta - \sec^2\theta) = 0$

Since $\sec\theta \neq 0$

$2\tan^2\theta - \sec^2\theta = 0$

or $\sqrt{2}\tan\theta = \sec\theta$

$\therefore \quad \theta = 45°$; or $z = 1$ ... (15.23)

Hence a triangular channel section will be most economical or most efficient when each of its slopping sides makes an angle of 45° with the vertical.

The hydraulic radius R of a triangular channel section can be expressed as

$$R = \frac{A}{P} = \frac{y^2\tan\theta}{2y\sec\theta}$$

Substituting the value of θ from Eq. 15.23 in the above expression

$$R = \frac{y}{2\sqrt{2}} \quad \text{... (15.24)}$$

Thus it can be seen that the most economical or most efficient triangular channel section will be half square described on a diagonal and having equal slopping sides.

It may, however, be noted that in all these cases the conditions for the discharge to be maximum would be the same if instead of the area of cross-section the perimeter is given.

(d) Most Economical or Most Efficient Circular Channel Section. For a circular channel section of any radius, as the depth of flow varies, the shape of the flow area also varies due to the convergence of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the boundaries towards the top. As such both the wetted area as well as the wetted perimeter varies with the depth of flow, and hence in the case of circular channels the condition of area of flow section being constant cannot be applied. Thus in the case of circular channels two separate conditions may be derived for the maximum discharge and the maximum mean velocity of flow, as explained below.

(*i*) ***Condition for maximum discharge through a circular channel section.*** For a circular channel section of radius r if y is the depth of flow, as shown in Fig. 15.7, the following expression for the wetted area A and the wetted perimeter P can be written

$$A = \frac{r^2}{2}(\theta - \sin\theta) \quad \text{...(i)}$$

$$P = r\theta \quad \text{... (ii)}$$

where θ is the angle subtended at the centre by the portion of circular arc in contact with water. In accordance with Chezy' formula, discharge Q passing through the channel may be expressed as

$$Q = AC\sqrt{RS} = CA\left(\sqrt{\frac{A}{P}S}\right)$$

$$= C\sqrt{\frac{A^3}{P}S}$$

where S is the bottom slope and C is Chezy coefficient. Assuming C and S to be constant, the discharge is maximum when $\left(\frac{A^3}{P}\right)$ is maximum. Thus differentiating $\left(\frac{A^3}{P}\right)$ with respect to θ and equating to zero for a maxima, we get

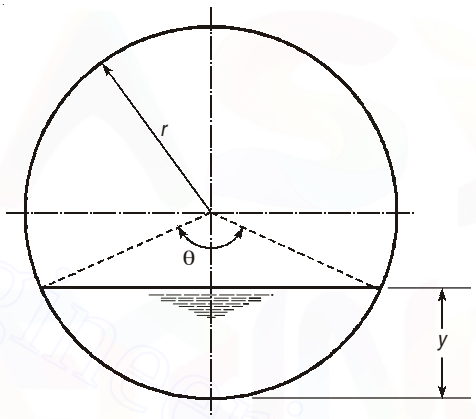


Figure 15.7 *Circular channel section*

$$\frac{d\left(\frac{A^3}{P}\right)}{d\theta} = \frac{P \times 3A^2\left(\frac{dA}{d\theta}\right) - A^3\left(\frac{dP}{d\theta}\right)}{P^2} = 0$$

or $$3P\frac{dA}{d\theta} - A\frac{dP}{d\theta} = 0$$

But from Eqs. (i) and (ii)

$$\frac{dA}{d\theta} = \frac{r^2}{2}(1 - \cos\theta)$$

and $$\frac{dP}{d\theta} = r$$

Therefore substituting for A, P, $\left(\frac{dA}{d\theta}\right)$ and $\left(\frac{dP}{d\theta}\right)$ the above expression becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{3}{2} r^3 \theta (1 - \cos\theta) - \frac{r^3}{2}(\theta - \sin\theta) = 0$$

or $3\theta (1 - \cos\theta) - (\theta - \sin\theta) = 0$

or $2\theta - 3\theta \cos\theta + \sin\theta = 0$

The solution of this equation is $\theta = 308°$ (approximately).

Then for maximum discharge the depth of flow in the channel is

$$y = r + r\cos\left(180° - \frac{\theta}{2}\right)$$

$$= r(1 + \cos 26°)$$

$$= 1.8988r \approx 0.95\, D \qquad \ldots(15.25)$$

where D is the diameter of the channel.

It is thus shown that for maximum discharge in a circular channel the depth of flow is equal to 0.95 times the diameter of the channel.

Further hydraulic radius for a circular channel is

$$R = \frac{A}{P} = \frac{r}{2\theta}(\theta - \sin\theta)$$

Substituting the value of θ as $308°$ for maximum discharge

$$R = \frac{r}{2\pi\left(\frac{308}{180}\right)}\left(\frac{\pi \times 308}{180} - \sin 308°\right)$$

$$= 0.5733\, r \approx 0.29\, D \qquad \ldots (15.25\ a)$$

That is, for maximum discharge in a circular channel the hydraulic radius is equal to 0.29 times the diameter of the channel.

The above derived conditions for the maximum discharge in a circular channel are based on the Chezy's formula which has been used in the above analysis. However, if instead of Chezy's formula Manning's formula is considered then a slightly different condition is obtained for maximum discharge in a circular channel as indicated below.

According to Manning's formula the discharge through the channel may be expressed as

$$Q = \frac{1}{n} AR^{2/3} S^{1/2} = \frac{1}{n}\left(\frac{A^5}{P^2}\right)^{1/3} S^{1/2}$$

Assuming n and S to be constant the discharge is maximum when $\left(\frac{A^5}{P^2}\right)$ is maximum. Thus differentiating $\left(\frac{A^5}{P^2}\right)$ with respect to θ and equating to zero for a maxima, we get

$$\frac{d\left(\frac{A^5}{P^2}\right)}{d\theta} = \frac{P^2 \times 5A^4\left(\frac{dA}{d\theta}\right) - A^5 \times 2P\left(\frac{dP}{d\theta}\right)}{P^4} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\left(5P\frac{dA}{d\theta}-2A\frac{dP}{d\theta}\right) = 0$$

Substituting the values of A, P, $\left(\frac{dA}{d\theta}\right)$ and $\left(\frac{dP}{d\theta}\right)$ the above expression becomes

$$\frac{5}{2}r^3\,\theta\,(1-\cos\theta)-r^3\,(\theta-\sin\theta) = 0$$

or $$3\theta-5\theta\cos\theta+2\sin\theta = 0$$

The solution of this equation is $\theta = 302°\ 20'$ (approximately).

The corresponding depth of flow is then

$$y = r + r\cos\left(180°-\frac{\theta}{2}\right)$$

$$= r\,(1+\cos\ 28°50')$$

$$= 1.876\,r = 0.938\,D.$$

That is, when Manning's formula is used, for maximum discharge in a circular channel the depth of flow is equal to 0.938 times the diameter of the channel.

(*ii*) ***Condition for maximum mean velocity of flow through a circular channel section.*** According to Chezy's or Manning's formula the mean velocity of flow through a circular channel is proportional to the hydraulic radius. As such the mean velocity of flow will have a maximum value when the hydraulic radius $\left(\frac{A}{P}\right)$ is maximum. Thus differentiating $\left(\frac{A}{P}\right)$ with respect to θ and equating to zero for a maxima, we get

$$\frac{d\left(\frac{A}{P}\right)}{d\theta} = \frac{P\left(\frac{dA}{d\theta}\right)-A\left(\frac{dP}{d\theta}\right)}{P^2} = 0$$

or $$P\,\frac{dA}{d\theta}-A\,\frac{dP}{d\theta} = 0$$

Substituting the values of A, P, $\left(\frac{dA}{d\theta}\right)$ and $\left(\frac{dP}{d\theta}\right)$ the above expression becomes

$$\frac{r^3\theta}{2}(1-\cos\theta)-\frac{r^3}{2}(\theta-\sin\theta) = 0$$

or $$\theta = \tan\theta$$

The solution of the above equation is $\theta = 257.5°$ (approximately).

The depth of flow in the channel, for maximum velocity is

$$y = r + r\cos\left(180°+\frac{\theta}{2}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= r\left[1+\cos\left(180°-\frac{257.5°}{2}\right)\right]$$

$$= r\,(1+\cos\ 51.25°\,)$$

$$= 1.626r \approx 0.81\ D \qquad \ldots(15.26)$$

That is, the depth of flow for maximum mean velocity of flow in circular channel is equal to 0.81 times the diameter of the channel.

Further the hydraulic radius for a circular channel is

$$R = \frac{A}{P} = \frac{r}{2\theta}(\theta - \cos\theta)$$

Substituting the value of θ as $257.5°$ for maximum mean velocity

$$R = \frac{r}{2\pi\left(\frac{257.5}{180}\right)}\left(\frac{\pi\times 257.5}{180} - \sin 257.5°\right)$$

$$= 0.6086r \approx 0.30\ D$$

That is, the hydraulic radius is equal to 0.30 times the diameter, for maximum mean velocity of flow in a circular channel.

15.7 OPEN CHANNEL SECTION FOR CONSTANT VELOCITY AT ALL DEPTHS OF FLOW

According to Chezy's or Manning's formula it is seen that the hydraulic radius is the sole shape parameter on which the velocity of flow in a channel laid on a constant bottom slope depends. As such the velocity of flow will be constant at all depths if the hydraulic radius is constant for any depth of flow.

Consider a profile of channel section as shown in Fig. 15.8, assumed to have constant hydraulic radius R for any depth of flow. For a small portion of the channel section considered at a depth of y and dy in thickness, as shown in Fig. 15.8, the area dA and wetted perimeter dP may be expressed as

$$dA = 2x(dy)$$

$$dP = 2\sqrt{(dx)^2 + (dy)^2}$$

where x and $(x + dx)$ are respectively half the bottom and top widths of the small portion of the area under consideration.

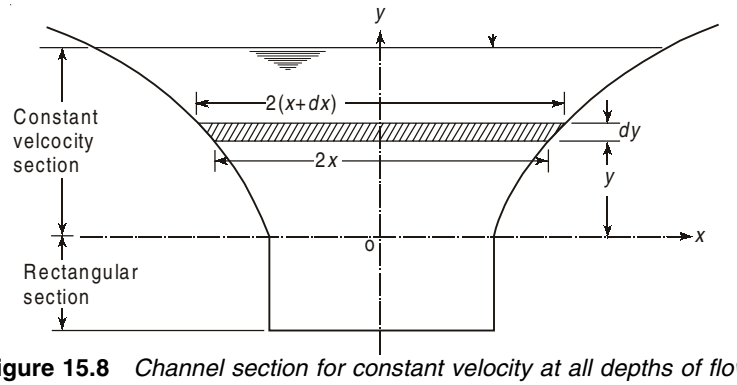


Figure 15.8 *Channel section for constant velocity at all depths of flow*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From the given condition

$$R = \frac{A}{P} = \frac{dA}{dP} = \frac{2xdy}{2\sqrt{(dx)^2 + (dy)^2}}$$

or $$R^2[(dx)^2 + (dy)^2] = x^2\,(dy)^2$$

or $$R^2\left[1 + \left(\frac{dy}{dx}\right)^2\right] = x^2\left(\frac{dy}{dx}\right)^2$$

Solving for $\left(\frac{dy}{dx}\right)$, we get

$$\frac{dy}{dx} = \frac{R}{\sqrt{x^2 - R^2}}$$

or $$dy = \frac{Rdx}{\sqrt{x^2 - R^2}}$$

Integrating the above equation, we get

$$y = R\log_e\left(x + \sqrt{x^2 - R^2}\right) + C \qquad \ldots(15.27)$$

where C is constant of integration.

Equation 15.27 defines the cross-section of a channel with constant velocity at all depths of flow. The two sides of this channel section are formed by two curves, mathematically known as catenaries. Since the theoretical section of the channel so developed is bottomless, an artificial bottom should be provided. As such below the point $y = 0$, the channel section may be either of the various types, viz., rectangular, trapezoidal, triangular, semi-circular, parabolic etc. Depending on the shape of the channel section to be provided for the bottom portion or on account of limitations imposed by certain conditions, the constant of integration C may be evaluated. For instance if the condition is such that at $y = 0$, $x = R$, the constant C becomes

$$C = -R\log_e(R)$$

Substituting this value of C in Eq. 15.27 it becomes

$$y = R\left[\log_e(x + \sqrt{x^2 - R^2} - \log_e R\right]$$

In practice it is observed that for an ordinary channel section with an increase in the depth of flow, the velocity is considerably increased, thereby causing damage to the channel section. On the contrary if the depth of flow is small, the velocity is considerably reduced. Hence in order to avoid such fluctuations of the velocity of flow and to prevent the damaging of the channel section it becomes necessary, in practice, to have a channel section having constant velocity at all depths. Particularly in the case of large sewers, in which the discharge ranges from a certain minimum value that flows daily to a very large value during the rainy season, channel sections of constant velocity are designed.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

15.8 COMPUTATION OF UNIFORM FLOW

The discharge of uniform flow in a channel may be expressed by using Chezy's formula as

$$Q = AV = CA\sqrt{RS} = K\sqrt{S} \quad \ldots(15.28)$$

where

$$K = CA\sqrt{R} \quad \ldots(15.29)$$

The term K is known as 'conveyance' of the channel section which is a measure of the carrying capacity of the channel section, since it is directly proportional to the discharge Q. Similarly when the Manning's formula is used, the discharge becomes

$$Q = AV = \frac{1}{n} AR^{2/3} S^{1/2} = K\sqrt{S} \quad \ldots(15.30)$$

where

$$K = \frac{1}{n} AR^{2/3} \quad \ldots(15.31)$$

From Eq. 15.28 or 15.30 the conveyance is

$$K = \frac{Q}{\sqrt{S}} \quad \ldots(15.32)$$

Equation 15.32 can be used to compute the conveyance when the discharge and bottom slope of the channel are given. On the other hand Eqs. 15.29 and 15.31 are used to compute the conveyance when the geometry of the wetted area of the channel section and the Chezy's C or Manning's n are given.

Since Manning's formula is used extensively, the expression $(AR^{2/3})$ is called the section factor for uniform flow computation, which is an important element in the computation of uniform flow as indicated below. From Eq. 15.31 this factor may be expressed as

$$(AR^{2/3}) = nK \quad \ldots(15.33)$$

and from Eq. 15.32

$$(AR^{2/3}) = \frac{nQ}{\sqrt{S}} \quad \ldots(15.34)$$

Equation 15.34 applies to a channel section when the flow is uniform. The right hand side of this equation contains the terms n, Q and S, but the left hand side contains the terms which depend only on the geometry of the wetted area. Therefore it indicates that for given values of n, Q and S, there is only one possible depth of flow at which the uniform flow will be maintained in any channel section, provided the value $(AR^{2/3})$ always increases with increase in depth, which is true in most of the cases. This depth of flow is known as *normal depth* which is represented by y_n. Similarly if n, S and the depth of flow are given for a channel section, it can be seen from Eq. 15.34 that there can be only one discharge for maintaining a uniform flow through the channel section. This discharge is known as *normal discharge*, which is represented by Q_n. Furthermore when Q, n and depth of flow are given for a channel section, Eq. 15.34 can be used to determine the bottom slope which will maintain the uniform flow at a given depth of flow y in the channel. The bottom slope thus determined is known as *normal slope* which is represented by S_n. Equation 15.34 is quite useful for computation and analysis of uniform flow.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

15.9 SPECIFIC ENERGY AND CRITICAL DEPTH

Since the free surface in the case of a channel flow represents the hydraulic gradient, the Bernoulli's equation applied between the sections (1) and (2) L distance apart may be written as

$$\frac{V_1^2}{2g} + y_1 + Z_1 = \frac{V_2^2}{2g} + y_2 + Z_2 + h_f$$

where h_f is the energy loss between the two sections. The term $\left(\frac{V^2}{2g} + y + Z\right)$ represents the total energy per unit weight of water at any section. However, in the solution of many problems of channel flow the concept of *specific energy* is quite useful, which was first introduced by Bakhmeteff in 1912. The specific energy of flow at any channel section is defined as the energy per unit weight of water measured with respect to the channel bottom as the datum. Thus specific energy E at any section is the sum of the depth of flow at that section and the velocity head, that is

$$E = y + \frac{V^2}{2g} = y + \frac{Q^2}{2gA^2} \qquad \ldots(15.35)$$

Since $$V = \left(\frac{Q}{A}\right)$$

It can be seen from Eq. 15.35 that, for a given channel section and discharge Q, the specific energy is a function of the depth of flow only. Thus for a given channel section and discharge, Eq. 15.35 may be represented graphically in which specific energy is plotted against the depth of flow. The curve so obtained is known as specific energy curve and is shown in Fig. 15.9. This curve has two limbs AC and BC. The lower limb AC approaches the horizontal axis (or the specific energy axis) asymptotically towards the right. The upper limb BC approaches asymptotically to the line OD which passes through the origin and has an angle of inclination equal to 45°. At any point P on this curve the ordinate represents the depth of flow and the abscissa represents the specific energy, which is equal to the sum of the depth of flow and the velocity head.

It can be seen from the specific energy curve that, there is one point C on the curve which has a minimum specific energy, thereby indicating that below this value of the specific energy the given discharge cannot occur. The depth of flow at which the specific energy is minimum is called *critical depth* y_c. Similarly the velocity of flow at the critical depth is known as *critical velocity* V_c. For any other value of the specific energy there are two possible depths, one greater than the critical depth and the other smaller than the critical depth, at which a given discharge can occur with the same specific energy. These two depths for given specific energy are called the *alternate depths*–y_1 smaller than y_c and y_2 greater than y_c as shown in Fig. 15.9. Apparently at point C the two alternate depths become one, equal to critical depth y_c. For any depth of flow greater than the critical depth, the specific energy increases with the increase in the depth. Moreover when the depth of flow is greater than the critical depth, the velocity of flow is less than the critical velocity for the given discharge. Hence the flow at the depths greater than the critical depth is known as *subcritical flow* or *tranquil flow.* On the other hand when the depth of flow is less than the critical depth, the specific energy increases as the depth of flow decreases. Furthermore for given discharge when the depth of flow is less than critical depth, the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

velocity of flow is greater than the critical velocity. Hence the flow at the depths less than the critical depth is known as *supercritical flow* or *rapid flow*. As shown in Fig. 15.9 y_1 is the depth of a supercritical flow, and y_2 is the depth of a subcritical flow.

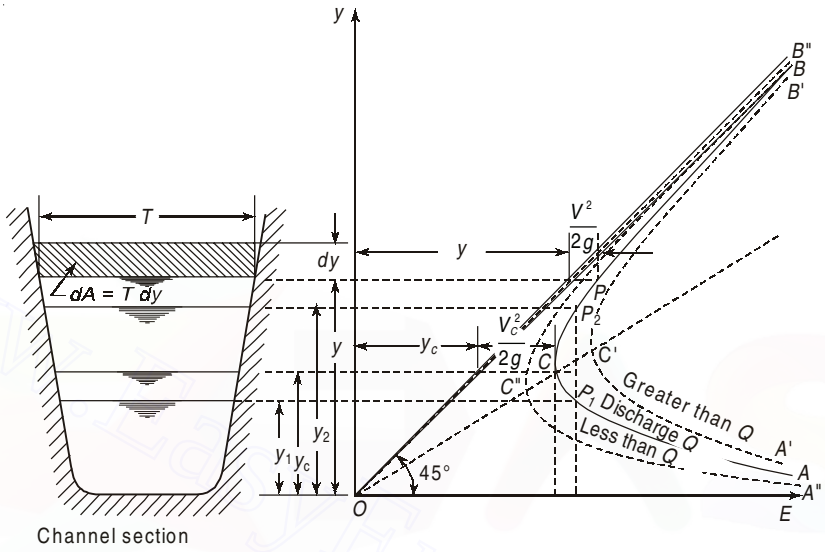


Figure 15.9 *Specific energy curve*

It is evident from Eq. 15.35 that for any channel section if the discharge changes, the specific energy will be changed accordingly. The two curves *A″ C″ B″* and *A′ C′ B′* shown in Fig. 15.9 represent the positions of the specific energy curve when the discharge is respectively less and greater than the discharge used for the construction of the curve *ACB*.

For a given discharge the condition for minimum specific energy can be obtained by differentiating Eq. 15.35 with respect to y and then considering $\left(\frac{dE}{dy}\right) = 0$. Therefore

$$\frac{dE}{dy} = \frac{d}{dy}\left(y + \frac{Q^2}{2gA^2}\right) = \left[1 - \frac{2Q^2}{2gA^3}\left(\frac{dA}{dy}\right)\right] = 0$$

Since Q is a constant and A is a function of y. As shown in Fig. 15.9, in a channel section if T is the top width of flow then differential water area dA near the free surface is equal to Tdy i.e., $dA = Tdy$ or $\left(\frac{dA}{dy}\right) = T$. By substituting this value of $\left(\frac{dA}{dy}\right)$ the above equation becomes

$$1 - \frac{Q^2 T}{gA^3} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{Q^2}{g} = \frac{A^3}{T} \quad \ldots(15.36)$$

Since $V = \left(\frac{Q}{A}\right)$ and hydraulic depth $D = \left(\frac{A}{T}\right)$, Eq. 15.36 may be written as

$$\frac{V^2}{g} = D; \text{ or } V^2 = gD \quad \ldots(15.37)$$

Equation 15.37 may also be written as $\left(\frac{V}{\sqrt{gD}}\right)$ =1. Since $\left(\frac{V}{\sqrt{gD}}\right)$ represents Froude number it may be stated that for the critical flow the Froude number, $Fr = 1$. Thus the critical state of flow may be defined in a more common way as 'it is that state of flow at which the specific energy is a minimum for the given discharge'. Further from the foregoing discussion it is obvious that in any channel section carrying a given discharge Q, if the flow is in the critical state, the cross-sectional area A and the top width T at the water surface must satisfy Eq. 15.36.

Moreover, as stated earlier when the flow is in subcritical state, since the velocity of flow is less than the critical velocity and the depth of flow is more than the critical depth, it then follows that the Froude number is less than one (i.e., $Fr < 1$). Similarly when the flow is in supercritical state, since the velocity of flow is more than the critical velocity and the depth of flow is less than the critical depth, it then follows that the Froude number is more than one (i.e., $Fr > 1$).

Solving Eq. 15.35 for Q, we get

$$Q = \sqrt{2A^2 g(E-y)} \quad \ldots(15.38)$$

In a given channel section for a given value of specific energy E, the condition for maximum discharge is obtained by putting $\left(\frac{dQ}{dy}\right) = 0$. Thus differentiating Eq. 15.38 with respect to y and equating it to zero, we get

$$\frac{dQ}{dy} = \sqrt{2g}\left[\frac{dA}{dy}\sqrt{E-y} - \frac{A}{2\sqrt{E-y}}\right] = 0$$

or $$E = \frac{1}{2}\frac{A}{T} + y \text{ ; since } \frac{dA}{dy} = T$$

Substituting the value of E obtained above in Eq. 15.38, it may be simplified and rearranged as

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

which is the criterion for the critical state of flow as represented by Eq. 15.36. It is thus observed that for a given specific energy the discharge in a given channel section is maximum when the flow is in the critical state.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

15.10 MOMENTUM IN OPEN-CHANNEL FLOW-SPECIFIC FORCE

If V is the mean velocity of flow of discharge Q in a channel, the momentum of the flow passing a channel section per unit time may be expressed by (wQV/g), where w is the specific weight of water. According to Newton's second law of motion the rate of change of momentum in the body of water flowing in a channel is equal to the resultant of all the forces that are acting on the body. Thus applying this principle to a channel of large bed slope, the following expression for the rate of change of momentum in the body of water enclosed between sections 1 and 2 may be written

$$\frac{wQ}{g}(V_2 - V_1) = P_1 - P_2 + W \sin\theta - F_f \quad \ldots(15.39)$$

where w, Q and V are previously defined with subscripts referring to sections 1 and 2; P_1 and P_2 are the resultant pressures acting on the two sections; W is the weight of water enclosed between the sections; θ is the angle of inclination of the channel bottom with the horizontal; and F_f is the total external force of frictional resistance acting in the direction opposite to the flow along the surface of contact between the water and the channel.

When the momentum equation is applied to a short horizontal reach of a prismatic channel, the external force of friction and the component of the weight of water can be ignored. Thus with $\theta = 0$ and $F_f = 0$ Eq. 15.39 becomes

$$\frac{wQ}{g}(V_2 - V_1) = P_1 - P_2 \quad \ldots(15.40)$$

Assuming hydrostatic pressure distribution at sections 1 and 2

$$P_1 = wA_1\bar{z}_1 \text{; and } P_2 = wA_2\bar{z}_2$$

where A_1, $\bar{z}_1$ and A_2, $\bar{z}_2$ are cross-sectional area and the vertical depth of the centroid of the area below the surface of flow at sections 1 and 2 respectively. Also $V_1 = (Q/A_1)$ and $V_2 = (Q/A_2)$, thus Eq. 15.40 may be written as

$$\frac{Q^2}{gA_1} + A_1\bar{z}_1 = \frac{Q^2}{gA_2} + A_2\bar{z}_2 \quad \ldots(15.41)$$

The terms on both sides of Eq. 15.41 are analogous, hence Eq. 15.41 may be expressed for any channel section by a general function

$$F = \frac{Q^2}{gA} + A\bar{z} \quad \ldots(15.42)$$

This function consists of two terms. The first term is the momentum of the flow passing through the channel per unit time per unit weight of water, and the second term is the force per unit weight of water. Since both these terms are force per unit weight of water, their sum is known as *specific force*. As such Eq. 15.41 may be expressed as $F_1 = F_2$, which means that the specific forces of sections 1 and 2 are equal, provided that external force of friction and the component of the weight of water in the channel reach between the two sections can be ignored.

It is evident from Eq. 15.42 that for a given channel section and discharge Q the specific force depends only on the depth of flow. Thus by plotting the depth of flow against the specific force for a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

given channel section and discharge, a specific force curve is obtained as shown in Fig. 15.10. Alike specific energy curve this curve also has two limbs *AC* and *BC*. The limb *AC* approaches the horizontal axis asymptotically towards right. The limb *BC* rises upward and extends indefinitely to the right. It can be seen from the curve that for a given value of specific force there are two possible depths y_1 and y_2. It has been shown in the next chapter that these two depths y_1 and y_2 constitute the initial and sequent depths of a hydraulic jump. At point *C* on the curve the two depths become one and corresponding to this depth the specific force is minimum.

For a given discharge the condition for minimum specific force can be obtained by differentiating Eq. 15.42 with respect to y and then considering $(dF/dy) = 0$. Thus

$$\frac{dF}{dy} = -\left(\frac{Q^2}{gA^2}\right)\frac{dA}{dy} + \frac{d(A\bar{z})}{dy} = 0$$

since Q is constant and both A and $\bar{z}$ are the functions of y. As shown in Fig.15.10 for a change dy in the depth, the corresponding change $d(A\bar{z})$ in the moment of the cross-sectional area about the free surface may be expressed as

$$\left[A(\bar{z}+dy)+\frac{T(dy)^2}{2}\right]-(A\bar{z}) = d(A\bar{z})$$

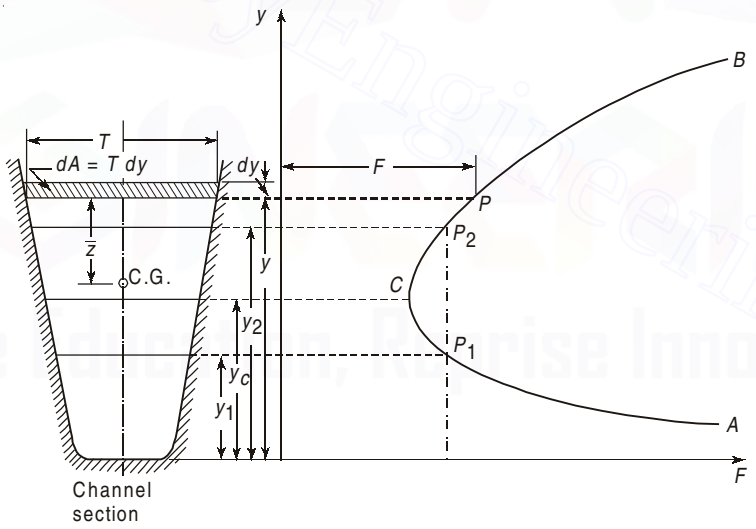


Figure 15.10 *Specific-force curve*

Assuming $(dy)^2 \approx 0$, the above expression becomes

$$d(A\bar{z}) = A(dy)$$

Substituting this value of $d(A\bar{z})$ the preceding equation becomes

$$\frac{dF}{dy} = -\left(\frac{Q^2}{gA^2}\right)\frac{dA}{dy} + \frac{A(dy)}{dy} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again since $(dA/dy) = T$, the above equation reduces to

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

which is the condition for the critical state of flow as indicated earlier. It is thus seen that the depth of flow at the minimum value of the specific force is equal to the critical depth.

Further solving Eq. 15.42 for Q, it may be expressed as

$$Q = \sqrt{(F - A\bar{z})gA} \qquad \ldots(15.43)$$

In a given channel section for a given value of specific force F, the condition for maximum discharge can be obtained by putting $(dQ/dy) = 0$. Thus differentiating Eq. 15.43 with respect to y

$$\frac{dQ}{dy} = \frac{g\left[F\left(\frac{dA}{dy}\right) - A\frac{d(A\bar{z})}{dy} - (A\bar{z})\frac{dA}{dy}\right]}{2\sqrt{(F - A\bar{z})(gA)}} = 0$$

or $$F = \frac{A^2}{T} + A\bar{z}$$

Substituting the value of F obtained above in Eq. 15.43 it may be simplified and rearranged as

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

which is again the criterion for the critical state of flow as indicated earlier. Therefore it may be stated that for a given specific force the discharge in a given channel section is maximum when the flow is in the critical state.

15.11 CRITICAL FLOW AND ITS COMPUTATION

When the depth of flow of water over a certain reach of a given channel is equal to the critical depth y_c, the flow is described as critical flow or in critical state. As indicated by the critical flow criterion, Eq. 15.36, the critical depth for a given discharge Q is the depth y_c corresponding to which the cross-sectional area A and top width T of the channel section are such that the value of (A^3/T) is given by the following expression,

$$\left(\frac{A^3}{T}\right)_c = \frac{Q^2}{g}$$

or $$\left(A\sqrt{\frac{A}{T}}\right)_c = Z_c = \frac{Q}{\sqrt{g}} \qquad \ldots(15.44)$$

Equation 15.44 indicates that the section factor for critical flow computation, $Z[= A\sqrt{A/T}\,]$ for a channel section at the critical state of flow is equal to $\left(Q/\sqrt{g}\right)$. For a prismatic channel the section factor Z is a function of the depth of flow. Hence it is evident from Eq. 15.44 that in a prismatic channel there is only one possible depth y_c which makes the given discharge Q, in a channel to flow in critical

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

state. Since the depth y_c will be same at all sections, critical flow in a prismatic channel is essentially a uniform flow. Similarly it may also be stated that, when the depth of flow in a channel section is fixed, there can be only one discharge, which maintains the critical flow at the chosen depth. This discharge is known as *critical discharge* and is represented by Q_c.

Furthermore, for a given channel section the value of the bottom slope may be so adjusted that for a given discharge it is possible to develop uniform flow at the depth of flow equal to the corresponding critical depth (or in other words it is possible to make the given discharge to flow in a critical state). This value of the bottom slope is known as *critical slope* and it is represented by S_c. On the other hand by adjusting the bottom slope and the discharge, a critical flow may be obtained at the given normal depth. The slope thus obtained is known as the *critical slope at the given normal depth or normal critical slope* which is represented by S_{cn}.

As indicated in Sections 15.9 and 15.10 the various characteristics of the critical state of flow through a channel section are as follows:

(i) The specific energy is a minimum for a given discharge.
(ii) The discharge is a maximum for a given specific energy.
(iii) The specific force is a minimum for a given discharge.
(iv) The discharge is a maximum for a given specific force.
(v) The velocity head is equal to half the hydraulic depth in a channel of small slope.
(vi) The Froude number is equal to unity.

For the computation and analysis of critical flow in an open channel, Eq. 15.44 is very useful. When the discharge is given critical section factor Z_c can be computed from Eq. 15.44. For a given channel section, since section factor Z is a function of the depth of flow y, that is $Z = f(y)$, a curve between Z and y can be prepared. With the help of this curve, corresponding to the known value of $Z = Z_c$ (determined from Eq. 15.44) the value of the depth of flow can be obtained which will be critical depth y_c. On the other hand when the depth of flow is given the corresponding value of the section factor Z can be obtained from the same curve and from this value of Z the critical discharge Q_c can be computed by Eq. 15.45, which has been obtained from Eq. 15.44 in the following form

$$Q_c = \left(\sqrt{\frac{A^3}{T}}\right) \times \left(\sqrt{g}\right) = Z\sqrt{g} \qquad \ldots(15.45)$$

For the channel sections of different shapes the computation of critical flow may be carried out by adopting the procedure as indicated below.

(a) Critical Flow in Rectangular Channels. Since for a channel of rectangular section the top width T is equal to the bottom width B, Eq. 15.36 can be further simplified. If it is assumed that q represents the discharge per unit width of the channel section then the total discharge Q passing through a channel of rectangular section of bottom width B may be expressed as $Q = (B \times q)$ or $q = \frac{Q}{B}$ Corresponding to a critical depth of flow y_c the area of rectangular channel section $A = (B \times y_c)$. By substituting these values in Eq. 15.44 it becomes

$$\frac{qB}{\sqrt{g}} = By_c\sqrt{\frac{By_c}{B}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad \frac{q^2}{g} = (y_c)^3$$

or
$$y_c = \left(\frac{q^2}{g}\right)^{1/3} \quad \ldots(15.46)$$

Equation 15.46 indicates that for a rectangular channel section the critical depth y_c can be determined if the discharge per unit width is known.

Further for a rectangular channel the specific energy Eq. 15.35 may be written as

$$E = y + \frac{(qB)^2}{2g(By)^2} = y + \frac{q^2}{2gy^2} \quad \ldots(15.47)$$

Solving Eq. 15.47 for q it becomes

$$q = \sqrt{2g(E-y)y^2} \quad \ldots(15.48)$$

For a given specific energy E, the condition for the discharge per unit width q to be maximum is obtained by putting $\left(\frac{dq}{dy}\right) = 0$. Thus differentiating Eq. 15.48 with respect to y and equating to zero, we get

$$\frac{dq}{dy} = \frac{\sqrt{2g}(2yE - 3y^2)}{2\sqrt{(E-y)y^2}} = 0$$

or
$$E = \frac{3}{2}y \quad \ldots(15.49)$$

Substituting the value of E from Eq. 15.49 in Eq. 15.48 it becomes

$$q = \sqrt{gy^3}$$

$$y^3 = \frac{q^2}{g} = y_c^3$$

It is thus seen that for a given specific energy E, the discharge per unit width q is maximum when the depth of flow y is equal to critical depth y_c. Equation 15.49 may now be expressed using y_c instead of y in the following two forms:

$$E = \frac{3}{2}y_c$$

or
$$y_c = \frac{2}{3}E \quad \ldots(15.50)$$

Using Eq. 15.48 a plot between q and y can be prepared for a given value of specific energy E. This is shown in Fig. 15.11 and is known as discharge diagram. As in the case the specific energy curve

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Fig. 15.9, for a given value of discharge, it can be seen that in the discharge diagram (Fig. 15.11) also there are two possible depths for a certain value of q, as long as q is less than a certain maximum value. Beyond this maximum value of discharge the flow is physically impossible at the given specific energy. Further corresponding to the maximum discharge the two depths become one, equal to the critical depth y_c.

For a discharge q the depth of flow less than y_c represents the supercritical state of flow, while that greater than y_c corresponds to the subcritical state of flow.

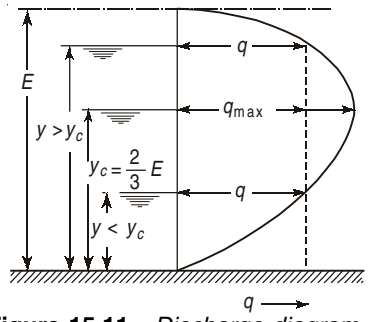


Figure 15.11 *Discharge diagram*

(b) Critical Flow in Channal Sections of other Shapes. Equation 15.36 represents a general condition for the critical flow in channel sections of any shape. As such using Eq. 15.36 the following expressions for the critical depth y_c may be obtained for the channel sections of different shapes.

(*i*)***Triangular channel section.*** For a channel of triangular section with side slope z horizontal to 1 vertical the critical depth y_c is given by the following expressions

$$y_c = \left(\frac{2Q^2}{gz^2}\right)^{1/5} \quad \ldots(15.51)$$

and

$$y_c = \frac{4}{5}E \quad \ldots(15.52)$$

(*ii*) ***Parabolic channel section.*** For a channel of parabolic section if the top width at the water surface is expressed as $T = ky^{1/2}$, the critical depth y_c is given by the following expressions

$$y_c = \left(\frac{27Q^2}{8gk^2}\right)^{1/4} \quad \ldots(15.53)$$

and

$$y_c = \frac{3}{4}E \quad \ldots(15.54)$$

(*iii*) ***Trapezoidal channel section.*** For a channel of trapezoidal section no explicit expressions for y can be obtained, but the following expressions in terms of dimensionless parameters may be developed which can be used for the computation of the critical depth y_c. Thus for a channel section of bottom width B and side slope z horizontal to 1 vertical, we have

$$\left(\frac{Q^2z^3}{gB^5}\right) = \frac{\left[\frac{B}{zy_c}+1\right]^3}{\left(\frac{B}{zy_c}\right)^5\left(\frac{B}{zy_c}+2\right)} \quad \ldots(15.55)$$

and

$$\frac{Ez}{B} = \frac{\left[3\frac{B}{zy_c}+5\right]}{2\frac{B}{zy_c}\left(\frac{B}{zy_c}+2\right)} \quad \ldots(15.56)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

15.12 APPLICATION OF SPECIFIC ENERGY AND DISCHARGE DIAGRAMS TO CHANNEL TRANSITIONS

In the case of long channels often it becomes necessary to provide transitions. A transition is the portion of a channel with varying cross-section, which connects one uniform channel to another, which may or may not have the same cross-sectional form. The variation of a channel section may be caused either by reducing or increasing the width or by raising or lowering the bottom of the channel. Various channel transitions may be broadly classified as sudden transitions and gradual transitions. Sudden transitions are those in which the change of cross-sectional dimensions occur in a relatively short length. On the other hand in the case of a gradual transition the change of cross-sectional area takes place gradually in a relatively long length of the channel. Some of the functions which channel transitions are made to serve are metering of flow, dissipation of energy, reduction or increase of velocities, change in channel section or alignment with a minimum of energy dissipation etc.

Some of the devices commonly used for measuring discharge through channels are venturi flume, standing wave flume and Parshall flume which are provided with transition as described later. The dissipation of energy may be caused by providing a sudden drop in the channel bottom. In the case of irrigation channels, in order to prevent scouring, velocities are required to be reduced, while for navigation channels in order to prevent shoaling higher velocities are required to be developed. Such reduction or increase of velocities may be made possible by expansion or contraction of channel section. The change in channel section with a minimum of energy dissipation is obtained by providing a gradual expansion or a gradual contraction with well rounded corners. Apart from the above mentioned transitions, often channel cross-section may be reduced if a bridge is constructed across it and bridge piers are constructed. Some of the channel transitions described above are shown in Fig. 15.12.

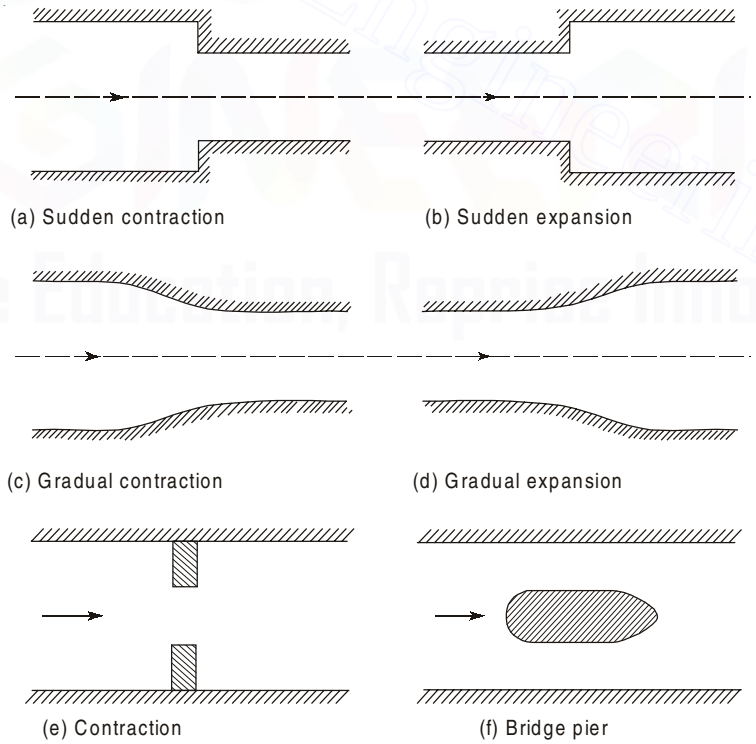


Figure 15.12 *Open channel transitions*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For a rectangular channel, transitions of decreasing channel cross-section may be analysed with the help of specific energy and discharge diagrams. The decrease in cross-section or a channel contraction may be obtained either by a decrease in width or by a decrease in depth or by a combination of both. In the following paragraphs transitions for rectangular channels with reduced width and raised bottom have been discussed separately. The combination can, however, be analysed by using the same principles in combination.

(a) Transition with Reduction in Width of a Rectangular Channel. Consider a channel of rectangular cross-section which has a reduced width as shown in Fig. 15.13. Since the flow is converging between sections 1–1 and 2–2, it may appropriately be assumed that no loss of energy takes place between the two sections. Further if it is assumed that the bottom of the channel is horizontal, the specific energy will be equal at both the sections. Let Q be the total discharge flowing through the channel. Since the total discharge Q must remain the same from section 1–1 to section 2–2, the discharge per unit width q will change from a smaller value at section 1–1 to a greater value at section 2–2.

Thus if B_1 and B_2 are the widths of the channel at sections 1–1 and 2–2 respectively then the discharge per unit width at section 1–1 is $q_1 = \left(\frac{Q}{B_1}\right)$ and at section 2–2 is $q_2 = \left(\frac{Q}{B_2}\right)$. Since B_1 is greater than B_2, q_1 is less than q_2. The contracted section 2–2 is often termed as 'throat section' or 'throat'. If the width of the throat is further reduced to B_3, as shown in Fig. 15.13, for the same discharge Q flowing

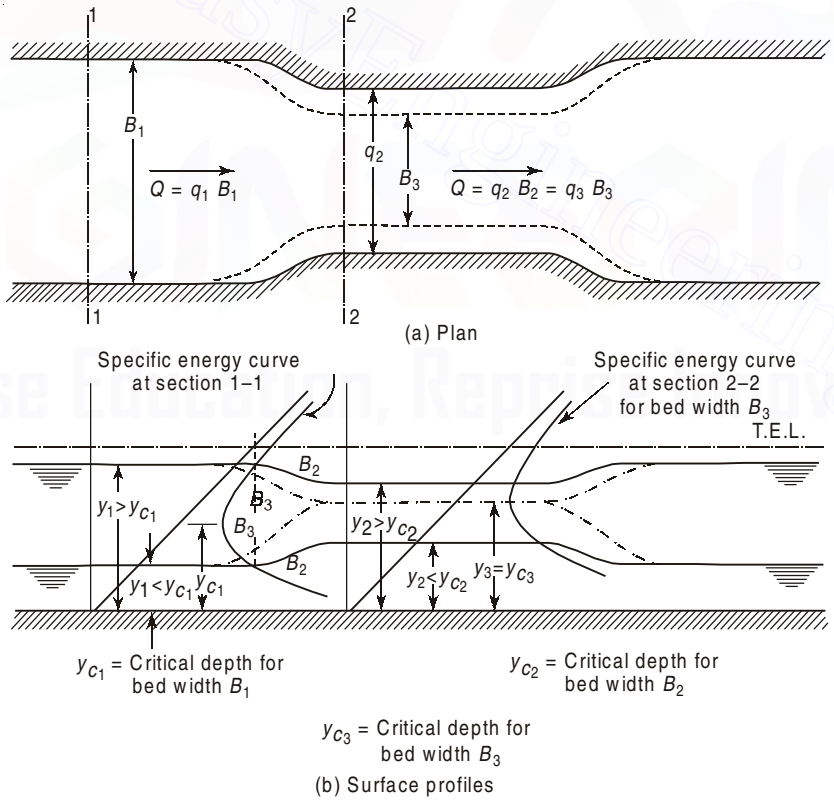


Figure 15.13 *Flow in rectangular channel with reduction in width*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

through the channel, the discharge per unit width will be increased to q_3 which is equal to $\left(\frac{Q}{B_3}\right)$. It can be seen from the discharge diagram, Fig. 15.11, that in the subcritical region of flow for a given specific energy as the discharge per unit width q is increased, the depth of flow y will decrease. This decrease in the depth of flow continues upto a certain minimum value of y equal to y_c corresponding to which the discharge per unit width is maximum equal to q_{max}. On the other hand, in the supercritical region of flow for a given specific energy, as q increases the depth of flow y also increases. Again this increase in depth of flow continues upto a certain maximum value of y equal to y_c, corresponding to which q becomes maximum equal to q_{max}, In either of the two above discussed flow conditions a further increase in q beyond q_{max} is not possible at that value of specific energy. However, if q is increased further by reducing the width of the channel section the specific energy will not remain same, but it will be increased as explained below. When the flow in the channel is in subcritical state and if the width of section 2–2 is reduced so that q becomes more than q_{max} then there will be rise in the free surface on the upstream side of section 2–2, thereby resulting in an increase in the specific energy E by such an amount that at this new value of E, the maximum discharge can pass through the contracted section. On the other hand when the flow in the channel is in supercritical state and if the width of the section 2–2 is reduced so that q becomes more than q_{max}, then free surface on the upstream side of section 2–2 will be lowered, thereby again increasing the specific energy E by such an amount that at this new value of E, the maximum discharge can pass through the contracted section.

From the foregoing discussion it is thus observed that for a given discharge flowing in a rectangular channel, there is a certain limiting value of the contracted width (say B_3) which when provided for the throat, then there will be critical depth of flow y_c (corresponding to the given discharge and the contracted section of the channel) developed at this section. If for the same total discharge flowing through the channel, the contraction at the throat is less than the above indicated value, then for subcritical flow entering the throat the depth of flow in the throat is greater than critical depth, and for a supercritical flow entering the throat, the depth of flow in the throat is less than critical depth. On the contrary for the same total discharge if the contraction at the throat is more than the above noted limiting value, then for a subcritical flow entering the throat there is heading up of water on the upstream side, and for a supercritical flow entering the throat the water surface on the upstream side is lowered; and in both these cases the depth of flow at the contracted section will be equal to the critical depth corresponding to the given discharge in the contracted section of the channel.

(b) Transition with Raised Bottom in a Rectangular Channel. Figure 15.14 shows channel of rectangular cross-section having uniform width and provided with a rise in the bottom of the channel, which is frequently known as hump. In this case since the width of the channel is not changing, the discharge per unit width will be same at different sections for a total discharge Q flowing through the channel. Since specific energy is measured with respect to channel bottom as the datum, a rise in the bottom of the channel causes a decrease in specific energy. As such if it is assumed that there is no loss of energy between the sections 1 and 2, (Fig, 15.14) the specific energy E_2 at the hump will be less than the specific energy E_1 at section 1 by an amount equal to the height of the hump ΔZ, that is

$$E_2 = E_1 - \Delta Z$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

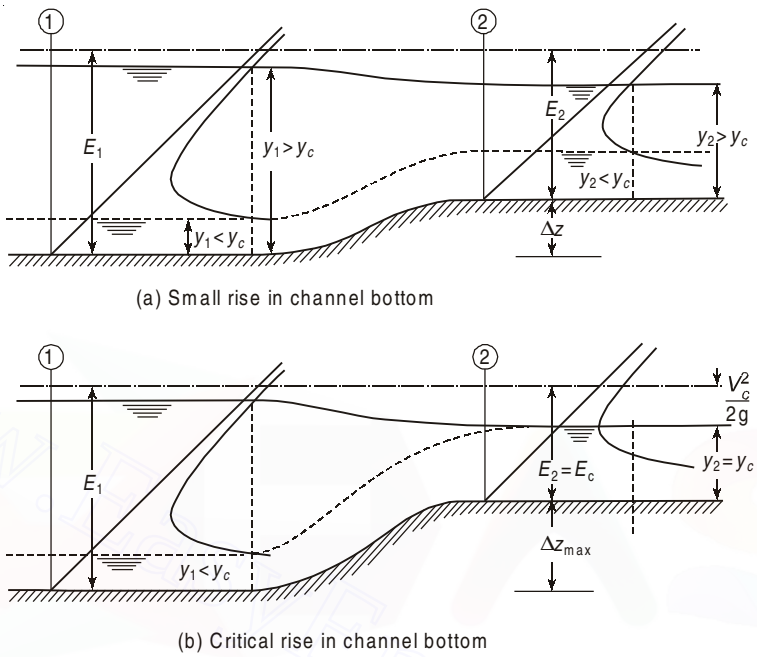


Figure 15.14 *Flow in a rectangular channel with a hump in the bottom*

Now if the flow approaching the hump is in subcritical state, the free surface drops down at the hump. This drop in the free surface at the hump can be explained with the help of specific energy diagram shown in Fig. 15.14. It can be seen from this diagram that as long as the flow is in subcritical state a decrease in specific energy is associated with a decrease in the depth of flow and increase in the velocity. Thus as shown in Fig. 15.14 (a) over the hump the depth of flow decreases and the velocity head increase. However, if the flow approaching the hump is in supercritical state, the depth of flow at the hump is increased. This increase in the depth of flow at the hump can also be explained with the help of specific energy diagram, from which it is indicated that as long as the flow is in supercritical state a decrease in specific energy is associated with an increase in the depth of flow and decrease in the velocity. It may be observed from the specific energy diagram that there is a limit upto which the specific energy for a given discharge can be reduced by increasing the height of the hump to a certain value ΔZ. Thus for given condition of flow at section 1 there is limiting or maximum value of ΔZ, as shown in Fig. 15.14 (b), at which the specific energy at section 2 is equal to the minimum specific energy for the given discharge. In other words, the height of hump can be increased to a certain maximum value without altering the condition of the approaching flow upstream of the hump. This limiting value of the height of the hump ΔZ_{max} may be computed from the following expression,

$$E_1 = E_c + \Delta Z_{max} \qquad \ldots(15.57)$$

in which E_1 is the specific energy at section 1 and E_c is the minimum possible specific energy for the given discharge. It is of course known that for a given discharge the specific energy at a channel section is minimum when the flow is in the critical state. As such when the height of the hump provided in the channel bottom is equal to that given by Eq. 15.57, the depth of flow over the hump will

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

be equal to the critical depth y_c for the given discharge and the velocity of flow will be equal to the corresponding critical velocity V_c.

However, if the height of the hump is increased further, then in order to pass the same discharge the specific energy will have to be increased. In this case for subcritical flow approaching the hump, the required increase in the specific energy will be provided by the increase in the depth of flow at section 1; and for supercritical flow approaching the hump the depth of flow at section 1 will be reduced, thereby providing the required increase in the specific energy. However, if the specific energy is held constant with the hump of height more than the above indicated limiting value, the discharge is decreased until the given specific energy is equal to the minimum specific energy corresponding to the new discharge.

15.13 METERING FLUMES

It was indicated in Section 15.12 that critical depth of flow may be obtained at certain sections in an open channel where the channel bottom is raised by the construction of a low hump or the channel is constricted by reducing its width. Since at critical state of flow the relationship between the depth of flow and discharge is definite and is independent of the channel roughness and other uncontrollable factors, it provides a theoretical basis for the measurement of discharge in open channels. As such various devices which have been developed for flow measurement are based on the principle of critical flow. Some of the devices commonly used for the measurement of flow in open channels are described below.

(a) Venturi Flume. As shown in Fig. 15.15, a venturi flume is a structure in a channel which has a contracted section called throat, downstream of which follows a flared transition section designed to restore the stream to its original width. It is an open channel counterpart of a venturi meter, which is used for measuring discharge in open channels. At the throat section there will be a drop in the water surface and this drop in water surface may be related to the discharge. The velocity of flow at the throat is always less than critical velocity and hence the discharge passing through it will be a function of the difference between the depths of flow upstream of the entrance section and at the throat. Further since the velocity of flow at the throat is less than critical velocity, standing wave or hydraulic jump will not be formed at any section in the venturi flume.

The discharge Q flowing through the channel can be calculated by measuring the depths of flow at the entrance and the throat of the flume and applying the following formula which may be readily derived

$$Q = k \frac{Aa\sqrt{2g}}{\sqrt{A^2 - a^2}} \left(\sqrt{H - h}\right) \qquad \ldots(15.58)$$

In Eq. 15.58 A, a and H, h are the areas and depths of flow section at entrance and throat of the flume respectively and k is the discharge coefficient of the flume. The discharge coefficient must be determined by calibration through the entire range of head. For a rectangular channel Eq. 15.58 may be expressed in the following form

$$Q = k \frac{BH \times bh \times \sqrt{2g}}{\sqrt{(BH)^2 - (bh)^2}} \left(\sqrt{H - h}\right) \qquad \ldots(15.59)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

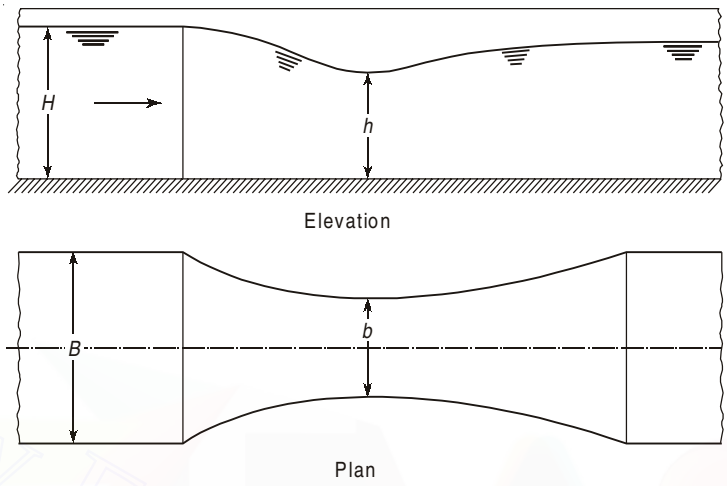


Figure 15.15 *Venturi flume*

in which B and b are the bottom widths at the entrance and the throat of the flume.

(b) Standing Wave Flume or Critical Depth Flume. A standing wave flume is a structure in a channel which has a narrowed throat having a raised floor (or hump) at the bottom as shown in Fig. 15.16, which acts as a broad-crested weir. The downstream of the throat section is followed by a flared

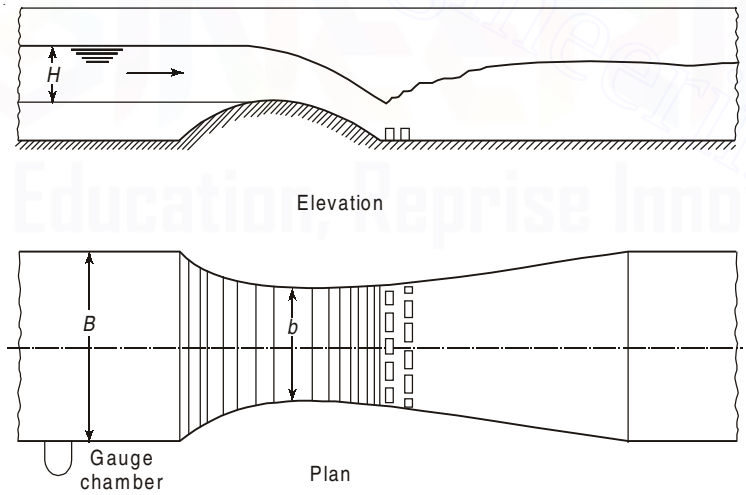


Figure 15.16 *Standing wave flume*

transition section designed to restore the stream to its original width. For any discharge flowing in the channel the velocity of flow at the throat of the flume is greater than critical velocity. As such a standing wave or hydraulic jump is invariably formed at or near the downstream end of the raised

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

floor. Moreover, since the velocity of flow at the throat is more than critical velocity, the depth of flow at a section upstream of the entrance of the flume remains unaffected by variations in the downstream depth until the downstream depth of submergence becomes greater than about 0.7 of the upstream depth. It may thus be concluded that as long as the dawnstream depth of flow is kept below the limiting value just mentioned above, the discharge passing through a standing wave flume will be a function of only the depth of flow H (above the raised floor) at a section well upstream of the entrance section, as shown in Fig. 15.16.

Thus in order to determine the discharge flowing in the channel, only the depth of flow H is required to be measured and the discharge Q may be expressed by the following relationship:

$$Q = CbH^{3/2} \qquad \ldots(15.60)$$

in which b is the width of the throat and C is the discharge coefficient of the flume. The value of C may be approximately taken as $[(2/3)^{3/2} \times \sqrt{g}\,]$ which in metric system of units becomes 1.706. However an exact value of C for any flume may be determined by calibration.

(c) Parshall Flume. The Parshall flume is one of the most widely used standing wave flume in U.S.A., which was developed by R.L. Parshall in 1920. As shown in Fig. 15.17 the converging section of the flume has a level floor, the throat section has a downward slopping floor, while the floor in the diverging section slopes upwards. The flume has no raised floor as in the case of an ordinary standing wave flume, but the upward slopping floor of the diverging section facilitates the formation of a standing wave in this portion of the flume as shown in Fig. 15.17. As such for the flumes of intermediate size having downstream depth of flow or submergence less than 0.7 times the upstream depth of flow, free flow will occur and the discharge flowing through the flume is a function of only the upstream depth. The depth–discharge relationship for such flumes may be expressed as

$$Q = CbH^{n} \qquad \ldots(15.61)$$

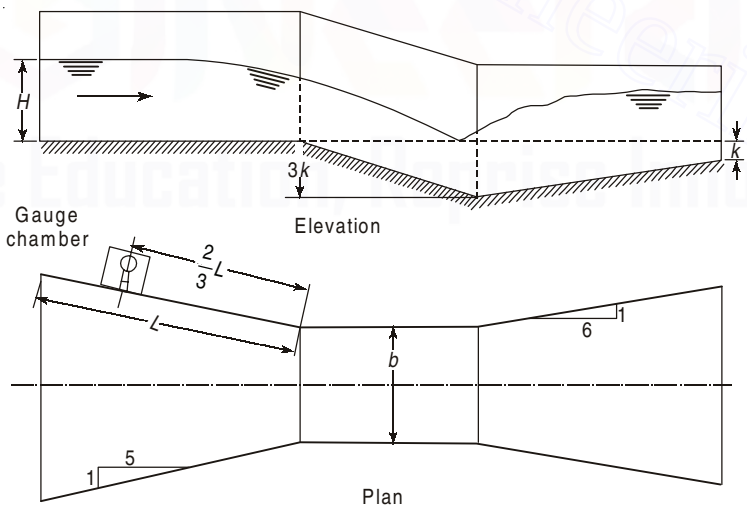


Figure 15.17 *Parshall flume*

In Eq. 15.61, Q is the free discharge, b is the width of the throat, H is the upstream depth of flow measured at gauge chamber shown in Fig. 15.17 and C and n are the constants of the flume which are

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

obtained by calibrations. When the submergence exceeds the above mentioned limit the discharge gets reduced and now if the equation for the free flow condition is used to compute the discharge, the obtained value of Q will have to be corrected by applying a suitable correction factor. Standard design for wood and concrete flumes of various capacities have been worked out for field use and the same are readily available.

15.14 DETERMINATION OF MEAN VELOCITY OF FLOW IN CHANNELS

In order to determine the discharge flowing through a channel the mean velocity of flow is required to be determined. The velocity of flow in channels may be measured by the following methods.

(a) Pitot Tube. As described in Chapter 7 pitot tube is used to measure the velocity of flow at any point. Thus with the help of a pitot tube, velocity distribution along vertical at any section of the channel can be obtained from which the mean velocity of flow can be computed. However, the mean velocity of flow in channels may also be directly obtained by measuring the velocity of flow at 0.6 times the depth below the free surface at any section; or at 0.2 and 0.8 times the depth below the free surface and then taking the mean of the two.

(b) Current Meter. A current meter as shown in Fig. 15.18 is another instrument which is commonly used for measuring the velocity of flow in channel. In general it consists of a wheel or revolving element containing blades or cups, and a tail on which flat vanes or fins are fixed. According to the shape of the revolving element the current meters may be classified as *cup type* and *screw type* or *propeller type*. In a cup type current meter the wheel or the revolving element has the form of a series of

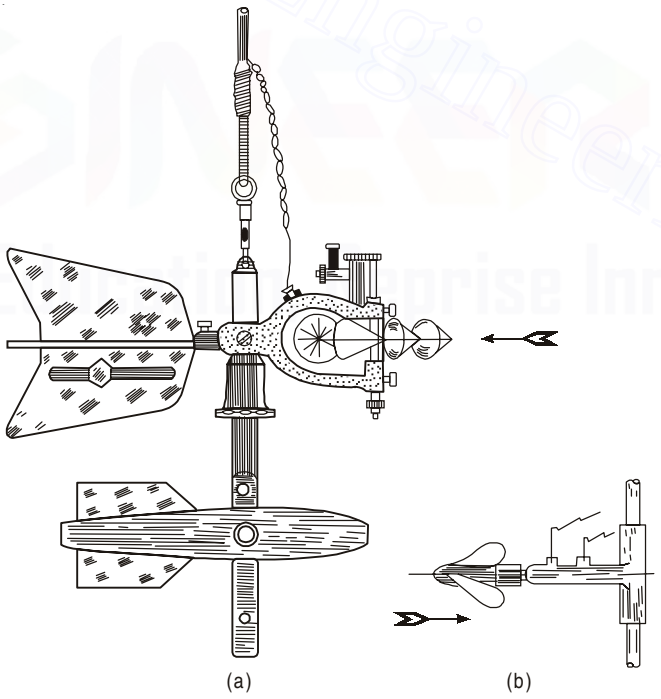


Figure 15.18 *Current meters : (a) Cup type; (b) Screw type*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

conical cups, mounted on a spindle which is held vertical at right angle to the direction of flow, Fig. 15.18 (a). In a screw or propeller type current meter the revolving element consists of a shaft, with its axis parallel to the direction of flow, which carries a number of curved vanes or propeller blades mounted around the periphery of the shaft, as shown in Fig. 15.18 (b).

The current meter is suspended by means of a cable and it is held vertically immersed in the flowing stream of water to the required depth, such that the wheel or the revolving element is facing towards the upstream direction. The wheel is thus rotated by the dynamic thrust exerted on it by the flowing water. The number of revolutions of the wheel per unit time are proportional to the velocity of the flowing water. Thus after setting the current meter to the required point in the flowing stream of water the number of revolutions of the wheel per unit time are noted and from the calibration curve the velocity of flow of water may be determined. In order to facilitate the counting of the number of revolutions of the wheel, usually an electrical transmission system is provided. An electric current is passed to the wheel from a battery above the water by means of wires, and a commutator is fixed to the spindle of the revolving blades or cups which makes and breaks the electric circuit during each revolution. An electric bell or a head phone is connected in series in the electric circuit to give an audible signal thereby facilitating the counting of the number of revolutions. Sometimes instead of these a revolution counter is provided. The number of signals in a given time is noted by the observer with a stop watch, from which the speed of the wheel in terms of revolutions per minute may be computed. The current meter is required to be calibrated experimentally. This is done by towing the current meter at various speeds through still water contained in a long tank, and noting the speed of the towing carriage and the rotational speed of the wheel of the current meter. The calibration curve for the current meter may then be prepared by plotting the rotational speed in revolution per minute versus the speed of the towing carriage in metres per second, which will correspond to the velocity of flow of water in the channel when the current meter is held stationary in the water flowing in the channel.

(c) Floats. A float is a small object made of wood or other suitable material which is lighter than water and thus capable of floating on the water surface. It provides a simple way of measuring the velocity of flow of water in rivers and channels. The surface velocity at any section may be obtained by a *single float*. For this the time taken by the float to traverse a known distance is measured and the velocity may be computed. Since the mean velocity of flow is equal to about 0.8 to 0.95 times the surface velocity, the approximate value of the mean velocity of flow may be determined from the known value of the surface velocity.

A better method of determining the velocity of flow of water is to use *double float*, as shown in Fig. 15.19 (a). It consists of a spherical surface float to which is attached a hollow metal sphere, heavier than water and suspended from the float by a cord of known length. The depth of the lower float which is completely submerged may be regulated by the length of the cord. The velocity is then obtained by noting the time taken by the float to traverse a known distance as in the previous case of a single float. However this float gives a velocity equal to the mean between the velocity of the surface and the velocity of the layer traversed by the lower float.

A still better type of float is the *rod float* as shown in Fig. 15.19 (b). It consists of a vertical wooden rod which is weighted at the bottom to keep it vertical with its top end emerging out of free water surface when floating. The length of the rod float should be such that it reaches the bottom of the stream

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

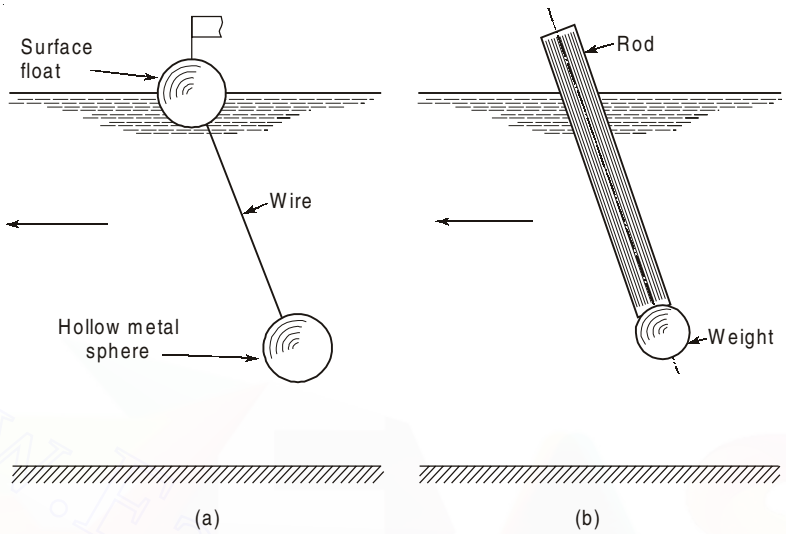


Figure 15.19 *(a) Double float; (b) Rod float*

without touching the sand, weed or mud at the bottom of the stream. Sometimes the rod floats are made telescopic so that the length may be adjusted to suit any depth of flow. The rod floats will travel with a velocity equal to the mean velocity of flow and hence a rod float gives the mean velocity of flow at the section.

15.15 PRACTICAL CHANNEL SECTIONS

In practice the most economical sections for channels obtained in Section 15.6 can not be adopted. This is so because of the two main reasons. Firstly the side slopes to be provided for a channel depends on the type of the soil through which the channel is carried and therefore it may not be possible to adopt 1 : 1 side slope for a triangular channel as indicated by Eq. 15.23. Secondly the sharp corners in the channel cross-section are virtually the zones of stagnation and may lead to the deposition of silt. As such in actual practice the side slopes of the channel sections are required to be modified and also it is desirable to have rounded corners. However for unlined channels the corners can not be made round during construction but due to silting the channel section gets modified. Moreover, for the design of an unlined channel altogether different considerations are required to be made as indicated in Chapter 28. For a lined channel the Indian practice is to adopt a triangular section with permissible side slopes and rounded bottom as shown in Fig. 15.20 (a) for discharges less than 55 cumec and a trapezoidal section with rounded corners as shown in Fig. 15.20 (b) for higher discharges. In a lined channel relatively higher mean velocity is permissible which is restricted to 2.5 m/s. As such the design of a lined channel is based on a limiting velocity rather than any other consideration.

The expressions for the area A, perimeter P and hydraulic mean radius R for the channel sections shown in Fig. 15.20 are as follows.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

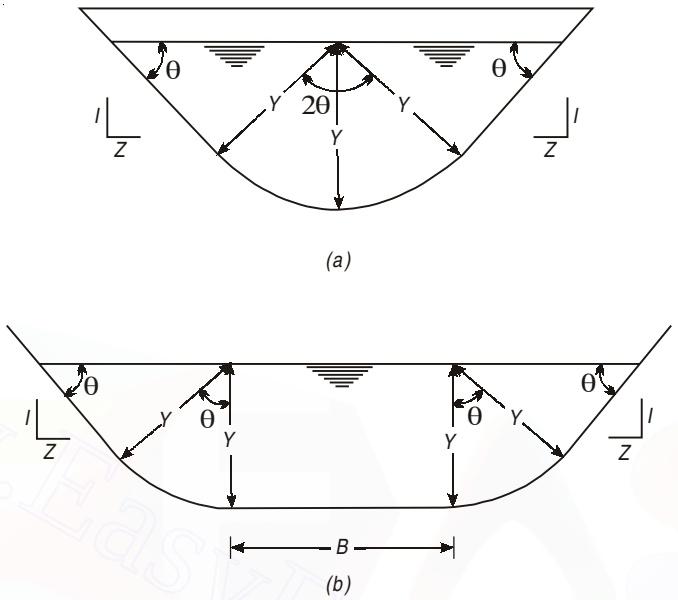


Figure 15.20 *Lined channel section (a) for discharge less than 55 cumec, (b) for discharge more than 55 cumec.*

(a) Triangular Channel Section

$$A = y^2(\theta + \cot\theta)$$
$$P = 2y(\theta + \cot\theta)$$
$$R = \frac{y}{2}$$

(b) Trapezoidal Channel Section

$$A = y(B + y\theta + y\cot\theta)$$
$$P = (B + 2y\theta + 2y\cot\theta)$$
$$R = \frac{y(B + y\theta + y\cot\theta)}{(B + 2y\theta + 2y\cot\theta)}$$

15.16 MEASUREMENT OF DISCHARGE IN RIVERS

A river is an open channel of irregular cross-section. The discharge through a small stream or river may be obtained by fitting a notch or weir across the stream and then measuring the head over the notch as indicated in Chapter 10. However, this method cannot be used for measuring the discharge through a large river. In order to obtain the discharge through a large river, the cross-section of the river is divided into a number of segments, *AB*, *BC*, *CD*, etc., of length l_1, l_2, l_3, etc. as shown in

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Fig. 15.22. The vertical lines 1, 2, 3, etc., represent the centre lines of each of the segments. The mean depth of flow d of each of the segments may be measured along the centre lines by soundings. Also the mean velocity of flow through each of the segments may also be measured with the help of a current meter by inserting it along the centre line of the segment to a depth equal to 0.6 times the mean depth below the free water surface. Thus the mean area of each segment may be computed and knowing the mean velocity of flow V through each segment the discharge through any segment is $(ld)V$ and the total discharge through the river is obtained by adding the discharges through each of the segments as

$$Q = [l_1 d_1 V_1 + l_2 d_2 V_2 + l_3 d_3 V_3 + ...] = \Sigma (ld)V.$$

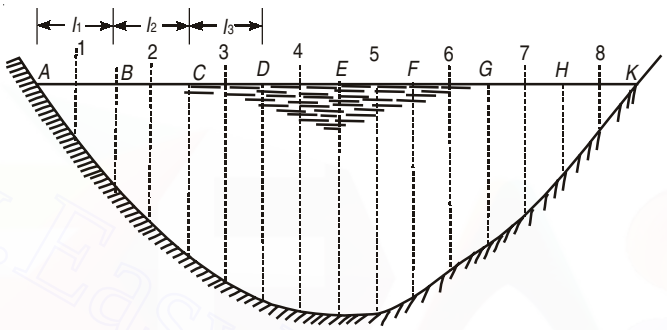


Figure 15.21 *Cross-section of river with unequal segments*

A greater accuracy in the computation of the discharge may be obtained by dividing the river width into the segments of equal width l as shown in Fig. 15.22 and then using either Simpson's rule or Weddle's rule. Simpson's rule requires that the number of segments should be a multiple of 2 and Weddle's rule requires that the number of segments should be a multiple of 6.

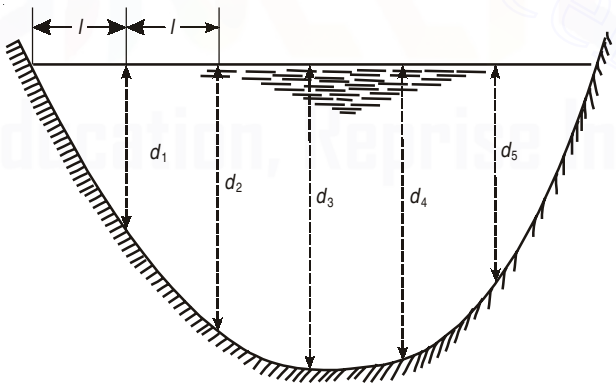


Figure 15.22 *Cross-section of river with equal segments*

Now if d_1, d_2, d_3, etc. and V_1, V_2, V_3, etc., are the respective depths and velocities at the ends of the segments marked 1, 2, 3, etc., and the velocity at the bank being taken zero, then according to Simpson's rule total discharge Q through the river is given as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = \frac{l}{3}[0 + 4d_1V_1 + 2d_2V_2 + 4d_3V_3 + 2d_4V_4 + 4d_5V_5 + 0]$$

$$= \frac{2l}{3}[d_2V_2 + d_4V_4 + 2(d_1V_1 + d_3V_3 + d_5V_5)]$$

Similarly according to Weddle's rule the total discharge Q through the river is given as

$$Q = \frac{3l}{10}[0 + 5d_1V_1 + d_2V_2 + 5d_3V_3 + d_4V_4 + 5d_5V_5 + 0]$$

$$= \frac{3l}{10}[d_2V_2 + d_4V_4 + 5(d_1V_1 + d_3V_3 + d_5V_5)]$$

If the cross-section of the river in the central portion is more or less uniform then it is preferred to use a combination of Simpson's and Weddle's rule. For this the channel is divided into ten equal segements. The discharge through the central six segments is calculated by Weddle's rule and that of the side portions by Simpson's rule.

ILLUSTRATIVE EXAMPLES

Example 15.1. *An irrigation channel of trapezoidal section, having side slopes 3 horizontal to 2 vertical, is to carry a flow of 10 cumec on a longitudinal slope of 1 in 5000. The channel is to be lined for which the value of friction coefficient in Manning's formula is n = 0.012. Find the dimensions of the most economic section of the channel.*

Solution

For most economic trapozoidal channel section from Eq. 15.17, we have

$$\frac{B + 2zy}{2} = y\sqrt{1 + z^2}$$

Since $z = \frac{3}{2} = 1.5$, we have

$$B + 3y = 2y\sqrt{1 + (1.5)^2} = 3.6y$$

or $$B = 0.6y$$

Also from Eq. 15.18

$$R = \frac{y}{2}$$

Using Manning's equation

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

or $$Q = \frac{1}{n} AR^{2/3} S^{1/2}$$

$$Q = 10 \text{ cumec}; S = \frac{1}{5000}; n = 0.012$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Also $A = (B + zy)y = (0.6y + 1.5y)y = 2.1y^2$

Thus by substitution, we get

$$10 = \frac{1}{0.012} \times 2.1y^2 \times \left(\frac{y}{2}\right)^{2/3} \left(\frac{1}{5000}\right)^{1/2}$$

or $y^{8/3} = 6.414$

$\therefore$ $y = 2$ m and $B = 1.2$ m.

Example 15.2. *A most efficient trapezoidal section is required to give a maximum discharge of 21.5 m^3/s of water. The slope of the channel bottom is 1 in 2500. Taking C = 70 $m^{1/2}/s$ in Chezy's equation, determine the dimensions of the channel. Also determine the value of Manning's n, taking the value of velocity of flow as obtained for the channel by Chezy's equation.*

Solution

For most efficient trapezoidal section of channel taking side slope $z = \left(1/\sqrt{3}\right)$, from Eq. 15.17 or 15.20, we get

$$\frac{B}{y} = \frac{2}{\sqrt{3}} = 1.1547$$

or $B = 1.1547y$

Using Chezy's equation

$$V = C\sqrt{RS}$$

or $$Q = CA\sqrt{RS}$$

$$Q = 21.5 \text{ m}^3/s; \text{C} = 70; \; S = \frac{1}{2500};$$

$$A = (B + zy)\, y$$

$$= \left(1.1547y + \frac{1}{\sqrt{3}}y\right)y = 1.7321y^2$$

and $$R = \frac{y}{2} = 0.5y \quad \text{(Eq. 15.18)}$$

Thus by substitution, we get

$$21.5 = 70 \times 1.7321y^2 \times \sqrt{0.5y \times \frac{1}{2500}}$$

or $y^{5/2} = 12.539$

$\therefore$ $y = 2.75$ m

and $B = 1.1547 \times 2.75 = 3.175$ m

The wetted perimeter P is given by

$$P = B + 2y\sqrt{1 + z^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 3.175 + 2 \times 2.75 \times \sqrt{1 + \left(\frac{1}{\sqrt{3}}\right)^2}$$

$$= 9.526 \text{ m}$$

As a check it may be shown that if side slope z is taken either more or less than $\left(1/\sqrt{3}\right)$ the wetted perimeter P is more than the one obtained above for side slope $z = \left(1/\sqrt{3}\right) = 0.5774$.

Thus taking side slope $z = 1$, we have from Eq. 15.17

$$\frac{B + 2zy}{2} = y\sqrt{1 + z^2}$$

or $$B = 0.828\, y$$

Using Chezy's equation

$$V = C\sqrt{RS}$$

or $$Q = CA\sqrt{R}$$

$$Q = 21.5 \text{ m}^3/\text{s}; \; C = 70; \; S = \frac{1}{2500};$$

$$A = (B + zy)\, y$$

$$= (0.828y + 1 \times y)y = 1.828y^2$$

and $$R = \frac{y}{2} = 0.5y \quad \text{(Eq. 15.18)}$$

Thus by substitution, we get

$$21.5 = 70 \times 1.828y^2 \times \sqrt{0.5y \times \frac{1}{2500}}$$

or $$y^{5/2} = 11.881$$

$\therefore$ $$y = 2.692 \text{ m}$$

and $$B = 2.229 \text{ m}$$

The wetted perimeter is given by

$$P = B + 2y\sqrt{1 + z^2}$$

$$= 2.229 + 2 \times 2.692 \times \sqrt{1 + (1)^2}$$

$$= 9.843 \text{ m}$$

which is more than the one obtained above for side slope $z = \left(1/\sqrt{3}\right) = 0.5774$.

Similarly taking side slope $z = 0.4$, we have from Eq. 15.18

$$\frac{B + 2zy}{2} = y\sqrt{1 + z^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $B = 1.354y$

Using Chezy's equation

$$V = C\sqrt{RS}$$

or $$Q = CA\sqrt{RS}$$

$$Q = 21.5\ \text{m}^3/\text{s};\ C = 70;\ S = \frac{1}{2500}$$

$$A = (B + zy)y$$
$$= (1.354y + 0.4y)y = 1.754y^2$$

and $$R = \frac{y}{2} = 0.5y$$

Thus by substitution, we get

$$21.5 = 70 \times 1.754y^2 \times \sqrt{0.5y \times \frac{1}{2500}}$$

or $$y^{5/2} = 12.382$$

$\therefore$ $$y = 2.736\ \text{m}$$

and $$B = 3.705\ \text{m}$$

The wetted perimeter is given by

$$P = B + 2y\sqrt{1 + z^2}$$

$$= 3.705 + 2 \times 2.736 \times \sqrt{1 + (0.4)^2}$$

$$= 9.599\ \text{m}$$

which is more than the one obtained above for side slope $z = \left(1/\sqrt{3}\right) = 0.5774$.

Thus the dimensions of the most efficient trapezoidal section of the channel are

$$B = 3.175\ \text{m};\ y = 2.75\ \text{m};\ \text{and}\ z = \left(1/\sqrt{3}\right) = 0.5774.$$

From Eq. 15.6, we have

$$C = \frac{1}{n}R^{1/6}$$

$$R = \frac{y}{2} = \frac{2.75}{2} = 1.375\ \text{m}$$

$$C = 70$$

Thus by substitution, we get

$$70 = \frac{1}{n} \times (1.375)^{1/6}$$

or $$n = \frac{(1.375)^{1/6}}{70} = 0.015$$

Manning's $$n = 0.015$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 15.3. *An irrigation channel is to carry a discharge of 14 cumec with a velocity of 0.9 m/s and bed slope of 1 in 2500. The side slopes are 1 to 1. Find the depth and bottom width. The values of Chezy's C for this channel for different values of hydraulic radius R are as tabulated below.*

Hydraulic Radius R	*0.7*	*0.8*	*0.9*	*1.0*	*1.1*	*1.2*	*1.3*	*1.4*
Chezy's C	*34*	*35*	*37*	*38*	*39*	*40*	*41*	*41*

Solution

Discharge $Q = 14\ m^3/s$

Velocity $V = 0.9\ m/s$

Area of flow section

$$A = \frac{Q}{V}$$

$$\therefore \quad A = \frac{14}{0.9} = 15.56\ m^2$$

From Chezy's formula

$$V = C\sqrt{RS}$$

Bed slope $S = \frac{1}{2500}$

Thus by substitution, we get

$$0.9 = C\sqrt{R \times \frac{1}{2500}}$$

or $C\sqrt{R} = 45$

Assuming $R = 1.0$, then $C = 38$

$\therefore \quad C\sqrt{R} = 38$

Assuming $R = 1.3$, then $C = 41$

$\therefore \quad C\sqrt{R} = 46.75$

Assuming $R = 1.241$, then $C = 40.41$

$\therefore \quad C\sqrt{R} = 45$

which is same as the value for this case as obtained above

$\therefore \quad R = 1.241$

We have

$$A = (B + y)y = 15.56 \qquad \ldots \text{(i)}$$

$$R = \frac{15.56}{B + 2y\sqrt{2}} = 1.241 \qquad \ldots \text{(ii)}$$

From Eq. (ii), we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$B = (12.54 - 2.83y)$$

Substituting the value of B in Eq. (i), we get

$$(12.54 - 2.83\,y + y)\,y = 15.56$$

or $$12.54\,y - 1.83\,y^2 = 15.56$$

or $$y^2 - 6.85\,y + 8.50 = 0$$

$$\therefore \quad y = 1.63 \text{ m}$$

Substituting the value of y in Eq. (i), we get

$$B = 7.92 \text{ m}$$

Hence the required dimensions of the channel are

$$B = 7.92 \text{ m; and } y = 1.63 \text{ m}$$

Example 15.4. *A channel carries a discharge of 5 cumec with a flow depth 1 m. The side slopes are 1 to 1 and bed slope 1 in 3520. Find the bed width and the velocity. The values of Chezy's C for this channel for different values of hydraulic radius R are as tabulated below.*

Hydraulic Radius R	*0.7*	*0.8*	*0.9*	*1.0*	*1.1*	*1.2*	*1.3*	*1.4*
Chezy's C	*34*	*35*	*37*	*38*	*39*	*40*	*41*	*41*

Solution

Discharge $Q = 5 \text{ m}^3/\text{s}$

Flow depth $y = 1 \text{ m}$

Bed slope $S = \dfrac{1}{3520}$

Assuming $R = 1.0$, then $C = 38$

From Chezy's formula

$$V = C\sqrt{RS}$$

Thus by substitution, we get

$$V = 38\sqrt{1 \times \frac{1}{3520}}$$

or $$V = 0.64 \text{ m/s}$$

$\therefore$ Area of flow section

$$A = \frac{5}{0.64} = 7.81 \text{ m}^2$$

We have

$$A = (B + y)\,y$$

Thus by substitution, we get

$$(B + 1)\,1 = 7.81$$

$$\therefore \quad B = 6.81 \text{ m}$$

Wetted perimeter $P = B + 2y\sqrt{2}$

or $$P = (6.81 + 2.83) = 9.64 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence the correct value of R is obtained as

$$R = \frac{7.81}{9.64} = 0.81$$

For this value of R, $C = 35.2$

Thus $V = 35.2\sqrt{0.81 \times \frac{1}{3520}}$

or $V = 0.53$ m/s

$$\therefore \quad A = \frac{5}{0.53} = 9.43 \text{ m}^2$$

We have $A = (B + 1) \times 1 = 9.43$

$\therefore$ $B = 8.43$ m

Hence $V = 0.53$ m/s; and $B = 8.43$ m

Example 15.5. *A channel is 2m wide at bottom; the length of each sloping side is 1.95 m; the width at water surface is 5.5 m the flow depth 1.2 m; and bed slope 1 in 5280. What is the discharge per minute? The values of Chezy's C for this channel for different values of hydraulic radius R are as tabulated below.*

Hydraulic Radius R	*0.7*	*0.8*	*0.9*	*1.0*	*1.1*	*1.2*	*1.3*	*1.4*
Chezy's C	*34*	*35*	*37*	*38*	*39*	*40*	*41*	*41*

Solution

Bottom width $B = 2$ m

Flow Depth $y = 1.2$ m

Bed slope $S = \frac{1}{5280}$

Length of slopping side

$$y\sqrt{1+z^2} = 1.95 \text{ m}$$

Width at water surface

$$B + 2zy = 5.5 \text{ m}$$

Substituting the value of B, we get

$$2 + 2zy = 5.5$$

or $zy = 1.75$

$\therefore$ Area of flow section

$$A = (B + zy)y$$
$$= (2 + 1.75)\ 1.2$$
$$= 4.5 \text{ m}^2$$

Wetted perimeter

$$P = B + 2y\sqrt{1+z^2}$$
$$= 2 + 2(1.95)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= 5.9$ m

$\therefore$ Hydraulic radius

$$R = \frac{4.5}{5.9} = 0.76$$

For this value of R, $C = 34.6$

$\therefore$ Veloity of flow

$$V = C\sqrt{RS}$$

Thus by substitution, we get

$$v = 34.6\sqrt{0.76 \times \frac{1}{5280}}$$

$$= 0.415 \text{ m/s}$$

$\therefore$ Discharge $Q = 4.5 \times 0.415$

$= 1.8675 \text{ m}^3/\text{s}$

$= 112.05 \text{ m}^3$ per minute

Example 15.6. *Water flows at a uniform depth of 2 m in a trapezoidal channel having a bottom width 6 m, side slopes 2 horizontal to 1 vertical. If it has to carry a discharge of 65 m^3/s, compute the bottom slope required to be provided. Take Manning's n = 0.025.*

Solution

According to Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

From the given data

$$A = (B + zy) = (6 + 2 \times 2)2 = 20 \text{ m}^2$$

$$P = B + 2y\sqrt{1+z^2} = 6 + 2 \times 2\left(\sqrt{5}\right) = 14.94 \text{ m}$$

$$R = \frac{A}{P} = \frac{20}{14.94} = 1.34$$

$$V = \frac{Q}{A} = \frac{65}{20} = 3.25 \text{ m/s}$$

By substituting, we have

$$3.25 = \frac{1}{0.025}(1.34)^{2/3} S^{1/2}$$

or $S^{1/2} = 0.066\ 85$

$\therefore$ $S = 0.0045$

Example 15.7. *A flow of 100 litres per second flows down in rectangular laboratory flume of width 0.6 m and having adjustable bottom slope. If Chezy's C is 56 determine the bottom slope necessary for uniform flow with a depth of flow 0.3 m. Also find the conveyance and the state of flow (i.e., tranquil or rapid).*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

The discharge flowing in the flume is

$$Q = 100\ l/\text{s} = 100 \times 10^{-3} = 0.1\ \text{m}^3/\text{s}$$

The area of flow section is

$$A = 0.6 \times 0.3 = 0.18\ \text{m}^2$$

Thus mean velocity of flow is

$$V = \frac{Q}{A} = \frac{0.10}{0.18} = 0.56\ \text{m/s}$$

The wetted perimeter of the flow section is

$$P = [B + 2y] = [0.6 + 2(0.3)] = 1.2\ \text{m}$$

The hydraulic radius is

$$R = \frac{A}{P} = \frac{0.18}{1.2} = 0.15\ \text{m}$$

From Chezy's formula, $V = C\sqrt{RS}$

Thus by substituting the computed quantities

$$0.56 = 56\sqrt{0.15 \times S}$$

$$\therefore \quad S = \frac{1}{0.15 \times 10^4} = \frac{1}{1500}$$

i.e., the required bottom slope is 1 in 1 500.

From Eq. 15.29, conveyance is given by

$$K = CA\sqrt{R} = 56 \times 0.18\ (0.15)^{1/2} = 3.904$$

The Froude number of the flow is

$$Fr = \frac{V}{\sqrt{gD}}$$

$$D = 0.30\ \text{m}$$

Thus

$$Fr = \frac{0.56}{\sqrt{9.81 \times 0.30}} = 0.326$$

Since $Fr < 1$, the flow in the flume is tranquil.

Example 15.8. *Derive the correlation between roughness factor f and Manning's coefficient n, in open channel flow.*

Solution

According to Chezy's formula from Eq. 15.2, we have

$$V = \sqrt{\frac{8g}{f}}\sqrt{RS} \qquad \ldots (i)$$

From Manning's formula, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V = \frac{1}{n} R^{2/3} S^{1/2} \quad \text{(ii)}$$

From Eq. (i) and (ii), we obtain

$$\sqrt{\frac{8g}{f}} R^{1/2} S^{1/2} = \frac{1}{n} R^{2/3} S^{1/2}$$

or $$\sqrt{\frac{8g}{f}} = \frac{1}{n} R^{1/6}$$

or $$\frac{8g}{f} = \frac{1}{n^2} R^{1/3}$$

which is the required correlation between roughness factor f, and Manning's coefficient n.

Example 15.9. *An earthen channel with a base width 2 m and side slope 1 horizontal to 2 vertical carries water with a depth of 1 m. The bed slope is 1 in 625. Calculate the discharge if n = 0.03. Also calculate the average shear stress at the channel boundary.*

Solution

$$B = 2 \text{ m}, y = 1 \text{ m}, z = \frac{1}{2}$$

Thus area of flow section is

$$A = (B + zy)y = \left(2 + 1 \times \frac{1}{2}\right) 1 = 2.5 \text{ m}^2$$

Also wetted perimeter is

$$P = (B + 2y\sqrt{1 + z^2}) = [2 + 2 \times 1 \times \sqrt{(5/4)}] = 4.24 \text{ m}$$

Thus $$R = \frac{A}{P} = \frac{2.5}{4.24} = 0.59$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

$$= \frac{1}{0.03} \times (0.59)^{2/3} \times \left(\frac{1}{625}\right)^{1/2} = 0.94 \text{ m/s}$$

∴ Discharge $Q = A \times V = (2.5 \times 0.94) = 2.35 \text{ m}^3/\text{s}$

Average shear stress at the channel boundary,

$$\tau_0 = wRS = 9810 \ 0.59 \times \left(\frac{1}{625}\right) = 9.26 \text{ N/m}^2$$

Example 15.10. *Compute the average shear stress at the sides and the bottom of a rectangular flume 3.2 m wide, flowing 1.1 m deep and laid on a slope of 1.3 m per 1000 m.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Bed slope $S = \frac{1.3}{1000}$

Aera of flow section $A = (3.2 \times 1.1)$

$= 3.52 \text{ m}^2$

Wetted perimeter $P = [3.2 + (2 \times 1.1)]$

$= 5.4 \text{ m}$

Hydraculic radius $R = \frac{3.52}{5.4}$

$= 0.652 \text{ m}$

Average shear stress on sides

$= 0.75\ wRS$

$= 0.75 \times 9810 \times 0.652 \times \frac{1.3}{1000}$

$= 6.236 \text{ N/m}^2$

Average shear stress on bottom

$= wRS$

$= 9810 \times 0.652 \times \frac{1.3}{1000}$

$= 8.315 \text{ N/m}^2$

Example 15.11. *A trapezoidal channel with bottom width 3.5 m and side slopes 1 H : 1 V on the left and 1.5H : 1V on the right, with n = 0.016, and a bed slope of 2.6 in 10000 carries a discharge of 8 m^3/s. Determine the normal depth and the average shear stress on the channel bed.*

Solution

Let y be the normal depth of flow.

Then

$$A = \left(By + \frac{1}{2}z_1y^2 + \frac{1}{2}z_2y^2\right)$$

$$= (3.5y + 0.5y^2 + 0.75y^2)$$

$$= (3.5y + 1.25y^2)$$

and $$P = \left(B + y\sqrt{1+z_1^2} + y\sqrt{1+z_2^2}\right)$$

$$= \left(3.5 + \sqrt{2}y + \frac{1}{2}\sqrt{13}y\right)$$

$$= (3.5 + 3.217y)$$

$$R = \frac{A}{p} = \frac{\left(3.5y + 1.25y^2\right)}{\left(3.5 + 3.217y\right)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From Manning's formula, we have

$$Q = \frac{1}{n} AR^{2/3} S^{1/2}$$

or
$$8 = \frac{1}{0.016}\left[\left(3.5y + 1.25y^2\right) \times \left\{\frac{\left(3.5y + 1.25y^2\right)}{\left(3.5 + 3.217y\right)}\right\}^{2/3} \times \left(\frac{2.6}{10000}\right)^{1/2}\right]$$

or
$$8 = \frac{1}{0.016}\left[\frac{\left(3.5y + 1.25y^2\right)^{5/3}}{\left(3.5 + 3.217y\right)^{2/3}} \times \left(\frac{2.6}{10000}\right)^{1/2}\right]$$

Solving this equation for y by trial, we get

$$y = 1.505 \text{ m}$$

Average shear stress on the channel bed is given by Eq. 15.1 as

$$\tau_0 = wRS$$

$$R = \frac{\left[3.5 \times 1.505 + 1.25(1.505)^2\right]}{\left[3.5 + (3.217 \times 1.505)\right]}$$

$$= \frac{8.09878}{8.34159} = 0.9709$$

∴ Average shear stress on the channel bed

$$\tau_0 = 9810 \times 0.9709 \times \frac{2.6}{10000}$$

$$= 2.476 \text{ N/m}^2$$

Example 15.12. *Water flows in a channel of the shape of isosceles triangle of bed width 'a' and sides making an angle of 45° with the bed. Determine the relations between depth of flow 'd' and bed width 'a' for maximum velocity condition and for maximum discharge condition. Use Manning's formula and note that 'd' is less than 0.5 'a'.*

Solution

From Manning's equation, we have

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

Thus for maximum velocity, we have

$$\frac{dR}{dd} = 0 \text{ ; or } \frac{d(A/P)}{dd} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $$\frac{d(A/P)}{dd} = \frac{P(dA/dd) - A(dP/dd)}{P^2}$$

$$\therefore \quad P(dA/dd) - A\,(dP/dd) = 0$$

In this case

$$A = (a-d)d$$

$$\therefore \quad (dA/dd) = (a-2d)$$

and $$P = \left(a + 2\sqrt{2}d\right)$$

$$\therefore \quad (dP/dd) = 2\sqrt{2}$$

Thus by substitution, we get

$$(a+2\sqrt{2}d)(a-2d) - (a-d)d \times 2\sqrt{2} = 0$$

or $$2\sqrt{2}\,d^2 + 2ad - a^2 = 0$$

or $$d = \frac{-2a \pm \sqrt{4a^2 + 8\sqrt{2}a^2}}{4\sqrt{2}}$$

$$\therefore \quad d = 0.338\,a$$

Discharge Q is given by

$$Q = \frac{1}{n} AR^{2/3} S^{1/2}$$

or $$Q = \frac{1}{n} A \left(\frac{A}{P}\right)^{2/3} S^{1/2}$$

or $$Q = \frac{1}{n}\left(\frac{A^5}{P^2}\right)^{1/3} S^{1/2}$$

Thus for maximum discharge, we have

$$\frac{d(A^5/P^2)}{dd} = 0$$

Since $$\frac{d(A^5/P^2)}{dd} = \frac{P^2 \times 5A^4(dA/dd) - A^5 \times 2P(dP/dd)}{P^4}$$

$$\therefore \quad 5P\,(dA/dd) - 2A\,(dP/dd) = 0$$

Thus by substitution, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$5\left(a+2\sqrt{2}d\right)\ (a-2d)-2(a-d)\ d\times 2\sqrt{2}\ =0$$

or $16\sqrt{2}\,d^2+(10-6\sqrt{2})\,ad-5a^2=0$

or $22.627\,d^2+1.515\,ad-5a^2=0$

or $$d=\frac{-1.515a\pm\sqrt{2.295a^2+452.54a^2}}{45.254}$$

$\therefore$ $$d=0.438\,a$$

Example 15.13. *A trapezoidal channel having a cross–sectional area A_1, wetted perimeter P_1, Manning's coefficient n and laid to a slope S, carries a certain discharge Q_1 at a depth of flow equal to d. To increase the discharge the base width of the channel is widened by x, keeping all other parameters viz., S, d, side slope and n same. Prove that*

$$\left[\frac{Q_2}{Q_1}\right]^3\left[1+\frac{x}{P_1}\right]^2=\left[1+\frac{xd}{A_1}\right]^5$$

where Q_2 is the new discharge in the channel.

Solution

Let b be the initial base width of the channel and the side slopes be z horizontal to 1 vertical. Then

$$A_1=(b+zd)\,d$$

and $$P_1=(b+2d\sqrt{1+z^2})$$

When the base width is increased by x, the new cross-sectional area A_2 and wetted perimeter P_2 are given by

$$A_2=(b+x+zd)\,d=(A_1+xd) \quad ...(i)$$

and $$P_2=(b+x+2d\sqrt{1+z^2})=(P_1+x) \quad ...(ii)$$

Discharge Q_1 is given by

$$Q_1=\frac{1}{n}\,A_1\left(\frac{A_1}{P_1}\right)^{2/3}S^{1/2}$$

or $$Q_1=\frac{1}{n}\frac{A_1^{5/3}}{P_1^{2/3}} \quad ...(iii)$$

Similarly discharge Q_2 is given by

$$Q_2=\frac{1}{n}\,A_2\left(\frac{A_2}{P_2}\right)^{2/3}S^{1/2}$$

or $$Q_2=\frac{1}{n}\frac{A_2^{5/3}}{P_2^{2/3}}S^{1/2} \quad ...(iv)$$

From Eqs (iii) and (iv), we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left[\frac{Q_2}{Q_1}\right]^3 = \left[\frac{A_2}{A_1}\right]^5 \left[\frac{P_1}{P_2}\right]^2$$

By substituting the values of A_2 and P_2 from Eqs (i) and (ii), we get

$$\left[\frac{Q_2}{Q_1}\right]^3 = \left[\frac{A_1 + xd}{A_1}\right]^5 \left[\frac{P_1}{P_1 + x}\right]^2$$

or $$\left[\frac{Q_2}{Q_1}\right]^3 \left[1 + \frac{x}{P_1}\right]^2 = \left[1 + \frac{xd}{A_1}\right]^5$$

Example 15.14. *An earth canal in good condition is 17 m wide at the bottom and has side slope of 2 horizontal to 1 vertical. One side slope extends to a height of 7.8 m above the bottom level and the other side extends to an elevation of 1.8 m, then extends flat to a distance of 150 m and rises vertically. If the slope of the canal is 0.7 m per 1 610 m, estimate the discharge when the depth of water is 2.5 m. Assume Chezy's C = 35.*

Solution

As shown in the above figure the area of flow section of the channel is

$$\begin{aligned} A &= ABK + KBCH + HCDG + GDEF \\ &= \left(\frac{1}{2} \times 5 \times 2.5\right) + (17 \times 2.5) + \left[\left(\frac{2.5 + 0.7}{2}\right) \times 3.6\right] + (150 \ 0.7) \\ &= 6.25 + 42.5 + 5.76 + 105 \\ &= 159.51 \text{ m}^2 \end{aligned}$$

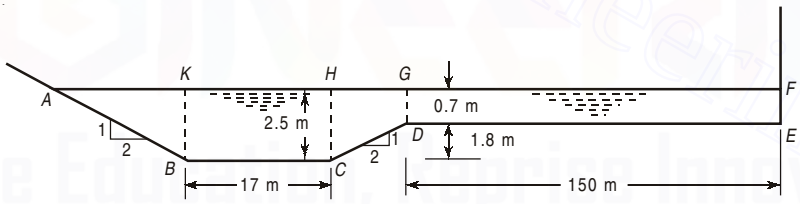


Fig. Ex. 15.14

Wetted perimeter is

$$\begin{aligned} P &= AB + BC + CD + DE + EF \\ &= (2.5\sqrt{5}) + 17 + (1.8\sqrt{5}) + 150 + 0.70 \\ &= 5.59 + 17 + 4.02 + 150 + 0.70 \\ &= 177.31 \text{ m} \end{aligned}$$

Thus $$R = \frac{A}{P} = \frac{159.51}{177.31} = 0.90$$

Using Chezy's formula

$$Q = AC\sqrt{RS}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$S = \frac{0.7}{1610} = \frac{1}{2300} \quad ; C = 35$$

Thus by substitution, we get

$$Q = 159.51 \times 35 \left(0.90 \times \frac{1}{2300}\right)^{1/2}$$

$$= 110.44 \text{ m}^3/\text{s}$$

Example 15.15. *A rectangular channel 5.4 m wide and 1.2 m deep has a slope of 1 in 1000 and is lined with good rubble masonry, for which Manning's n = 0.017. It is desired to increase the discharge to a maximum by changing the channel slope or the form of section. The dimensions of the section may be changed but the channel must contain the same amount of lining. Compute the new dimensions and probable increase in discharge.*

Solution

According to Manning's formula the discharge through the channel with original dimensions is

$$Q_1 = \frac{1}{n} AR^{2/3} S^{1/2}$$

$$A = (5.4 \times 1.2) = 6.48 \text{ m}^2$$

$$P = (5.4 + 2 \times 1.2) = 7.8 \text{ m}$$

$$R = \frac{A}{P} = \frac{6.48}{7.8} = 0.83$$

$$S = \frac{1}{1000}$$

Thus by substitution

$$Q_1 = \frac{1}{0.017} \times 6.48 \times (0.83)^{2/3} \left(\frac{1}{1000}\right)^{½}$$

$$= 10.65 \text{ m}^3/\text{s}$$

Let B and y be the bottom width and the depth of flow respectively for the new section of the channel. In order to have the same amount of lining as in the previous case,

$$P = (B + 2y) = 7.8 \quad \text{... (i)}$$

Further the condition for the discharge to be maximum is

$$B = 2y \quad \text{...(ii)}$$

Solving Eq. (i) and (ii), we have

$$B = 3.9 \text{ m}; y = 1.95 \text{ m}$$

The area of cross-section of the new channel section becomes

$$A = (3.90 \times 1.95) = 7.605 \text{ m}^2$$

$$R = \frac{A}{P} = \frac{7.605}{7.8} = 0.975 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again using Manning's formula the discharge through the new section is

$$Q_2 = \frac{1}{0.017} \times 7.605 \times (0.975)^{2/3}\left(\frac{1}{1000}\right)^{1/2}$$

$$= 13.91 \text{ m}^3/\text{s}$$

The probable increase in the discharge is

$$Q_2 - Q_1 = (13.91 - 10.65) = 3.26 \text{ m}^3/\text{s}$$

and percentage increase in the discharge

$$= \frac{Q_2 - Q_1}{Q_1} \times 100 = \frac{3.26}{10.65} \times 100 = 30.61\%$$

If the original section of the channel is retained then the same increase in the discharge would be obtained if the channel slope is changed to 1 in 586.

Example 15.16. *A trapezoidal channel having bottom width 6 m and side slopes of 2 horizontal to 1 vertical is laid on a bottom slope of 0.0016. If it carries a uniform flow of water at the rate of 10 m^3/s, compute the normal depth and the mean velocity of flow. Take Manning's n as 0.025.*

Solution

Let y be the normal depth of flow.

Then

$$A = (B + zy)\, y = (6 + 2y)y$$

and

$$P = \left(B + 2y\sqrt{1+z^2}\right) = 6 + 2y\sqrt{5}$$

$$R = \frac{A}{P} = \frac{(6+2y)y}{6+2y\sqrt{5}} = \frac{(3+y)y}{3+y\sqrt{5}}$$

Using Manning's formula, we have

$$\frac{10}{(6+2y)y} = \frac{1}{0.025} \times \left[\frac{(3+y)y}{3+y\sqrt{5}}\right]^{2/3} \times (0.0016)^{1/2}$$

or

$$16.57 + 12.35\, y = [(3+y)y]^{5/2}$$

Solving this equation for y by trial, normal depth $y = 0.963$ m

Alternative Method

From Eq. 15.32, conveyance

$$K = \frac{Q}{\sqrt{S}} = \frac{10}{\sqrt{0.0016}} = 250$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

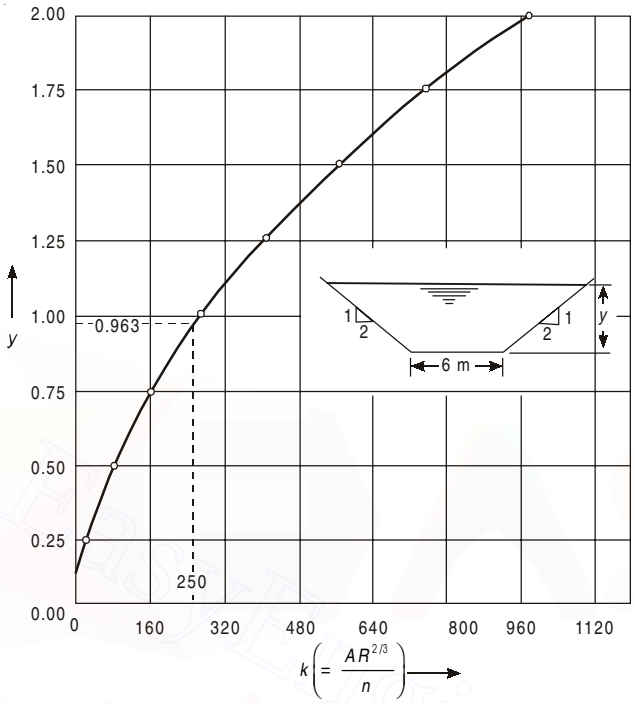


Figure Ex. 15.16

From Eq. 15.31, we have

$$K = \frac{1}{n} AR^{2/3} = \frac{[(6+2y)y]^{5/3}}{0.025(6+2y\sqrt{5})^{2/3}} \quad \text{...(i)}$$

For different values of y the values of conveyance K are computed from Eq. (i) and the same are tabulated below.

y	0	0.25	0.50	0.75	1.00	1.25	1.50	1.75	2.00
K	0	24.28	79.13	160.30	267.42	401.07	562.20	751.96	971.54

A plot of K v/s y is as shown in Fig. Ex. 15.13 from which for $K = 250$, $y = 0.963$ m which is the required normal depth.

The corresponding area $A = 7.62$ m^2, and the mean velocity

$$V = \frac{10}{7 \cdot 62} = 1.31 \text{ m/s}$$

Example 15.17. *A circular sewer 0.6 m inside diameter has a slope of 1 in 400. Find the depth when the discharge is 0.283 m^3/s, taking C in Chezy's formula as 50.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Refer Fig. 15.7; $r = 0.3$ m

$$A = \frac{r^2}{2}(\theta - \sin\theta) = \frac{0.09}{2}(\theta - \sin\theta)$$

$$P = r\theta = 0.3\,\theta$$

$$R = \frac{A}{P} = \frac{r(\theta - \sin\theta)}{2\theta} = \frac{0.3(\theta - \sin\theta)}{2\theta}$$

Using Chezy's formula

$$Q = AC\sqrt{RS}$$

$$Q = 0.283 \text{ m}^3/\text{s};\ C = 50$$

Thus by substitution, we get

$$0.283 = \frac{0.09}{2}(\theta - \sin\theta)\,50\left[\frac{0.3(\theta - \sin\theta)}{2\theta} \times \frac{1}{400}\right]^{1/2}$$

or $$6.495 = \frac{(\theta - \sin\theta)^{3/2}}{\theta^{1/2}}$$

or $$42.19\,\theta = (\theta - \sin\theta)^3$$

Solving the above equation by trial, we get

$$\theta = 286° \text{ (approximately)}$$

Thus depth of flow is

$$y = r + r\cos\left(180° - \frac{\theta}{2}\right)$$

$$= 0.3\left[1 + \cos\left(180° - \frac{286°}{2}\right)\right]$$

$$= 0.3\,[1 + \cos 37°]$$

$$= 0.54 \text{ m}$$

Example 15.18. *A channel has vertical walls 1.2 m apart and a semicircular invert. If the centreline depth is 0.9 m and the bed slope is 1 in 2500, find the discharge using Chezy's formula with C = 54.*

Solution

As shown in the Fig. Ex. 15.15 area of flow section is

$$A = \left[\frac{\pi \times (0.6)^2}{2} + (1.2 \times 0.3)\right] = 0.925 \text{ m}^2$$

Wetted perimeter

$$P = [\pi \times 0.6 + 2 \times 0.3] = 2.485 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus,

$$R = \frac{A}{P} = \frac{0.925}{2.485} = 0.372 \text{ m}$$

Using Chezy's formula

$$Q = AC\sqrt{RS}$$

$$S = \frac{1}{2500}\ ;\ C = 54$$

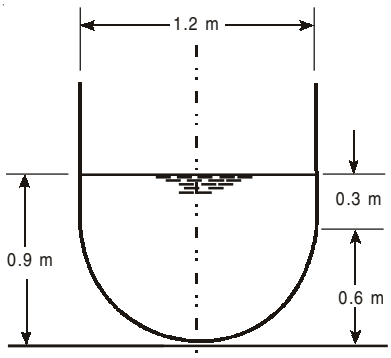


Figure Ex. 15.18

Thus by substitution, we get

$$Q = 0.925 \times 54 \times \sqrt{0.372 \times \frac{1}{2500}}$$

$$= 0.609 \text{ m}^3/\text{s}$$

Example 15.19. *A channel is to be designed to give a constant mean velocity of flow of 1.8 m/s at all depths of flow. The lower portion of the channel to carry the minimum discharge is rectangular and has the best proportion, the bottom width being 1.5 m. Determine the depth of flow when the width of the water surface is 9 m.*

If Manning's n = 0.015, find the channel bed slope.

Solution

The bottom portion is rectangular and has the best proportion, thus

$$B = 2y$$

or $$1.5 = 2y\ ;\ y = 0.75 \text{ m}$$

Also for the best channel section

$$R = \frac{y}{2} = \frac{0.75}{2} = 0.375 \text{ m}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

$$V = 1.8 \text{ m/s}\ ;\ n = 0.015$$

Thus by substitution, we have

$$1.8 = \frac{1}{0.015} \times (0.375)^{2/3} S^{1/2}$$

$$\therefore \quad S = 0.0027\ ;\ \text{or } 1 \text{ in } 370$$

From Eq. 15.27, we have

$$y = R \log_e \left[x + \sqrt{x^2 - R^2} \right] + C$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For $x = \frac{1.5}{2} = 0.75 \text{ m}; y = 0$

$\therefore \quad C = -R \log_e \left[0.75 + \sqrt{(0.75)^2 - R^2}\right]$

Thus $y = R \log_e \left[\frac{x + \sqrt{x^2 - R^2}}{0.75 + \sqrt{(0.75)^2 - R^2}}\right]$

For $x = \frac{9}{2} = 4.5 \text{ m}; R = 0.375 \text{ m}$

Thus by substitution, we get

$$y = (0.375)\log_e \left[\frac{4.5 + \sqrt{(4.5)^2 - (0.375)^2}}{0.75 + \sqrt{(0.75)^2 - (0.375)^2}}\right]$$

or $y = 0.375 \log_e (6.42) = 0.697 \text{ m}$

$\therefore$ Total depth of flow

$= (0.75 + 0.697) = 1.447 \text{ m}$

Example 15.20. *A rectangular channel which is laid on a bottom slope of 0.0064 is to carry 20 m^3/s of water. Determine the width of the channel when the flow is in critical condition. Take Manning's n = 0.015.*

Solution

Let the width of the channel be B.

Then $q = \frac{Q}{B} = \frac{20}{B}$

The critical depth for a rectangular channel section is expressed by Eq. 15.46 as

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left(\frac{400}{B^2 \times 9.81}\right)^{1/3} = \frac{3.442}{B^{2/3}}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

or $$\frac{20}{By_c} = \frac{1}{0.015}\left(\frac{By_c}{B + 2y_c}\right)^{2/3} (0.0064)^{1/2}$$

or $$\frac{20}{B^{1/3}(3.442)} = \frac{0.08}{0.015}\left[\frac{3.442B^{1/3}}{B + (6.884 / B^{2/3})}\right]^{2/3}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\left(\frac{3.75}{3.442}\right)^{3/2} \times \frac{1}{B^{1/2}} = \frac{3.442B}{B^{5/3} + 6.884}$$

or $$0.33\,(B^{5/3} + 6.884) = B^{3/2}$$

Solving this equation for B by trial, we obtain

$$B = 2.41 \text{ m}$$

Example 15.21. *For a constant specific energy of 1.8 N.m /N, calculate the maximum discharge that may occur in a rectangular channel 5.0 m wide.*

Solution

For a given specific energy the discharge is maximum when the flow is in critical state.

Thus the depth of flow $y = y_c$

But from Eq. 15.49

$$y = y_c = \frac{2}{3}E = \frac{2}{3} \times 1.8 = 1.2 \text{ m}$$

Further for a rectangular channel from Eq. 15.46

$$y_c = \left(\frac{q^2}{g}\right)^{1/3}$$

or $$1.2 = \left(\frac{q^2}{g}\right)^{1/3}$$

or $$q = [(1.2)^3 \times 9.81]^{1/2}$$

or $$q = 4.12 \text{ m}^3/\text{s per m}$$

$\therefore$ Total discharge

$$Q = (4.12 \times 5) = 20.6 \text{ m}^3/\text{s}$$

Example 15.22. *A trapezoidal channel has a bottom width of 6 m and side slopes of 2 horizontal to 1 vertical. If the depth of flow is 1.2 m at a discharge of 10 m^3/s, compute the specific energy and the critical depth.*

Solution

$$A = (B + zy)\,y = [(6 + 2 \times 1.2)1.2]$$
$$= 10.08 \text{ m}^2$$

Specific energy is given by

$$E = y + \frac{1}{2g}\left(\frac{Q}{A}\right)^2$$

$$= 1.2 + \frac{1}{2 \times 9.81}\left(\frac{10}{10.08}\right)^2$$

$$= 1.25 \text{ m}$$

For critical flow

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{(10)^2}{9.81} = \frac{[(6+2y_c)y_c]^3}{6+4y_c}$$

Solving the above Equation for y_c by trial, we get

$$y_c = 0.611 \text{ m}$$

Alternative Method

Section factor for critical flow computation is given by

$$Z = \left(\frac{A^3}{T}\right)^{1/2} = \frac{[(6+2y)y]^{3/2}}{(6+4y)^{1/2}} \quad \text{... (i)}$$

For different values of y the values of Z are computed from Eq. (i) and the same are tabulated below.

y	0	0.25	0.50	0.75	1.00	1.25	1.50
Z	0	0.78	2.32	4.45	7.16	10.44	14.32

A plot of Z v/s y is made as shown in the Fig. Ex. 15.22

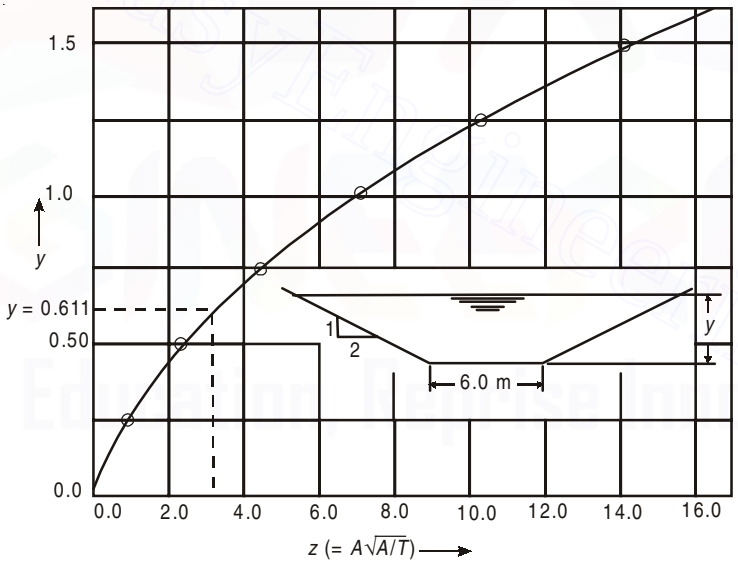


Figure Ex. 15.22

For critical flow from Eq. 15.44, we have

$$\left(\frac{A^3}{T}\right)^{1/2} = Z_c = \frac{Q}{\sqrt{g}} = \frac{10}{\sqrt{9.81}} = 3.19$$

Thus from the plot of Z v/s y, for $Z = 3.19$, $y = 0.611$ m which is the required critical depth.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 15.23. *In a rectangular channel 3.5 m wide laid at a slope of 0.0036, uniform flow occurs at a depth of 2 m. Find how high can the hump be raised without causing afflux? If the upstream depth of flow is to be raised to 2.5 m, what should be the height of the hump? Take Manning's n equal to 0.015.*

Solution

The area of flow section at the upstream section is

$$A = (3.5 \times 2) = 7.0 \text{ m}^2$$

The wetted perimeter is

$$P = 3.5 + (2 \times 2) = 7.5 \text{ m}$$

Thus $$R = \frac{A}{P} = \frac{7.0}{7.5} = 0.93 \text{ m}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

$$= \frac{1}{0.015} \times (0.93)^{2/3} (0.0036)^{1/2}$$

$$= 3.81 \text{ m/s}$$

Discharge $$Q = (7.0 \times 3.81) = 26.67 \text{ m}^3/\text{s}$$

Specific energy at upstream section is

$$E_1 = y_1 + \frac{V_1^2}{2g}$$

$$= 2.0 + \frac{(3.81)^2}{2 \times 9.81} = 2.74 \text{ m}$$

For the rectangular channels the critical depth is given by Eq. 15.46 as

$$y_c = \left(\frac{q^2}{g}\right)^{1/3}$$

$$q = \frac{Q}{B}$$

$$= \frac{26.67}{3.5} = 7.62 \text{ m}^3/\text{s per metre}$$

Thus $$y_c = \left[\frac{(7.62)^2}{9.81}\right]^{1/3} = 1.81 \text{ m}$$

The corresponding minimum specific energy is

$$E_c = \frac{3}{2} y_c = \frac{3}{2} \times 1.81 = 2.715 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From Eq. 15.57 the maximum height of the hump without causing afflux is obtained as

$$\Delta Z_{max} = (E_1 - E_c) = (2.74 - 2.715) = 0.025 \text{ m}$$

If the upstream depth of flow is raised to 2.5 m, the area of flow section becomes

$$A = (3.5 \times 2.5) = 8.75 \text{ m}^2$$

Then for the same discharge the velocity of flow will be

$$V = \frac{26.67}{8.75} = 3.05 \text{ m/s}$$

Specific energy at the upstream section will be

$$E_1 = 2.5 + \frac{(3.05)^2}{2 \times 9.81} = 2.97 \text{ m}$$

$\therefore$ The required height of the hump is

$$\Delta Z = (E_1 - E_c) = (2.97 - 2.715) = 0.255 \text{ m}$$

Example 15.24. *For purpose of discharge measurement the width of a rectangular channel is reduced gradually from 3 m to 2 m and the floor is raised by 0.3 m at a given section. When the approaching depth of flow is 2 m, what rate of flow will be indicated by a drop of 0.15 m in the water surface elevation at the contracted section?*

Solution

From the given data the depth of flow at the contracted section is

$$y = (2.0 - 0.30 - 0.15) = 1.55 \text{ m}$$

If Q is the discharge flowing through the channel then specific energy at the upstream section is

$$E_1 = 2.0 + \frac{Q^2}{2g(3 \times 2)^2} = 2.0 + \frac{Q^2}{72 \times 9.81}$$

Also the specific energy at the contracted section is

$$E_2 = 1.55 + \frac{Q^2}{2g(2 \times 1.55)^2} = 1.55 + \frac{Q^2}{19.22 \times 9.81}$$

Assuming no loss of energy between two sections, we have

$$E_1 = E_2 + \Delta Z$$

Thus $$2.0 + \frac{Q^2}{72 \times 9.81} = 1.55 + \frac{Q^2}{19.22 \times 9.81} + 0.30$$

or $$Q = 6.21 \text{ m}^3/\text{s}$$

Example 15.25. *A trapezoidal channel of base width 6 m and side slopes of 2 horizontal to 1 vertical carries a flow of 60 cumec at a depth of 2.5 m. There is a smooth transition to a rectangular section 6 m wide. (i) If the channel bed is horizontal determine whether the upstream flow is possible as specified; if not determine the upstream depth of flow. (ii) Determine the minimum amount by which the bed must be lowered for the upstream flow to be possible as specified. (iii) Determine the depth of flow in the rectangular section and the change in the water surface level if the transition is accompanied by a gradual lowering of the channel bed by 0.6 m. (iv) Determine the amount by which the bed must be lowered if the drop in water surface is to be restricted to 0.3 m. Assume no losses.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

At the trapezoidal section

$$A_1 = (6 + 2 \times 2.5)\, 2.5 = 27.5 \text{ m}^2$$

$$V_1 = \frac{60}{27.5} = 2.182 \text{ m/s}$$

and

$$E_1 = 2.5 + \frac{(2.182)^2}{2 \times 9.81} = 2.743 \text{ m}$$

(i) The upstream flow as specified would be possible if

$$E_1 \leq E_c$$

where E_c is specific energy at critical flow condition at the rectangular section.

$$E_c = \frac{3}{2} y_c$$

and

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(60/6)^2}{9.81}\right]^{1/3} = 2.168 \text{ m}$$

$$\therefore \quad E_c = \frac{3}{2}(2.168) = 3.252 \text{ m}$$

Since $E_1 < E_c$, the upstream flow as specified would not be possible.

Assuming the upstream flow to be subcritical there will be increase in the depth of flow at which the flow will take place.

Thus if y is the new depth of flow in the trapezoidal section then, we have

$$A_1' = (6 + 2y)y = 2(3 + y)y$$

$$V_1' = \frac{60}{2(3+y)y} = \frac{30}{(3+y)y}$$

and

$$E_1' = y + \frac{1}{2g}\left[\frac{30}{(3+y)y}\right]^2$$

For no loss of energy between the two sections, we have

$$E_1' = E_c$$

or

$$y + \frac{1}{2g}\left[\frac{30}{(3+y)y}\right]^2 = 3.252$$

or

$$y + \frac{900}{2 \times 9.81} \times \frac{1}{(3+y)^2 y^2} = 3.252$$

or

$$y + \frac{450}{9.81} \times \frac{1}{(3+y)^2 y^2} = 3.252$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solving for y by trial and error, we get

$$y = 3.127 \text{ m}$$

Thus in this case there will be heading up of water on the upstream side in the trapezoidal section with new depth of flow being 3.127 m.

(ii) Let x be the minimum amount by which the bed must be lowered for the upstream flow to be possible as specified, i.e., the depth of flow in the trapezoidal section to be 2.5 m as given.

Thus in this case, we have

$$E_1 + x = E_c$$

or $$2.743 + x = 3.252$$

$\therefore$ $$x = 0.509 \text{ m}$$

(iii) Let y be the depth of flow at the rectangular section, then

$$A_2 = 6 \times y$$

$$V_2 = \frac{60}{6 \times y} = \frac{10}{y}$$

and $$E_2 = y + \frac{(10/y)^2}{2 \times 9.81} = \left(y + \frac{5.097}{y^2} \right)$$

For no loss of energy between the two sections, we have

$$E_1 + 0.6 = E_2$$

or $$2.743 + 0.6 = y + \frac{5.097}{y^2}$$

or $$y^3 - 3.343\, y^2 + 5.097 = 0$$

Solving for y by trial and error, we get

$$y = 2.573 \text{ m; or } y = 1.844 \text{ m}$$

These are the alternate depths out of which one being greater than the critical depth will be the depth of subcritical flow, and the other one being less than the critical depth will be the depth of supercritical flow.

Thus if subcritical flow occurs, then

Change in water surface level

$$= (2.5 + 0.6) - 2.573 = 0.527 \text{ m}$$

However, if supercritical flow occurs, then

Change in water surface level

$$= (2.5 + 0.6) - 1.844 = 1.256 \text{ m}$$

(iv) Let the channel bed be required to be lowered by an amount x when the drop in water surface level is to be restricted to 0.3 m, and the depth of flow at the rectangular section be y, then

$$x + 2.5 = y + 0.3$$

or $$y - x = 2.2 \quad \text{... (i)}$$

Again for no loss of energy between the two sections, we have

$$E_1 + x = E_2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$2.743 + x = y + \frac{5.097}{y^2}$$ ...(ii)

Adding Eq. (i) and (ii), we get

$$2.743 = \frac{5.097}{y^2} + 2.2$$

or $y = 3.064$ m

$\therefore$ $x = 0.864$ m

i.e., the channel bed is required to be lowered by 0.864 m.

Again with a lowering of channel bed by 0.864 m, there will be two possible depths of flow at the rectangular section as indicated below.

If y is the depth of flow at the rectangular section, then

$$E_2 = y + \frac{5.097}{y^2}$$

For no loss of energy between the two sections, we have

$$E_1 + 0.864 = E_2$$

or $$2.743 + 0.864 = y + \frac{5.097}{y^2}$$

or $$y^3 - 3.607\, y^2 + 5.097 = 0$$

Solving for y by trial and error, we get

$y = 3.064$ m; or $y = 1.589$ m

These are the alternate depths out of which one being greater than the critical depth will be the depth of subcritical flow, and the other one being less than the critical depth will be the depth of supercritical flow.

Thus if subcritical flow occurs, then

Change in water surface level

$= (2.5 + 0.864) - 3.064 = 0.3$ m

However, if supercritical flow occurs, then

Change in water surface level

$(2.5 + 0.864) - 1.589 = 1.775$ m

Example 15.26. *A trapezoidal channel has a bed width 10 m and side slopes of 2 horizontal to 1 vertical. At a section 200 m downstream the bed rises by 0.08 m gradually, bed width increases to 0.15 m and side slopes become 3 horizontal to 1 vertical. Rise in bed level, increase in bed width and flattening of side slopes are gradual. Flow rate in the channel is 200 m^3/s and the flow depth at the upstream section is 7 m. If Manning's n = 0.035 estimate the depth at the downstream section.*

Solution

At the upstream section

$$A_1 = (10 + 2 \times 7)\, 7 = 168 \text{ m}^2$$

$$P_1 = 10 + 2\sqrt{5} \times 7 = 41.305 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$R_1 = \frac{168}{41.305} = 4.067 \text{ m}$$

$$V_1 = \frac{200}{168} 1.19 \text{ m/s}$$

If E_1 and E_2 are the specific energies at the upstream and the downstream sections and h_L is the head loss between these sections, then

$$E_1 = E_2 + 0.08 + h_L \quad \text{...(i)}$$

$$E_1 = 7 + \frac{(1.19)^2}{2 \times 9.81} = 7.072 \text{ m}$$

Let y be the depth of flow at the downstream section.

Thus $A_2 = (15 + 3y)\, y$

$$P_2 = 15 + 2\sqrt{10}\, y$$

$$V_2 = \frac{200}{(15+3y)y}$$

$$\therefore \quad E_2 = y + \frac{(200)^2}{(15+3y)^2 y^2 \times 2 \times 9.81}$$

Further if S_{f1} is the slope of the energy line at the upstream section, then

$$S_{f1} = \left(\frac{V_1 n}{R_1^{2/3}}\right)^2$$

$$= \left[\frac{1.19 \times 0.035}{(4.067)^{2/3}}\right]^2 = 2.67 \times 10^{-4}$$

Assuming the energy line slope to be the same at both the sections, we have

$$\frac{h_L}{200} = 2.67 \times 10^{-4}$$

$$\therefore \quad h_L = 0.053\,4 \text{ m}$$

By substituting in Eq. (i), we get

$$7.072 = y + \frac{(200)^2}{(15+3y)^2 y^2 \times 2 \times 9.81} + 0.08 + 0.0534$$

or $$6.938\,6 = y + \frac{2038.736}{(15+3y)^2 y^2}$$

Solving for y by trial and error, we get

$$y = 6.905 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $A_2 = (15 + 3 \times 6.905)\, 6.905 = 246.61 \text{ m}^2$

$$P_2 = \left(15 + 2\sqrt{10} \times 6.905\right) = 58.67 \text{ m}$$

$$R_2 = \frac{246.61}{58.67} = 4.203 \text{ m}$$

$$V_2 = \frac{200}{246.61} = 0.811 \text{ m/s}$$

If S_{f2} is the slope of the energy line at the downstream section, then

$$S_{f2} = \left(\frac{V_2 n}{R_2^{2/3}}\right)^2$$

$$= \left[\frac{0.811 \times 0.035}{(4.203)^{2/3}}\right]^2 = 1.19 \times 10^{-4}$$

Thus mean value of slope S_f of the energy line is

$$S_f = \frac{S_{f_1} + S_{f_2}}{2}$$

$$= \frac{(2.67 + 1.19)10^{-4}}{2}$$

$$= 1.93 \times 10^{-4}$$

Thus $\frac{h_L}{200} = 1.93 \times 10^{-4}$

$\therefore$ $h_L = 0.0386 \text{ m}$

Again by substituting in Eq. (i), we get

$$7.072 = y + \frac{(200)^2}{(15 + 3y)^2 y^2 \times 2 \times 9.81} + 0.08 + 0.0386$$

or $$6.953\ 4 = y + \frac{2038.736}{(15 + 3y)^2 y^2}$$

Solving for y by trial and error, we get

$$y = 6.92 \text{ m}$$

Example 15.27 *Calculate the possible depths of flow at which a discharge of 26.67 cumec may be carried in a rectangular channel 3.5 m wide with a specific energy equal to 2.74 m.*

Solution

Let y be the depth of flow at which the given discharge may be carried with the given special energy in the given channel.

From Eq. 15.35, specific energy is given as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$E = y + \frac{Q^2}{2gA^2}$$

or $$E = y + \frac{Q^2}{2g(By)^2}$$

By substituting the given values, we get

$$2.74 = y + \frac{(26.67)^2}{2 \times 9.81 \times (3.5 \times y)^2}$$

or $$y^3 - 2.74y^2 + 2.96 = 0$$

Solving for y by trial and error, we obtain the following three values of y, viz.,

$$y = 2.00 \text{ m}, y = 1.64 \text{ m and } y = -0.90 \text{ m}$$

The third value of y being negative has no practical significance and hence it may be ignored.

Therefore the two possible depths of flow for the given discharge and given specific energy in the given rectangular channel are 1.64 m and 2.00 m, which are known as alternate depths.

Note: It may be noted that for a given discharge carried in a given channel with a given specific energy, Eq. 15.35 reduces to a cubic equation for the depth of flow y. The solution of this cubic equation gives three values of y, out of which the two values of y are always positive and the third value is negative. It therefore indicates that for given discharge a plot of specific energy, E v/s depth of flow, y has three limbs, the two of which are as shown in the ordinary specific energy curve (Fig. 15.9), corresponding to the positive values of y. The third limb corresponds to the negative values of y which is normally not considered because it has no practical significance.

Example 15.28. *Design a concrete lined channel to carry a discharge of 500 m^3/s at a slope of 1 in 4000. The side slopes of the channel may be taken as 1:1. The Manning's roughness coefficient for the lining, is 0.014. Assume permissible velocity in the section as 2.5 m/s.*

Solution

For a lined channel section as shown in Fig. 15.20, we have

$$A = By + y^2(\theta + \cot\theta)$$

and $$P = B + 2y(\theta + \cot\theta)$$

$$\theta = 45° = \frac{\pi}{4}$$

$$Q = 500 \text{ m}^3/\text{s}; V = 2.5 \text{ m/s}$$

$$\therefore \quad A = \frac{Q}{V} = \frac{500}{2.5} = 200 \text{ m}^2$$

Thus $$200 = By + y^2\left(\frac{\pi}{4} + 1\right) \quad \text{...(i)}$$

From Manning's equation, we have

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\text{or} \qquad 2.5 = \frac{1}{0.014} \times R^{2/3} \times \left(\frac{1}{4000}\right)^{1/2}$$

$$\therefore \qquad R = 3.293 \text{ m}$$

$$\text{Also} \qquad R = \frac{A}{P} = \frac{200}{P}$$

$$\therefore \qquad P = \frac{200}{3.293} = 60.735 \text{ m}$$

$$\text{Thus} \qquad 60.735 = B + 2y\left(\frac{\pi}{4} + 1\right) \qquad \text{... (ii)}$$

From Eq. (i) and (ii), we have

$$\left(\frac{\pi}{4} + 1\right) y^2 - 60.735\, y + 200 = 0$$

$$\text{or} \qquad y^2 - 34.018\, y + 112.02 = 0$$

$$\therefore \qquad y = 3.694 \text{ m}$$

From Eq. (ii), we have

$$B = 60.735 - 2y\left(\frac{\pi}{4} + 1\right)$$

$$\text{or} \qquad B = 60.735 - 2 \times 3.694\left(\frac{\pi}{4} + 1\right)$$

$$\therefore \qquad B = 47.544 \text{ m}$$

Note. In this case if a trapezoidal channel section with sharp corners is designed then a very wide and shallow section is obtained which is not a practical section.

Example 15.29. *Water flows at a velocity of 1 m/s and a depth of 2 m in an open channel of rectangular cross-section 3 m wide. At a certain section the width is reduced to 1.8 m and the bed is raised by 0.65 m. Will the upstream depth be affected? If so, to what extent?*

Solution

The area of flow section at the upstream section is

$$A = (3 \times 2) = 6 \text{ m}^2$$

and velocity of flow is

$$V = 1 \text{ m/s}$$

$$\therefore \text{ Discharge} \qquad Q = (6 \times 1) = 6 \text{ m}^3/\text{s}$$

Specific energy at the upstream section is

$$E_1 = y_1 + \frac{V_1^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 2.0 + \frac{(1)^2}{2 \times 9.81} = 2.051 \text{ m}$$

The upstream depth will not be affected if

$$E_1 \geq E_c + 0.65$$

where E_c is specific energy at critical flow condition at the contracted section.

$$E_c = \frac{3}{2} y_c$$

and $$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(6/1.8)^2}{9.81}\right]^{1/3} = 1.042 \text{ m}$$

$$\therefore \quad E_c = \frac{3}{2} \times 1.042 = 1.563 \text{ m}$$

and $$E_c + 0.65 = (1.563 + 0.65) = 2.213 \text{ m}$$

Since $E_1 < E_c + 0.65$, the upstream depth will be affected and there will be heading up of water at the upstream section to such an extent that the flow at the contracted section will be in a critical state.

Let y be the new depth of flow at the upstream section, then

$$V = \frac{6}{3 \times y} = \frac{2}{y}$$

and $$E_1 = y + \frac{(2/y)^2}{2 \times 9.81} = y + \frac{0.204}{y^2}$$

Specific energy at the contracted section is

$$E_2 = E_c + 0.65$$
$$= (1.563 + 0.65) = 2.213 \text{ m}$$

For no loss of energy between the two sections, we have

$$E_1 = E_2$$

or $$y + \frac{0.204}{y^2} = 2.213$$

Solving for y by trial and error, we get

$$y = 2.17 \text{ m}$$

i.e., the depth of flow at the upstream section will increase by 0.17 m.

Example 15.30. *An unlined irrigation canal has its bed and side composed of cohesionless material having mean diameter 6 mm. Angle of repose of the material is 40°. The bed width of the canal is 5 m and the side slope 1.5(H) : 1(V). Determine the minimum discharge that can be admitted into the canal without any sediment movement. Longitudinal slope of the canal is 1 in 5000, Manning's n = 0.025 and specific gravity of the material s = 2.65.*

Solution

Angle of repose $\phi = 40°$; $\tan 40° = 0.8391$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Side slope $= 1.5(H) : 1(V)$

$\therefore \quad \tan\theta = \dfrac{1}{1.5} = 0.6667 \; ; \cos\theta = \dfrac{3}{\sqrt{13}} = 0.8321$

The ratio of unit tractive force τ_s on the side of the channel and unit tractive force τ_L on the level surface is given by

$$K = \frac{\tau_s}{\tau_L} = \cos\theta \sqrt{1 - \frac{\tan^2\theta}{\tan^2\phi}}$$

Thus by substitution we get

$$K = 0.8321 \sqrt{1 - \frac{(0.6667)^2}{(0.8391)^2}} = 0.5053$$

$\therefore \quad \tau_s = 0.5053 \tau_L$

According to Shield's formula critical unit tractive force τ_{cr} for just no movement of the sediment is given as

$$\tau_{cr} = 0.06\, w(s-1)d$$

$$= 0.06 \times 9.81(2.65 - 1) \times \frac{6}{1000}$$

$$= 0.0058 \text{ kN/m}^2$$

$\therefore \quad \tau_L = \tau_{cr} = 0.0058 \text{ kN/m}^2$

$\therefore \quad \tau_s = 0.5053 \times 0.0058 \text{ kN/m}^2$

But $\tau_s = 0.75\, wRS_0 = 0.75 \times 9.81 \times R \times \dfrac{1}{5000}$

$\therefore \quad 0.75 \times 9.81 \times R \times \dfrac{1}{5000} = 0.5053 \times 0.0058$

or $R = 2 \text{ m}$

$A = (5 + 1.5y)\, y$

$P = 5 + \sqrt{13y}$

$\therefore \quad 2 = \dfrac{(5 + 1.5y)\, y}{5 + \sqrt{13y}}$

or $1.5y^2 - 2.21y - 10 = 0$

$y = 3.422 \text{ m}$

$$Q = \frac{1}{n} A R^{2/3} S_0^{1/2}$$

$$= \frac{1}{0.025} (5 + 1.5 \times 3.422)\, 3.422 \times (2)^{2/3} \times \frac{1}{5000}$$

$$= 0.44 \text{ m}^3/\text{s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. A channel is a flow passage through which water flows under atmospheric pressure, and hence in channels the flow of water takes place with a free surface which is subjected to atmospheric pressure.
2. In order to overcome the resistance and to cause the flow of water in a channel, it is constructed with its bottom slopping towards the direction of flow.
3. A channel may be classified as natural channel and artifical channel. A *natural channel* is the one which has irregular sections of varying shapes, which is developed in a natural way. The examples of natural channels are rivers, streams, etc. An *artificial channel* is the one which is built artificially for carrying water for various purposes. The artificial channels may be further classified according to the shape of the cross-section as *rectangular channel, trapezoidal channel, triangular channel, parabolic channel* and *circular channel*.

   A channel having the same shape of various sections along its length and laid on a constant bottom slope is known as *prismatic channel*, otherwise the channel is *non-prismatic*.
4. The flow is channels may be classified as *uniform flow* and *non-uniform* (*or varied*) flow.

   The flow in a channel is said to be uniform if the depth, slope, cross-section and velocity remain constant over a given length of the channel. Thus in a prismatic channel the flow will be uniform if only the depth of flow y is same at every section of the channel, that is $(\partial y/\partial w) = 0$.

   Flow in channels is termed as non-uniform or varied if the depth of flow y, changes from section to section along the length of the channel, that is $(\partial y/\partial s)$ is not equal to zero.

   Varied flow may be further classified as rapidly varied flow (R.V.F.) and gradually varied flow (G.V.F.). If the depth of flow changes abruptly over a comparatively short distance, the flow is characterized as a rapidly varied flow. In a gradually varied flow the change in the depth of flow takes place gradually in a long reach of channel.
5. The flow in channels may also be characterized as laminar, turbulent or in a transitional state on the basis of Reynolds number Re which in the case of channels is defined as

$$\text{Re} = \frac{\rho V R}{\mu}$$

where

ρ = mass density of water;
V = mean velocity of flow;
R = hydraulic radius or hydraulic mean depth of the channel section $\left(= \frac{A}{P}\right)$; and
μ = absolute viscosity of water.

The flow in channels may be considered to laminar if Re is less than 500 and for R*e* greater than 2000 the flow in channels is turbulent. For Re in between 500 to 2000 the flow in channels may be considered to be in a transitional state.

6. The mean velocity of flow in channels is given by Chezy's formula as

$$V = C\sqrt{RS} \quad \text{or } V = C\sqrt{mi}$$

where

V = mean velocity of flow;
R or m = hydraulic radius or hydraulic mean depth of the channels section $\left(= \frac{A}{P}\right)$;
S or i = slope of the channel bottom; and
C = Chezy's coefficient.

7. The value of Chezy's coefficient is given by empirical formula as

   (i) Gangulliet–Kutter's formula

$$C = \frac{23 + \dfrac{0.00155}{S} + \dfrac{1}{n}}{1 + \left(23 + \dfrac{0.00155}{S}\right)\dfrac{n}{\sqrt{R}}}$$

where

n = Kutter's roughness coefficient.

   (ii) Bazin's formula

$$C = \frac{157.6}{1.81 + \dfrac{m}{\sqrt{R}}}$$

where

m = Bazin's roughness coefficient

8. The mean velocity of flow in channels is given by Manning's formula as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

where n = Manning's roughness coefficient.

The values of Manning's n and Kutter's n are approximately the same.

A comparison of Manning's formula with Chezy's formula indicates that

$$C = \frac{1}{n} R^{1/6}$$

9. A channel section is considered to be the most economical or most efficient when it can pass maximum discharge for given cross-sectional area, resistance coefficient and bottom slope.

The conditions for the most economical or most efficient channel section are

(i) For rectangular channel section

$$B = 2y$$

and $$R = \frac{y}{2}$$

where B =Bed width of the channel; and

y = depth of flow.

(ii) For trapezoidal channel section

(a) When side slope z is constant

$$\frac{B + 2zy}{2} = y\sqrt{1 + z^2}$$

and $$R = \frac{y}{2}$$

Also a semi-circle drawn from the mid-point of the top width as centre and radius equal to the depth of flow will touch the three sides of a most economical or most efficient trapezoidal channel section, viz., the bottom and the two slopping sides tangentially

(b) When the bottom width B is constant

$$\frac{B}{y} = \frac{1 - z^2}{z}$$

(c) When the depth of flow is constant

$$z = \frac{1}{\sqrt{3}}; \text{ or } \theta = 60°$$

where

θ = angle made by the sloping size with the horizontal

(iii) For triangular channel section

$q = 45°$; or $z = 1$

and $$R = \frac{y}{2\sqrt{2}}$$

where

θ = angle of inclination of each of the slopping sides with the vertical.

(iv) For circular channel section, area cannot be maintained constant and hence there are two different conditions, one for maximum velocity and the other for maximum discharge.

(a) For maximum velocity through a circular channel section

$$y = 0.81\, D$$

and $$R = 0.30\, D$$

(b) For maximum discharge through a circular channel section

$$y = 0.95D$$

and $$R = 0.29D$$

where

D = diameter of the circular channel section.

For a circular channel section

Area of flow section $A = \frac{r^2}{2}(\theta - \sin\theta)$; and

Wetted perimetes $P = r\theta$

where

r = radius of channel section; and

θ = angle subtended at the centre by the portion of the circular arc in contact with water.

10. The discharge Q of uniform flow in a channel may be expressed as

$$Q = K\sqrt{S}$$

where

K = conveyance of the channel section.

The value of K is given as

(i) When Chezy's formula is used

$$K = CA\sqrt{R}$$

(ii) When Manning's formula is used

$$K = \frac{1}{n} AR^{2/3}$$

11. In channels total energy or head per unit weight of flowing water at any channel section is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{V^2}{2g} + y + Z$$

12. The specific energy of flow at any channel section is defined as the energy per unit weight of flowing water measured with respect to the channel bottom as the datum. Thus specific energy E at any channel section is given by

$$E = y + \frac{V^2}{2g} = y + \frac{Q^2}{2gA^2}$$

For a given channel section and discharge Q, the specific energy may be plotted against the depth of flow to obtain specific energy curve. It may be seen from this curve that there are two possible depths for the same specific energy. These two depths for given specific energy are known as *alternate depths*.

13. The depth of flow at which the specific energy is minimum is called *critical depth* y_c.
14. The velocity of flow at critical depth is known as *critical velocity* V_c.
15. The flow in a channel with depth of flow greater than critical depth is known as *subcritical flow* or *tranquil flow*.
17. The flow in a channel with depth of flow less than critical depth is known as *supercritical flow* or *rapid flow*.
18. For critical flow Froude number $Fr = 1$; for subcritical flow $Fr < 1$; and for supercritical flow $Fr > 1$. The Froude number Fr is defined as

$$Fr = \frac{V}{\sqrt{gD}}$$

where

D = hydraulic depth = $\frac{A}{T}$; and

T = top width

19. For a rectangular channel the critical depth y_c is given as

(i) $$y_c = \left(\frac{q^2}{g}\right)^{1/3}$$

where

q = discharge per unit width of the channel section

(ii) $$y_c = \frac{2}{3}E$$

20. The specific force F is given by

$$F = \frac{Q^2}{gA} + A\bar{z}$$

where

$\bar{z}$ = vertical depth of centroid of the area of flow section below the free surface.

For a given channel section and discharge Q, the specific force may also be plotted against the depth of flow to obtain specific force curve. It may be seen from this curve that there are two possible depths for the same specific force. These two depths for the given specific force are known as *conjugate* depths.

21. For a given channel section and discharge, the specific force is also minimum at critical depth of flow.
22. The section factor for critical flow computation Z is given by

$$Z = \left(\frac{A^3}{T}\right)^{1/2} = A\sqrt{D}$$

PROBLEMS

15.1 What do you understand by (a) steady and unsteady flow; (b) uniform and non-uniform flow in the case of channels?

15.2 What is Chezy's formula? How is it derived?

Show that Chezy's coefficient $C = \left(\frac{R^{1/6}}{n}\right)$, where

R is the hydraulic radius and n is Manning's roughness coefficient.

15.3 Explain the significance of channels of most efficient section.

15.4 Show that for a trapezoidal channel of given area of flow, the condition of maximum flow requires that hydraulic mean depth is equal to one half the depth of flow.

15.5 State the conditions under which the rectangular section of an open channel will be most economical. Derive these conditions.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

15.6 Show that the discharge formula for a trapezoidal channel, having Manning's coefficient $n = 0.0126$ and carrying maximum flow is given by

$$Q = 100\, y^{8/3}\left(\sqrt{1+z^2} - \frac{z}{2}\right) S_0^{1/2}$$

where Q is discharge in m^3/s; y is depth of flow in m in the channel of side slopes 1 vertical to z horizontal and S_0 is the bed slope of the channel.

15.7 An open channel is V-shaped, each side being inclined at 45° to the vertical. If it carries a discharge of 0.04 m^3/s, when the depth of flow at the centre is 225 mm, calculate the slope of the channel assuming that Chezy's $C = 50$.

[*Ans*. 0.00314]

15.8 An open channel is to be constructed of trapezoidal section and with side slopes 1 vertical to $1\frac{1}{2}$ horizontal. Find the proportions (i.e., relation between bottom width and depth of flow) for minimum excavation (i.e., best hydraulic section).

If the flow is to be 2.7 cumec, calculate the bottom width and the depth of flow assuming C in Chezy's formula as 44.5 and the bed slope is 1 in 4000. [*Ans*. 0.606; 0.885 m; 1.46 m]

15.9 Comment on the statement: "The maximum discharge from a circular cross-section not under pressure, occurs at a little less than full depth''.

15.10 (a) Explain the terms "cross-section of greatest efficiency" and "critical velocity" for flow in an open channel. (b) Discuss the factors that influence the adoption of safe permissible velocities in open earthen channels. (c) It is required to excavate a canal of rectangular section out of rock to bring 15 m^3 of water per second from a distance of 6.4 km with a velocity of 2.25 m/s. Determine the most suitable section for the canal and its gradient. Take $n = 0.02$.

[*Ans*. 3.65 m; 1.825 m; 0.002 29]

15.11 Determine the most efficient section of a trapezoidal channel with side slopes 1 vertical to 2 horizontal, carrying a discharge of 11.25 m^3/s with a velocity of 0.75 m/s. What should be the bed slope of the channel ? Take Manning's $n = 0.025$.

[*Ans*. 2.46 m; 1.16 m; 0.000 27]

15.12 A canal is to have a trapezoidal section with one side vertical and the other sloping at 45 degrees. It has to carry 30 m^3 of water per second with a mean velocity of 1 m/s. Compute the dimensions of the section which will require the minimum lining.

[*Ans*. $y = 3.96$ m; $B = 5.60$ m]

15.13 An open channel laid at a constant slope is required to carry a maximum discharge of 5 m^3/s and a minimum discharge of 1 m^3/s at a constant velocity of 1 m/s at all depths of flow. Compute graphically or otherwise the top width at the free surface and the depths of flow corresponding to minimum and maximum discharges. For minimum discharge a rectangular channel section of the most economical type may be designed.

[*Ans*. 12.56 m; 0.707 m; 1.505 m]

15.14 A trapezoidal channel with side slopes of 2 horizontal to 1 vertical has to carry a discharge of 20 m^3/s. If the bottom width is 4 m, calculate the bottom slope required to maintain a uniform flow at a depth of 1.5 m. Take Manning's $n = 0.015$.What would be the normal depth of flow for the above channel to carry a discharge of 27 m^3/s?

[*Ans*. 1 in 1193; 1.745 m]

15.15 Show that relation between the alternate depths y_1 and y_2 in a rectangular channel can be expressed by

$$\frac{2y_1^2 y_2^2}{(y_1 + y_2)} = y_c^3$$

15.16 (a) Explain the energy variation in a short hydraulic jump using the specific energy equation.

(b) A uniform flow of 9 m^3/s occurs at a depth of 1.5 m in a long rectangular channel 3 m wide. Compute the minimum height of a flat top hump that can be built on the channel floor in order to produce a critical depth.

[*Ans*. 246 mm]

15.17 A trapezoidal channel having bottom width 5 m and side slopes 1:1 carries a discharge of 12 m^3/s. Compute the critical depth and the critical velocity. If Manning's $n = 0.02$ determine the bottom slope required to maintain the critical depth. [*Ans*. 0.793 m; 2.61 m/s; 1 in 200]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

15.18 For a constant energy of 2.4 joules per newton (or 2.4 newton metre per newton) calculate the maximum discharge that may occur in a rectangular channel 4 m wide.
[*Ans*. 25.36 m^3/s]

15.19 A rectangular channel 7.5 m wide carries 12 m^3 of water per second with a velocity of 1.5 m/s. Compute the specific energy. Also find the depth of flow in the channel when the specific energy would be minimum. What will be the value of critical velocity as well as minimum specific energy ?
[*Ans*. 1.18 J/N ; 0.639 m; 2.503 m/s; 0.959 J/N]

15.20 Uniform flow occurs at a depth of 1.5 m in a long rectangular channel 3 m wide and laid at a slope of 0.0009. If Manning's n is given as 0.015, calculate the width of contraction which will produce critical depth without increasing the upstream depth of flow. [*Ans*. 2.08 m]

15.21 A rectangular channel is reduced gradually from 2 m to 1.5 m and the floor is raised by 0.25 m at a given section. When the approaching depth of flow is 1.5 m, what rate of flow will be indicated if the flow at the contracted section is at critical depth. [*Ans*. 3.95 m^3/s]

15.22 Define 'critical tractive stress' and state the main factors on which it depends. A wide unlined channel carrying silt free water has a depth of 2.0 m. The maximum tractive stress permissible on the bed to prevent scour is 1.962 N/m^2. What is the maximum slope that can be given to the channel?
[**Hint**. For 'critical tractive stress' refer Chapter 28. However, for this problem tractive stress may be taken same as boundary shear stress $\tau_0 = wRS_0 = wyS_0$]. [*Ans*. 1 in 10000]

15.23 A rectangular channel 2.4 m wide carries uniform flow of 7 cumec at a depth of 1.5 m. If there is a local rise of 150 mm in the bed level, calculate the change of water elevation. What can be the maximum rise in the bed elevation such that the upstream depth is not affected?
[*Ans*. 73.3 mm; 262 mm]

15.24 A trapezoidal channel with a base width of 6 m and side slopes of 2 hor. to 1 ver. conveys water at 17 m^3/s with a depth of 1.5 m. Is the flow situation sub or super critical?
[*Ans*. $Fr = 0.38$, hence flow is subcritical]

15.25 In a wide rectanguler channel if the normal depth is increased by 20%, determine the corresponding percentage increase in the discharge. Use Manning's equation.
[*Ans*. 35.5%]

15.26 A 3.6 m wide rectangular channel carries water to a depth of 1.8 m. In order to measure the discharge, the channel width is reduced to 2.4 m and a hump of 0.3 m height is provided in the bottom. Calculate the discharge if water surface in the contracted section drops by 0.15 m. Assume no losses. [*Ans*. 6.42 m^3/s]

15.27 A trapezoidal channel has a bottom width of 6 m and side slopes of 1:1. The depth of flow is 1.5 m at a discharge of 15 m^3/s. Determine the specific energy. If the critical depth is 0.9 m, discuss the type of flow corresponding to the critical depth.
[*Ans*. 1.591 m; Flow is in critical state with min. specific energy = 1.197 m]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Non-uniform Flow in Channels

Chapter 16

16.1 INTRODUCTION

As explained in Chapter 15 the non-uniform or varied flow in a channel is the one in which depth of flow changes from section to section along the length of the channel. It may be further classified as *gradually varied flow* (G.V.F.) and *rapidly varied flow* (R.V.F.). The gradually varied flow is a steady non-uniform flow in which the depth of flow varies gradually. Many cases of gradually varied flow are of practical interest to engineers such as flow upstream of a weir or a dam, flow downstream of a sluice gate, flow in channels with break in bottom slopes etc., wherein study of back water and the location of hydraulic jump is of major importance. In a rapidly varied flow the depth of flow changes abruptly over a comparatively short distance. Typical examples of rapidly varied flow are hydraulic jump and hydraulic drop. In this chapter both the types of non-uniform flows have been discussed.

16.2 GRADUALLY VARIED FLOW

The problem of gradually varied flow is that of predicting overall flow pattern, or in other words prediction of the water surface profile to be expected in a given channel with given steady discharge. Such problems can be solved by writing the differential equation for the water surface profile and then integrating it.

1. Dynamic Equation of Gradually Varied Flow. The dynamic equation for gradually varied flow can be derived from the basic energy equation with the following assumptions:

(a) The uniform flow formulae (such as Manning's or Chezy's) may be used to evaluate the energy slope of a gradually varied flow and the corresponding coefficients of roughness developed primarily for uniform flow are applicable to the gradually varied flow also.

Thus, $$(S_f)_{\text{G.V.F.}} = \left(\frac{Vn}{R^{2/3}}\right)^2 \quad \text{...(Manning's)}$$

$$(Sf)_{\text{G.V.F.}} = \left(\frac{V}{C\sqrt{R}}\right)^2 \quad \text{...(Chezy's)}$$

(b) The bottom slope of the channel is very small.

(c) The channel is prismatic.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(d) The energy correction factor α is unity.

(e) The pressure distribution in any vertical is hydrostatic.

(f) The roughness coefficient is independent of the depth of flow and it is constant throughout the channel reach considered.

Considering a short reach of channel having gradually varied flow as shown in Fig. 16.1, the energy equation at any section may be written as

$$H = \frac{V^2}{2g} + y + z$$

or

$$H = \frac{Q^2}{2gA^2} + y + z \qquad \left(\text{since} V = \frac{Q}{A}\right)$$

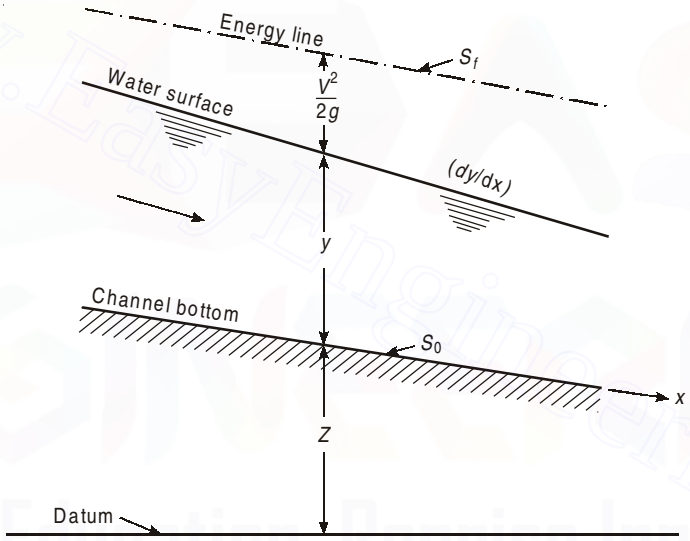


Figure 16.1 *Gradually varied flow*

Differentiating each term of the above equation with respect to x, where x is measured along the channel bottom, the following differential equation can be obtained

$$\frac{dH}{dx} = \frac{d}{dx}\left(\frac{Q^2}{2gA^2}\right) + \frac{dy}{dx} + \frac{dz}{dx}$$

or

$$\frac{dH}{dx} = -\frac{Q^2}{gA^3}\frac{dA}{dx} + \frac{dy}{dx} + \frac{dz}{dx}$$

In the above differential equation, $\frac{dH}{dx}$ is the slope of the energy line and hence $\frac{dH}{dx} = -S_f$;

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\frac{dz}{dx}$ is the slope of the channel bed and hence $\frac{dz}{dx} = -S_0$ (the negative signs for S_f and S_0 indicate that as x increases H and z decreases ; and $\frac{dy}{dx}$ is the slope of the water surface with respect to the channel bottom. Further the term $\frac{dA}{dx}$ can be written as $\frac{dA}{dx} = \frac{dA}{dy} \times \frac{dy}{dx} = T\frac{dy}{dx}$, since $\frac{dA}{dy}$ is equal to the surface width T.

The substitution of these terms in the above differential equation yields

$$-S_f = -\frac{Q^2T}{gA^3}\frac{dy}{dx} + \frac{dy}{dx} - S_0$$

Solving for $\frac{dy}{dx}$, the following differential equation for the water surface slope can be obtained,

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2T}{gA^3}} = \frac{S_0 - S_f}{1 - Fr^2} \qquad ...(16.1)$$

Since $\frac{Q^2T}{gA^3} = Fr^2$

Equation 16.1 is the basic differential equation for the gradually varied flow. It may be observed from Eq. 16.1 that when $\left(\frac{dy}{dx}\right) = 0$; $S_0 = S_f$ and the water surface is parallel to channel bottom thus representing a uniform flow as explained in Chapter 15. When $\left(\frac{dy}{dx}\right)$ is positive the water surface is rising and when $\left(\frac{dy}{dx}\right)$ is negative the water surface is falling.

2. Alternative Derivation for Gradually Varied Flow Equation. Equation 16.1 can also be derived by considering the total energy (or total head) at sections 1 and 2, dx apart along the bottom of the channel. Thus

$$z_1 + y_1 + \frac{V_1^2}{2g} = z_2 + y_2 + \frac{V_2^2}{2g} + S_f(dx)$$

or $$(z_1 - z_2) - S_f\,(dx) = (y_2 - y_1) + \frac{V_2^2}{2g} - \frac{V_1^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$S_0(dx) - S_f\,(dx) = dy + d\left(\frac{V^2}{2g}\right)$$

or $$S_0 - S_f = \frac{dy}{dx} + \frac{d}{dx}\left(\frac{V^2}{2g}\right)$$

or $$S_0 - S_f = \frac{dy}{dx} + \frac{d}{dy}\left(\frac{V^2}{2g}\right)\frac{dy}{dx}$$

or $$\frac{dy}{dx} = \frac{S_0 - S_f}{1 + \frac{d}{dy}\left(\frac{V^2}{2g}\right)}$$

In the above equation the term $\frac{d}{dy}\left(\frac{V^2}{2g}\right)$ represents the change in velocity head which may also be expressed as

$$\frac{d}{dy}\left(\frac{V^2}{2g}\right) = \frac{d}{dy}\left(\frac{Q^2}{2gA^2}\right) = \frac{Q^2}{gA^3}\left(\frac{dA}{dy}\right) = -\frac{Q^2T}{gA^3}$$

Thus by substitution the value of $\frac{d}{dy}\left(\frac{v^2}{2g}\right)$ in the above equation, it becomes

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2T}{gA^3}}$$

which is same as Eq. 16.1 derived earlier.

Equation 16.1 may also be derived by adopting another approach which is based on the concept of the specific energy as described below. For a given discharge Q flowing in a channel the specific energy at a channel section may be expressed as

$$E = y + \frac{Q^2}{2gA^2}$$

Obviously E is function of the depth of flow. However, in uniform flow since the depth of flow remains constant from section to section, $(dE/dx) = 0$. On the other hand in varied flow since the depth of flow varies from section to section, the specific energy will also vary. Thus differentiating both the sides of the above noted expression for E with respect to the direction of flow x, we have

$$\frac{dE}{dx} = \frac{dy}{dx} + \frac{Q^2}{2g}\left(\frac{2}{A^3}\right)\frac{dA}{dx}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{dE}{dx} = \frac{dy}{dx} - \frac{Q^2}{gA^3}\left(\frac{dA}{dy}\times\frac{dy}{dx}\right)$$

or $$\frac{dE}{dx} = \frac{dy}{dx}\left(1-\frac{Q^2T}{gA^3}\right)$$

The above expression represents the change of specific energy in a small reach dx of a channel having the nonuniform flow. Further if S_0 is the bottom slope of the channel then $(S_0\,dx)$ is the work done by gravity in a small reach dx. Similarly if S_f is the slope of the energy line then $(S_f dx)$ is the loss of energy which is spent in overcoming the resistance in a small reach dx. Thus in a small reach dx of the channel the net change of specific energy dE becomes

$$dE = (S_0\,dx - S_f\,dx)$$

or $$\frac{dE}{dx} = S_0 - S_f$$

By substituting this value of (dE/dx) and solving for (dy/dx)

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1-\frac{Q^2T}{gA^3}}$$

which is again same as Eq. 16.1 derived earlier.

3. Dynamic Equation for G.V.F. in Wide Rectangular Channel. Equation 16.1 can be expressed in simplified form for a wide rectangular channel as indicated below.

For a wide rectangular channel section of width B the hydraulic radius can be replaced by the depth of flow y since

$$R = \frac{By}{B+2y} \approx \frac{By}{B} \approx y$$

Further according to Manning's formula

$$Q = \frac{1}{n}(By)\,y^{2/3}\,S_f^{1/2}$$

$$= \frac{1}{n}(By_n)\,(y_n)^{2/3}\,S_0^{1/2}$$

It may be noted that the hydraulic radius has been replaced by the depth of flow and n is assumed to be same for uniform and non-uniform flows. Therefore

$$\frac{S_f}{S_0} = \left(\frac{y_n}{y}\right)^{10/3}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, if Chezy's formula is used instead of the Manning's, the value of

$$\frac{S_f}{S_0} = \left(\frac{y_n}{y}\right)^3$$

In the same manner for a channel of rectangular section

$$\frac{Q^2T}{gA^3} = \frac{Q^2B}{g(B^3y^3)} = \frac{q^2}{gy^3} = \left(\frac{y_c}{y}\right)^3$$

By substituting in Eq. 16.1 it becomes

$$\frac{dy}{dx} = S_0 \frac{1-\left(\frac{y_n}{y}\right)^{10/3}}{1-\left(\frac{y_c}{y}\right)^3} \quad ...(16.2)$$

and

$$\frac{dy}{dx} = S_0 \frac{1-\left(\frac{y_n}{y}\right)^{3}}{1-\left(\frac{y_c}{y}\right)^3} \quad ...(16.3)$$

4. Relation between Water Surface Slopes and Channel Bottom Slope. The term (dy/dx) represents the slope of the water surface with respect to the channel bottom. But often the water surface slope S_w with respect to horizontal may be required to be determined. As such a relation between the water surface slope S_w, , the channel bottom slope S_0 and the slope (dy/dx) may be developed which facilitates the determination of S_w when S_0 and (dy/dx) are known. For rising water surface as shown in Fig. 16.2 (a), from triangle *abd*,

$$S_w = \sin\alpha = \frac{bd}{ba} = \frac{cd-cb}{ba}$$

But $$\frac{cd}{ba} = \sin\theta = S_0, \text{ and } \frac{cb}{ba} = \frac{dy}{dx}$$

$\therefore$ $$S_w = S_0 - \left(\frac{dy}{dx}\right) \quad ...(16.4)$$

However, in this case if the water surface is such that point *b* lies above point *d*, then $(dy/dx) > S_0$, and hence Eq. 16.4 becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$S_w = \left(\frac{dy}{dx}\right) - S_0 \qquad ...(16.4\,a)$$

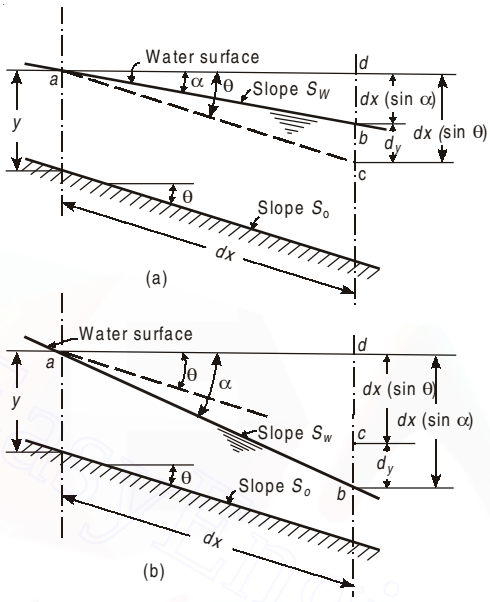


Figure16.2 *Relation between water surface and channel bottom slopes*

For falling water surface as shown in Fig. 16.2 (b), from triangle *abd*,

$$S_w = \sin\alpha = \frac{bd}{ba} = \frac{cd + cb}{ba}$$

Again $\dfrac{cd}{ba} = \sin\theta = S_0$, and $\dfrac{cb}{ba} = \dfrac{dy}{dx}$

$$\therefore \quad S_w = S_0 + \left(\frac{dy}{dx}\right) \qquad ...(16.5)$$

16.3 CLASSIFICATION OF CHANNEL BOTTOM SLOPES

The channel bottom slopes are classified in the various categories as mentioned below:

(i) Critical Slope. The channel bottom slope is designated as critical when the bottom slope S_0 is equal to the critical slope S_c, i.e., $S_0 = S_c$. Thus in this case the normal depth of flow will be equal to the critical depth, i.e., $y_n = y_c$.

(ii) Mild Slope. The channel bottom slope is designated as mild when the bottom slope S_0 is less than the critical slope S_c, i.e., $S_0 < S_c$. The application of Manning's or Chezy's formula will then indicate that when the bottom slope is mild, the normal depth of flow is greater than the critical depth i.e., $y_n > y_c$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) Steep Slope. The channel bottom slope is designated as steep when the bottom slope S_0 is greater than the critical slope, i.e., $S_0 > S_c$. Again the application of Manning's or Chezy's formula will indicate that when the bottom slope is steep, the normal depth of flow is less than the critical depth i.e., $y_n < y_c$.

(iv) Horizontal Slope. When the channel bottom slope is equal to zero i.e., $S_0 = 0$, the bottom slope is designated as horizontal. Obviously for a channel with horizontal bottom the normal depth of flow $y_n = \infty$ (infinity).

(v) **Adverse Slope.** When the channel bottom slope instead of falling rises in the direction of flow it is designated as an adverse slope. Thus in a channel with adverse bottom slope, S_0 is less than zero [i.e., $(S_0 < 0)$] or it is negative. Obviously for an adverse-slopped channel the normal depth of flow y_n is imaginary or it is non-existent.

16.4 CLASSIFICATION OF SURFACE PROFILES

The various water surface profiles occurring in the channels are designated with reference to the bottom slopes of the channels. Thus surface profiles which occur in mild-sloped channels are known

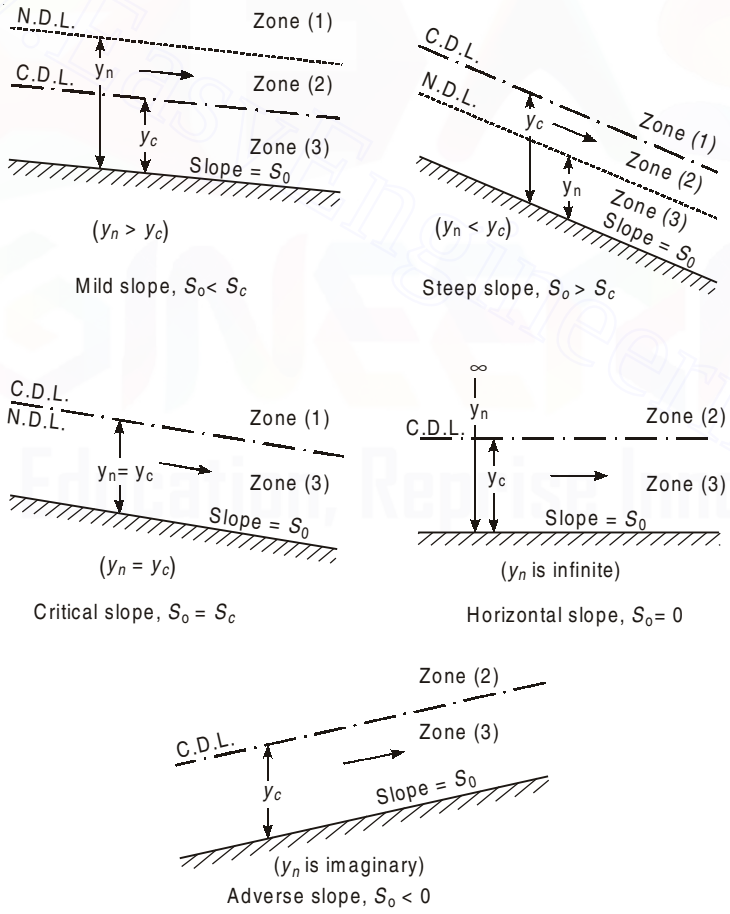


Figure 16.3 *Different zones for water surface profiles in channels of different bottom slopes*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

as *M*-curves; those which occur in steep-slopped channels are known as *S*-curves; those which occur in critical-slopped channels are known as *C*-curves; those which occur in horizontal channels are known as *H*-curves, and those which occur in adverse-slopped channels are known as *A*-curves. These water surface profiles may be further classified depending upon the position of water surface relative to critical depth y_c and normal depth y_n. For the given discharge and channel section the normal depth line (N.D.L.) and the critical depth line (C.D.L.) divide the entire space above the channel bottom into three zones (Fig. 16.3). In zone 1 the given depth y lies above y_n and y_c; in zone 2 the given depth y lies between y_n and y_c and in zone 3, y lies below y_n and y_c. For mild and steep slopes three such zones are possible and the corresponding three surface profiles are designated as M_1, M_2, M_3 and S_1, S_2, S_3 on the mild and steep slopes respectively. On the critical slope the normal depth y_n and the critical depth y_c being the same, zone 2 vanishes and only two zones 1 and 3 exist, so that the given depth, y lies either above the critical depth or below the critical depth resulting in either C_1 or C_2 curves. For horizontal-slopped channels normal depth is infinite and for adverse-slopped channels it is imaginary, as such only two zones 2 and 3 are possible, resulting in only two types of profiles H_2, H_3 and A_2, A_3 on these two slopes. Thus it can be seen that in all twelve surface profiles are possible as shown in Fig. 16.4.

16.5 CHARACTERISTICS OF SURFACE PROFILES

Figure 16.4 shows the various surface profiles. The surface profiles can be classified as back water curves and drawdown curves depending on whether the depth of flow increases or decreases in the direction of flow (or in other words, dy/dx is positive or negative). The study of surface profiles shown in Fig. 16.4 will indicate that all the surface profiles with subscript 1 and 3, that is M_1, M_3, S_1, S_3, C_1, C_3, H_3 and A_3, are back water (or rising) curves while those with subscript 2, that is M_2, S_2, H_2 and A_2 are drawdown (or falling) curves. In order to plot a certain type of profile, it is necessary to know the characteristics of each profile, which may be determined from Eq. 16.2 or 16.3 as discussed below.

1. Surface Profile in Mild-Sloped Channels. In a mild sloped channel there will be three zones viz., $y > y_n > y_c$, $y_n > y > y_c$, and $y_n > y_c > y$, in which respectively M_1, M_2 and M_3 curves will be formed. In the first zone, that is, when $y > y_n > y_c$, the given depth y can have the limiting values as $y \to y_n$ on the upstream side and $y \to \infty$ on the downstream side. Equation 16.2 or 16.3 shows that when $y > y_n > y_c$, $\frac{dy}{dx}$ is positive and as $y \to y_n$, $\frac{dy}{dx} \to 0$ and as $y \to \infty$, $\frac{dy}{dx} \to S_0$. This indicates that M_1 curve meets the normal depth (or y_n) line asymptotically on the upstream side and it tends to be horizontal on the downstream side.

In the second zone, that is, when $y_n > y > y_c$, $\frac{dy}{dx}$ is negative and as $y \to y_n$, $\frac{dy}{dx} \to 0$ and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$, thereby indicating that M_2 curve meets the normal depth (or y_n) line asymptotically on the upstream side and it meets the critical depth (or y_c) line normally on the downstream side.

In the third zone, that is, when $y_n > y_c > y$, $\frac{dy}{dx}$ is positive, and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$ and as $y \to 0$, $\frac{dy}{dx} \to \infty$, thereby indicating that M_3 curve meets the critical depth (or y_c) line and the channel bottom line normally.

2. Surface Profiles in Steep-Sloped Channels. In a steep-sloped channel also there will be three zones viz., $y > y_c > y_n$, $y_c > y > y_n$, and $y_c > y_n > y$, in which respectively S_1, S_2 and S_3 curves will be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

formed. In the first zone, that is when $y > y_c > y_n$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to \infty$ and when $y \to \infty$, $\frac{dy}{dx} \to S_0$. That is on the upstream side S_1 curve meets the critical depth (or y_c) line normally and on the downstream side it tends to be horizontal.

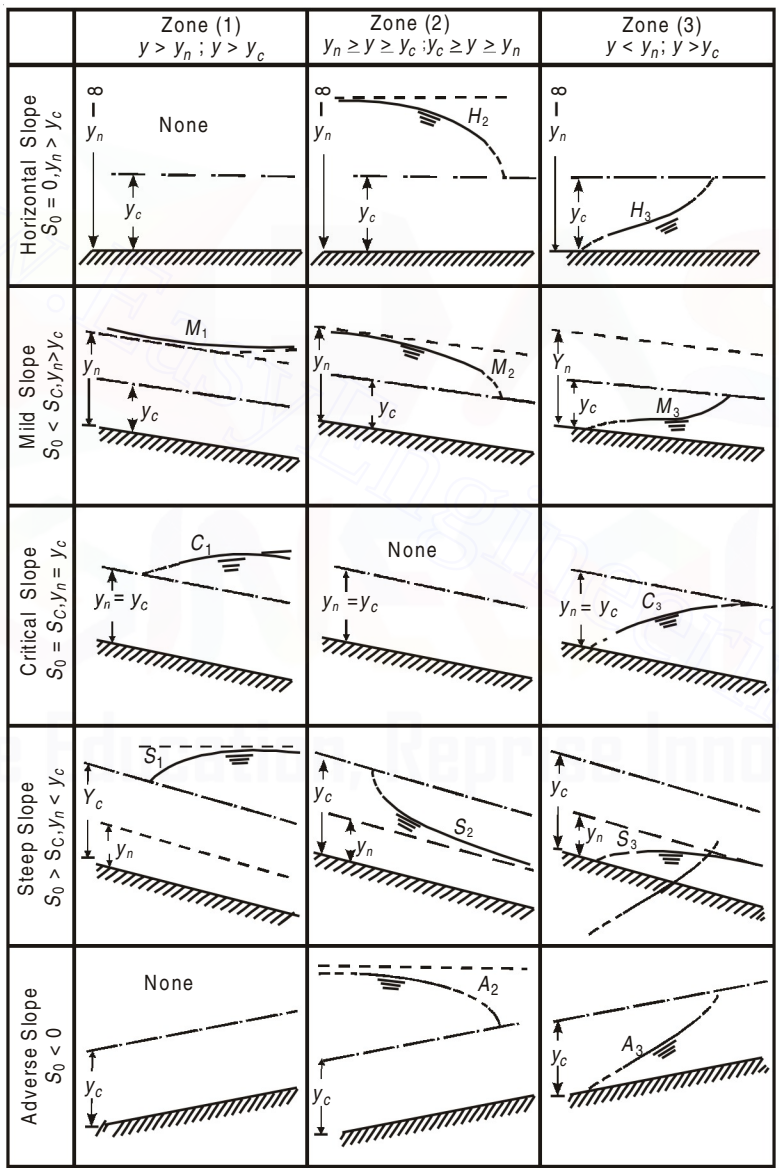


Figure 16.4. *Flow profiles of gradually varied flow*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In the second zone, that is, when $y_c > y > y_n$, $\frac{dy}{dx}$ is negative and as $y \to y_c$, $\frac{dy}{dx} \to \infty$ and as $y \to y_n$, $\frac{dy}{dx} \to 0$, thereby indicating that the S_2 curve meets the critical depth (or y_c) line normally on the upstream side and on the downstream side it meets the normal depth (or y_n) line asymptotically.

In the third zone, that is, when $y_c > y_n > y$, $\frac{dy}{dx}$ is positive and as $y \to y_n$, $\frac{dy}{dx} \to 0$, and as $y \to 0$, $\frac{dy}{dx} \to \infty$, thereby indicating that the S_3 curve meets the channel bed normally and it is asymptotic to the normal depth (or y_n) line.

3. Surface Profiles in Critical-Sloped Channels. In a critical-slopped channel as discussed earlier only two zones 1 and 3 exist and hence only C_1 and C_3 curves will be formed. In the first zone, that is, when $y > y_c = y_n$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to S_0 = S_c$, and as $y \to \infty$, $S_0 = S_c$, thereby indicating that C_1 curve will be more or less a horizontal line.

In the third zone, that is, when $y < y_c = y_n$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to S_0 = S_c$, and as $y \to 0$, $\frac{dy}{dx} \to S_0 = S_c$, thereby indicating that C_3 curve will also be more or less a horizontal line.

It may however be stated that if Chezy's formula is used and accordingly Eq. 16.3 is considered then the C_1 and C_3 curves will be horizontal straight lines, but if Manning's formula is used and accordingly Eq. 16.2 is considered then the C_1 and C_3 curves will be slightly curved.

4. Surface Profiles in Horizontal Channels. In a channel with horizontal bottom $S_0 = 0$ and hence Eq. 16.1 may be expressed as

$$\frac{dy}{dx} = \frac{-S_f}{1 - \frac{Q^2 T}{gA^3}} = \frac{-\frac{n^2 V^2}{y^{4/3}}}{1 - \left(\frac{y_c}{y}\right)^3} \qquad \ldots(16.6)$$

if Manning's formula is used, and

$$\frac{dy}{dx} = \frac{-\frac{V^2}{C^2 y}}{1 - \left(\frac{y_c}{y}\right)^3} \qquad \ldots(16.7)$$

if Chezy's formula is used.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further in a channel with horizontal bottom since $y_n = \infty$ only two zones 2 and 3 exist and accordingly only H_2 and H_3 curves will be formed.

In the second zone, that is, when $y > y_c$, $\frac{dy}{dx}$ is negative and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$, thereby indicating that H_2 curve meets the critical depth (or y_c) line normally at the downstream end and at the upstream end it tends to approach horizontal line tangentially.

In the third zone, that is, when $y < y_c$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$ and as $y \to 0$, $\frac{dy}{dx} \to \infty$, thereby indicating that H_3 curve meets the critical depth (or y_c) line and the channel bottom line normally (i.e., vertically).

TABLE 16.1 *Types of Flow Profiles in Prismatic Channels*

Channel Slope	*Symbol*	*Depth Relations*	$\frac{dy}{dx}$	*Type of profile*	*Type of flow*
Horizontal $[S_0 = 0]$	None	$y > y_n > y_c$		None	None
	H_2	$y_n > y > y_c$	–	Drawdown	Subcritical
	H_3	$y_n > y_c > y$	+	Backwater	Supercritical
Mild $[0 < S_0 < S_c]$	M_1	$y > y_n > y_c$	+	Backwater	Subcritical
	M_2	$y_n > y > y_c$	–	Drawdown	Subcritical
	M_3	$y_n > y_c > y$	+	Backwater	Supercritical
Critical $[S_0 = S_c > 0]$	C_1	$y > y_c = y_n$	+	Backwater	Subcritical
	None	$y_c = y = y_n$		None	None
	C_3	$y_c = y_n > y$	+	Backwater	Supercritical
Steep $[S_0 > S_c > 0]$	S_1	$y > y_c > y_n$	+	Backwater	Subcritical
	S_2	$y_c > y > y_n$	–	Drawdown	Supercritical
	S_3	$y_c > y_n > y$	+	Backwater	Supercritical
Adverse $[S_0 < 0]$	None			None	None
	A_2	$y > y_c$	–	Drawdown	Subcritical
	A_3	$y_c > y$	+	Backwater	Supercritical

5. Surface Profiles in Adverse-Slopped Channels. In a channel with adverse bottom slope since $S_0 < 0$ (i.e., S_0 is negative) the normal depth is imaginary and hence only two zones 2 and 3 exist in which A_2 and A_3 curves will be formed.

In the second zone, that is, when $y > y_c$, Eq. 16.1 indicates that $\frac{dy}{dx}$ is negative. Again when $y \to y_c$, $\frac{dy}{dx} \to -\infty$ and when $y \to \infty$, $\frac{dy}{dx} \to S_0$ thereby indicating that A_2 curve tends to be horizontal at the upstream end and at the downstream end it meets the critical depth (or y_c) line normally.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In the third zone, that is, when $y < y_{c'}$ $\frac{dy}{dx}$ is positive, and as $y \to y_{c'}$, $\frac{dy}{dx} \to -\infty$, and as $y \to 0$, $\frac{dy}{dx} \to \infty$ thereby indicating that A_3 curve meets the critical depth (or y_c) line and the channel bottom line normally.

For the sake of convenience the above discussed characteristics of the various surface profiles have been tabulated in Table 16.1.

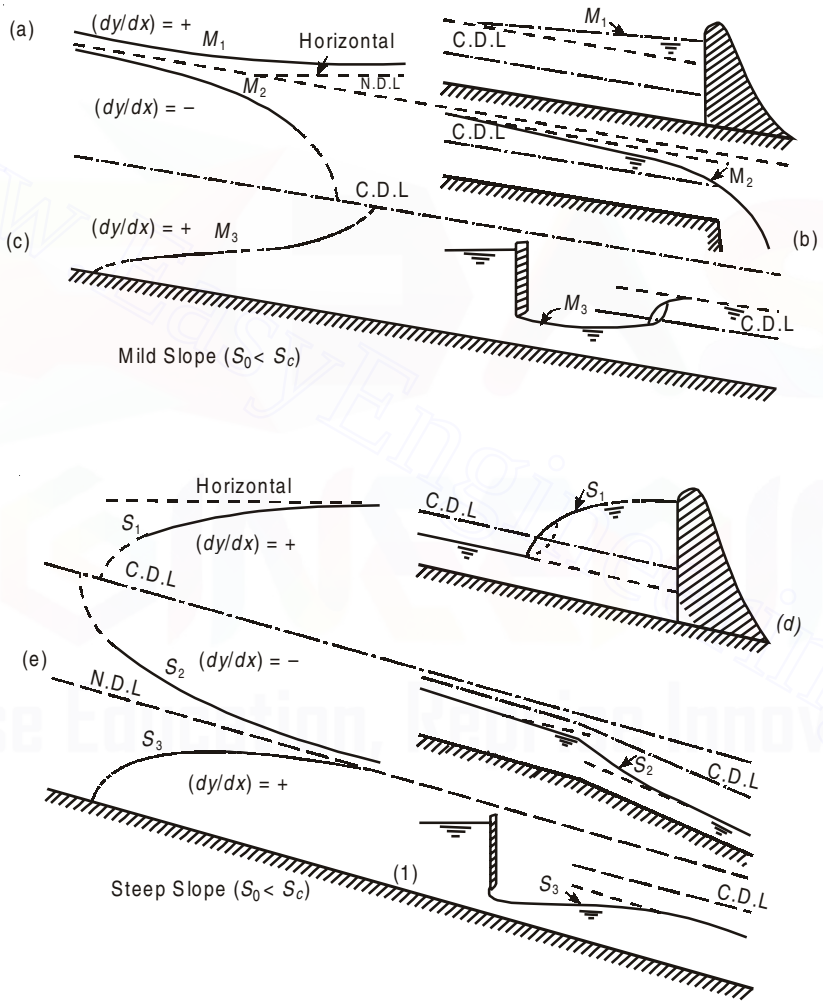


Figure 16.5 *Practical examples of M and S surface profiles*

Some of the practical examples of the various surface profiles are shown in Figs. 16.5 and 16.6. Since the flow profiles near the critical depth line and the channel bottom cannot be accurately defined by the theory of gradually varied flow, they are shown with dotted lines.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In order to determine the class of surface profile in a given channel the following procedure may be adopted :

(i) Compute y_n and y_c for the given discharge and plot the lines representing channel bed and the lines of normal and critical depths.

(ii) By comparing the normal depth and the critical depth determine whether the channel slope is mild, critical, steep, adverse or horizontal.

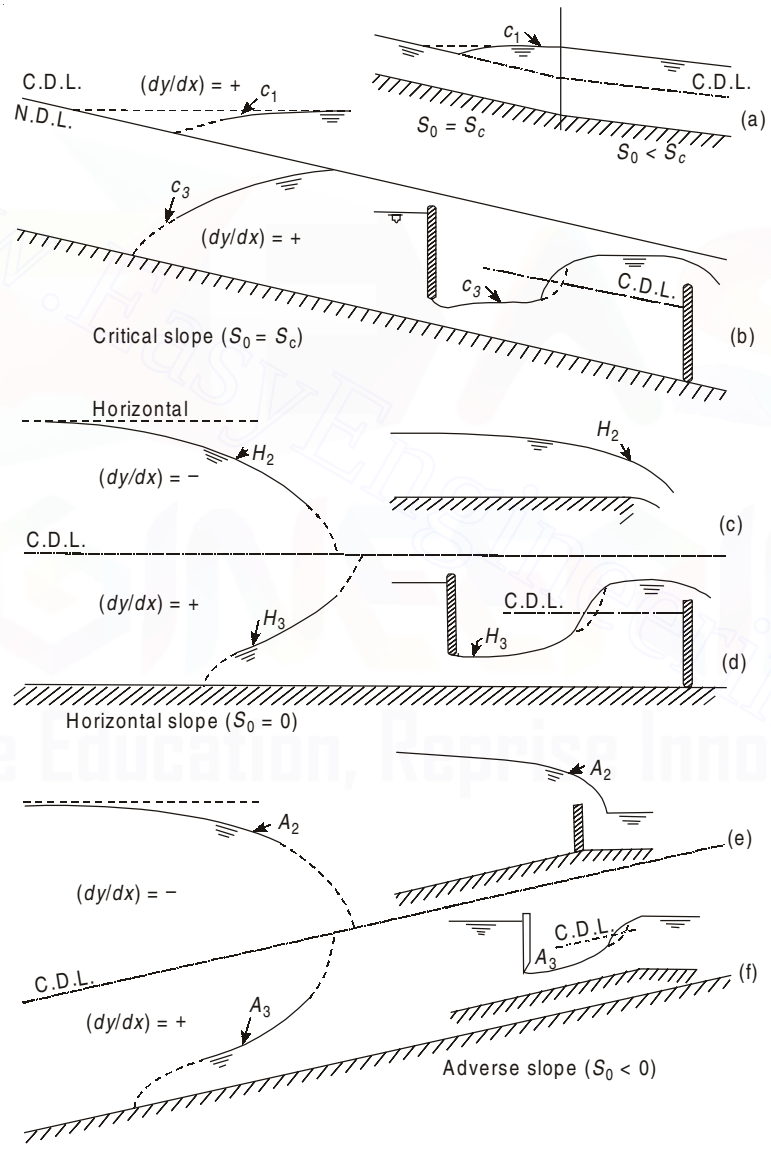


Figure 16.6 *Practical examples of C, H and A surface profiles.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) Knowing the normal depth and the depth at the control section, determine the type of the surface profile.

A *control section* is that at which for a given discharge the depth of flow is known or it can be controlled to a required value. If the depth of non-uniform flow y is above the critical depth it is governed by a downstream control and if it is less than the critical depth it is governed by an upstream control.

Thus knowing the values of the given depth y, normal depth y_n and the critical depth y_c the appropriate surface profile can be sketched.

(iv) If in any reach the supercritical depth has to meet the subcritical depth, i.e., when the stream of water has to cross the critical depth line then there will be a hydraulic jump developed in between, as shown in Figs. 16.5 and 16.6.

16.6 INTEGRATION OF THE VARIED FLOW EQUATION

In practice it is often required to determine the distance upto which the surface profile of gradually varied flow extends. For instance, if a weir is constructed across a river having a mild slope then it may be required to estimate the distance on the upstream side upto which the effect of the resulting M_1 profile exists. In order to solve the problems of this type it is necessary to integrate the dynamic equation of gradually varied flow. The various methods developed for integrating the varied flow equation may be broadly classified as follows:

(a) The step method.

(b) The graphical integration method.

(c) The direct integration method.

(a) The Step Method. In the step method of integration the entire length of the channel is divided into short reaches and the computation is carried out step by step from one end of the reach to the other. Figure 16.7 illustrates a channel reach of length dx, which is sufficiently small so that in this reach the

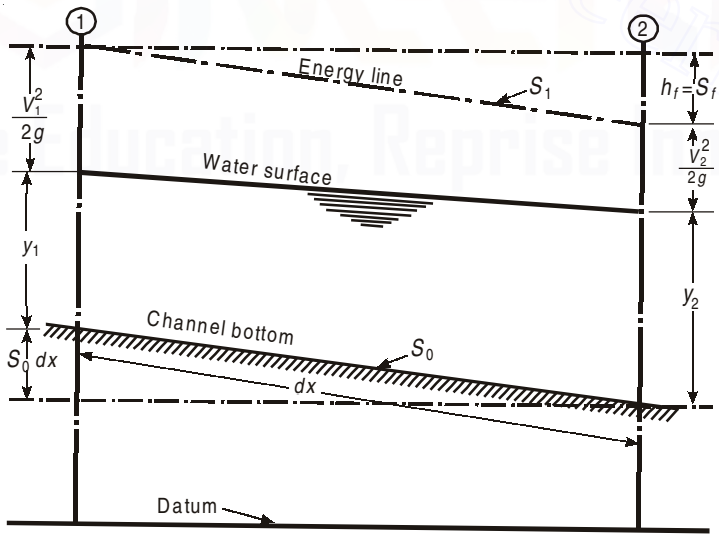


Figure 16.7 *Short reach of channel*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

surface profile may be approximated by a straight line. Equating the total energies at the sections 1 and 2 at the either end of the channel reach under consideration the following expression is obtained

$$(S_0)\,dx + y_1 + \frac{V_1^2}{2g} = y_2 + \frac{V_2^2}{2g} + (S_f)\,dx$$

in which all the notations are as shown in Fig. 16.7.

Since $E_1 = y_1 + \frac{V_1^2}{2g}$ and $E_2 = y_2 + \frac{V_2^2}{2g}$

where E_1 and E_2 are the values of the specific energy at sections 1 and 2, the above expression may be written as

$$(S_0)\,dx + E_1 = E_2 + (S_f)\,dx$$

Solving the above expression for dx

$$dx = \frac{E_2 - E_1}{S_0 - S_f} = \frac{\Delta E}{S_0 - S_f} \qquad \ldots(16.8)$$

where ΔE represents the change in specific energy between the sections 1 and 2.

The step method of integration is based on Eq. 16.8. For the computation of the length x of the surface profile the various quantities that should be known are discharge Q, the channel shape, the bottom slope S_0, Manning's n or Chezy's C and the depth of flow at one of the sections (usually at one of the control sections). The procedure for computation is as follows:

1. Compute the area of flow section A, wetted perimeter P and the hydraulic radius R corresponding to the known value of the depth of flow y.

2. Compute the mean velocity V, the velocity head $(V^2/2g)$, the specific energy E and the energy line slope S_f at the channel section where the depth of flow is known. The energy line slope S_f may be computed by using either Manning's or Chezy's formula as

$$S_f = \frac{n^2V^2}{R^{4/3}}$$

or

$$S_f = \frac{V^2}{C^2R}$$

3. Assume a new depth of flow for another section lying very close to the previous one and from the assumed depth of flow y compute the various quantities as indicated in steps (1) and (2) above. The assumed depth will be either more or less than the known depth at the control section, depending on whether the surface profile is rising or falling from the control section. In order to keep the distance dx between the two adjacent sections small, as far as possible small incremental increase or decrease in depth should be assumed.

4. Obtain the value of the change in specific energy ΔE, from the computed values of the specific energies E_1 and E_2 at the two adjacent sections.

5. In Eq. 16.8 the average value of the energy line slopes at the two adjacent sections is to be used. The average value of S_f is denoted by $\bar{S}_f$ and is equal to $\frac{1}{2}(S_{f1} + S_{f2})$, where S_{f1} and S_{f2} are the values

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the slope of the energy line at the two adjacent sections. Thus compute $\bar{S}_f$ from the computed values of S_{f1} and S_{f2} at the two adjacent sections.

6. Substitute the values of ΔE, S_0 and $\bar{S}_f$ in Eq. 16.8 and compute dx, which represents the length of the portion of the surface profile lying between the two adjacent sections.

7. Repeat the process by assuming different values of the depths of flow y at the adjacent sections until the required section is reached. Add the resulting dx values to obtain the required length x of the flow profile.

The step method of integration is very simple. However, the accuracy of this method depends on the depth increments chosen.

(b) The Graphical Integration Method. The graphical method of integration of varied flow equation can be used for both prismatic as well as non-prismatic channels. Consider Eq. 16.1 viz.,

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2 T}{gA^3}}$$

Since for a given discharge, A, T, and S_f are functions of the depth of flow y, we may write

$$dx = f(y)\, dy \qquad ...16.9)$$

in which $$f(y) = \left(1 - \frac{Q^2 T}{gA^3}\right) \Big/ \left(S_0 - S_f\right)$$

Now if y_1 and y_2 are the depths of flow at points distant x_1 and x_2 from the control section, then by integrating Eq. 16.9, we get

$$(x_2 - x_1) = \int_{x_1}^{x_2} dx = \int_{y_1}^{y_2} f(y)dy \qquad ...(16.10)$$

By assuming several values of y, the corresponding values of $f(y)$ may be computed. A curve of y versus $f(y)$ is then prepared and the area under the curve between $y = y_1$ and $y = y_2$ is determined which will give the longitudinal distance $(x_2 - x_1)$ required for this change in the depth of flow, in accordance with Eq. 16.10. If y_1 and y_2 do not differ much then approximately

$$(x_2 - x_1) = \frac{f(y_1) + f(y_2)}{2}(y_2 - y_1)$$

in which $f(y_1)$ and $f(y_2)$ are the values of $f(y)$ at $y = y_1$ and $y = y_2$.

(c) The Direct Integration Method. There are several methods of direct integration of varied flow equation, however, only Bresse's method is discussed here.

Bresse's Method. Bresse's method of integrating the varied flow equation is applicable in the case of very wide rectangular channels. In this method Chezy's formula is used for the evaluation of the effect of frictional resistance to the flow. The general varied flow equation may be expressed for this case as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{dy}{dx} = S_0 \frac{1-\left(\frac{y_n}{y}\right)^3}{1-\left(\frac{y_c}{y}\right)^3} \qquad \text{...(16.10 a)}$$

Now if $(y / y_n) = u$, then $dy = y_n du$, and the above equation can be rearranged as

$$dx = \frac{y_n}{S_0}\left[1-\left(1-\frac{y_c^3}{y_n^3}\right)\frac{1}{1-u^3}\right]du$$

Integrating the above equation, we get

$$x = \frac{y_n}{S_0}\left[u-\left\{1-\left(\frac{y_c}{y_n}\right)^3\right\}\int\frac{du}{1-u^3}\right] + \text{constant} \qquad \text{...(16.11)}$$

The distance x can be calculated from Eq. 16.11 if the value of the integral $\left[\int\frac{du}{1-u^3}\right]$ is known. The function $\int\frac{du}{1-u^3}$ is known as Bresse's varied flow function and the tables are available giving the numerical values of this function for different values of u. However, it can also be calculated from the following expression.

$$\int\frac{du}{1-u^3} = \left[\frac{1}{6}\log_e\frac{u^2+u+1}{(u-1)^2} - \frac{1}{\sqrt{3}}\cot^{-1}\left(\frac{2u+1}{\sqrt{3}}\right)\right] \qquad \text{...(16.12)}$$

Further the ratio $\left(\frac{y_c}{y_n}\right)$ in a wide rectangular channel can be expressed as

$$\left(\frac{y_c}{y_n}\right)^3 = \frac{C^2 S_0}{g}$$

where C is Chezy's coefficient.

Thus substituting the above expression in Eq. 16.11 and writing $\int\frac{du}{1-u^3}$ as $\phi(u)$, the Bresse's solution for the varied flow equation becomes

$$(x_2 - x_1) = \frac{y_n}{S_0}\left[(u_2 - u_1) - \left(1-\frac{C^2 S_0}{g}\right)\times\{\phi(u_2)-\phi(u_1)\}\right] \qquad \text{...(16.13)}$$

in which u_2 and u_1 are the values of u at x_2 and x_1 respectively and $\phi(u_2)$ and $\phi(u_1)$ are the values of $\phi(u)$ for u equal to u_2 and u_1 respectively.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

16.7 HYDRAULIC JUMP

The hydraulic jump is defined as the sudden and turbulent passage of water from a supercritical state to subcritical state. It has been classified as rapidly varied flow, since the change in depth of flow from rapid to tranquil state is in an abrupt manner over a relatively short distance. The flow in a hydraulic jump is accompanied by the formation of extremely turbulent rollers and there is a considerable dissipation of energy.

A hydraulic jump will form when water moving at a supercritical velocity in a relatively shallow stream strikes water having a relatively large depth and subcritical velocity. It occurs frequently in a canal below a regulating sluice, at the foot of a spillway, or at the place where a steep channel bottom slope suddenly changes to a flat slope.

In order to study the conditions of flow before and after the hydraulic jump the application of the energy equation does not provide an adequate means of analysis, because hydraulic jump is associated with an appreciable loss of energy which is initially unknown. As such in the analysis of hydraulic jump the momentum equation is used by considering the portion of the hydraulic jump as the control volume. The following assumptions are, however, made in this analysis:

(1) It is assumed that before and after jump formation the flow is uniform and the pressure distribution is hydrostatic.

(2) The length of the jump is small so that the losses due to friction on the channel floor are small and hence neglected.

(3) The channel floor is horizontal or the slope is so gentle that the weight component of the water mass comprising the jump is negligibly small.

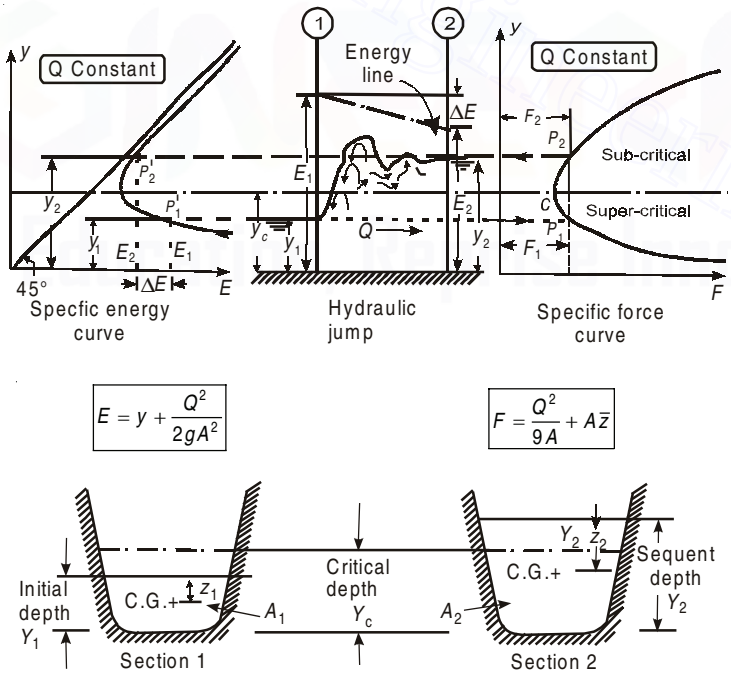


Figure 16.8 *Hydraulic jump in a prismatic channel*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider a hydraulic jump formed in a prismatic channel with horizontal floor carrying a discharge Q as shown in Fig. 16.8. Let the depth of flow before the jump at section 1 be y_1 and the depth of flow after the jump at section 2 be y_2. The depth y_1 is known as *initial depth* and y_2 is known as *sequent depth.* The symbols A_1, V_1 and $\bar{z}_1$ represent the area of cross-section, mean velocity of flow and the depth of the centroid of area A_1 below the free surface respectively at section 1 before the jump and A_2, V_2 and $\bar{z}_2$ are the corresponding quantities at section 2 after the jump.

The only external forces acting on the mass of water between the sections 1 and 2 are the hydrostatic pressures P_1 and P_2 at sections 1 and 2 respectively, as the frictional loss has been assumed to be negligible. Hence in accordance with the momentum equation

$$(P_2 - P_1) = \rho Q (V_1 - V_2)$$

or
$$wA_2\,\bar{z}_2 - wA_1\,\bar{z}_1 = \frac{w}{g}\,Q\left(\frac{Q}{A_1} - \frac{Q}{A_2}\right)$$

or
$$\frac{Q^2}{gA_1} + A_1\,\bar{z}_1 = \frac{Q^2}{gA_2} + A_2\,\bar{z}_2 \qquad ...(16.14)$$

As already explained in Chapter 15, the sum $\left(\frac{Q^2}{gA} + A\bar{z}\right)$ is called the *specific force,* designated by F.

Thus if F_1 and F_2 represent the specific force at sections 1 and 2 respectively then Eq. 16.14 may be written as

$$F_1 = F_2 \qquad ...(16.15)$$

The specific force F is a function of the depth of flow y and hence it can be plotted against the depth of flow y to obtain specific force curve as shown in Fig. 16.8. It may be noted from the specific force curve that alike specific energy the specific force also attains a minimum value at critical depth y_c.

It is obvious from Eq. 16.15 that the specific force F_1 corresponding to y_1 and the specific force F_2 corresponding to y_2 are same which may also be seen from the specific force curve shown in Fig. 16.8. The initial depth y_1 and the sequent depth y_2 are commonly known as the *conjugate depths,* which indicate the same specific force (in order to distinguish them from the alternate depths which indicate the same specific energy).

Equation 16.14 enables the determination of y_2, if y_1 is given or vice versa for a known discharge Q flowing in a given channel section. Alternatively from the specific force curve also, knowing y_1 and hence F_1, y_2 can be determined or vice versa, since both y_1 and y_2 correspond to same specific force.

Knowing the conjugate depths y_1 and y_2 for a known discharge Q in a given channel section the specific energies E_1 and E_2 at the sections 1 and 2 respectively may be computed from which the loss of energy in the jump can be determined. Alternatively the loss of energy in the jump may also be determined by using the specific energy curve in combination with specific force curve as indicated below.

Knowing y_1 the corresponding specific force F_1 can be found from specific force curve as indicated by the point P_1. A vertical through point P_1 will cut the curve at point P_2. Since $F_1 = F_2$, the ordinate of point P_2 will indicate the depth y_2. Horizontal lines drawn through points P_1 and P_2 will cut the specific energy curve at points P'_1 and P'_2 respectively, indicating the values of the specific energies

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

before and after the jump, i.e., E_1 and E_2 respectively. The horizontal distance between points P'_1 and P'_2 is the energy loss ΔE, due to hydraulic jump.

(a) Hydraulic Jump in Rectangular Channels. For rectangular channels Eq. 16.14 can be further simplified and a relation between the conjugate depths y_1 and y_2 can be obtained. Thus if B is the width of the rectangular channel, then $A_1 = By_1$; $A_2 = By_2$, $\bar{z}_1 = (y_1/2)$, $\bar{z}_2 = (y_2/2)$ and $q = (Q/B)$. By substituting these values in Eq. 16.14, it becomes

$$\frac{Q^2}{g(By_1)} + By_1\left(\frac{y_1}{2}\right) = \frac{Q^2}{g(By_2)} + By_2\left(\frac{y_2}{2}\right)$$

or $$\frac{Q^2}{gB^2}\left(\frac{1}{y_1} - \frac{1}{y_2}\right) = \frac{1}{2}(y_2^2 - y_1^2)$$

or $$\frac{q^2}{g}\left(\frac{y_2 - y_1}{y_1 y_2}\right) = \frac{1}{2}(y_2 - y_1)(y_2 + y_1)$$

or $$\frac{2q^2}{g} = y_1 y_2 (y_1 + y_2) \quad \ldots(16.16)$$

Equation 16.16 is the momentum equation for hydraulic jump in rectangular channels.

Equation 16.16 can be considered as quadratic equation in terms of y_1 or y_2 and its solution gives the relations between the conjugate depths y_1 and y_2 as

$$y_2 = -\frac{y_1}{2} + \sqrt{\left(\frac{y_1}{2}\right)^2 + \frac{2q^2}{gy_1}} \quad \ldots(16.17)$$

$$y_1 = -\frac{y_2}{2} + \sqrt{\left(\frac{y_2}{2}\right)^2 + \frac{2q^2}{gy_2}} \quad \ldots(16.18)$$

In Eqs 16.17 and 16.18 the negative sign before the square root has not been used since it gives negative values.

Equations 16.17 and 16.18 may also be written as

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + \frac{8q^2}{gy_1^3}}\right] \quad \ldots(16.19)$$

$$\frac{y_1}{y_2} = \frac{1}{2}\left[-1 + \sqrt{1 + \frac{8q^2}{gy_2^3}}\right] \quad \ldots(16.20)$$

Since for a rectangular channel $\left(\frac{q^2}{g}\right) = y_c^3$, where y_c is the critical depth for discharge q. The Eqs 16.19 and 16.20 may also be written as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1+\sqrt{1+8\left(\frac{y_c}{y_1}\right)^3}\right] \quad \ldots(16.21)$$

$$\frac{y_1}{y_2} = \frac{1}{2}\left[-1+\sqrt{1+8\left(\frac{y_c}{y_2}\right)^3}\right] \quad \ldots(16.22)$$

Since the Froude numbers Fr_1 and Fr_2 before and after the hydraulic jump respectively are given as

$$Fr_1 = \frac{q}{\sqrt{gy_1^3}} \quad \text{and } Fr_2 = \frac{q}{\sqrt{gy_2^3}}$$

Equations 16.21 and 16.22 may be written as

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1+\sqrt{1+8Fr_1^2}\right] \quad \ldots(16.23)$$

and

$$\frac{y_1}{y_2} = \frac{1}{2}\left[-1+\sqrt{1+8Fr_2^2}\right] \quad \ldots(16.24)$$

When the conjugate depths are known the energy loss ΔE in a hydraulic jump may be computed as

$$\Delta E = \left(y_1+\frac{V_1^2}{2g}\right)-\left(y_2+\frac{V_2^2}{2g}\right)$$

or

$$\Delta E = \left[y_1+\frac{Q^2}{2g(By_1)^2}\right]-\left[y_2+\frac{Q^2}{2g(By_2)^2}\right]$$

or

$$\Delta E = \frac{q^2}{2g}\left(\frac{1}{y_1^2}-\frac{1}{y_2^2}\right)-(y_2-y_1)$$

or

$$\Delta E = \frac{q^2}{2g}\frac{y_2^2-y_1^2}{(y_1y_2)^2}-(y_2-y_1)$$

By substituting for $\left(\frac{q^2}{g}\right)$ from Eq. 16.16 the above expression becomes

$$\Delta E = \frac{1}{4}\frac{y_1y_2(y_1+y_2)(y_2^2-y_1^2)}{(y_1y_2)^2}-(y_2-y_1)$$

which on simplification becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\Delta E = \frac{(y_2 - y_1)^3}{4y_1y_2} \qquad ...(16.25)$$

However, as indicated in Illustrative Example 16.12, the energy loss ΔE in a hydraulic jump in a rectangular channel may also be expressed as

$$\Delta E = \frac{(V_1 - V_2)^3}{2g(V_1 + V_2)} \qquad ...(16.25\ a)$$

where V_1 and V_2 are the mean velocities of flow before and after the jump respectively.

The *height of the jump* h_j may be defined as the difference between the depths after and before the jump, i.e., $h_j = (y_2 - y_1)$.

The *length of the jump* L_j may be defined as the distance measured from the front face of the jump to a point on the surface immediately downstream from the roller. However, the length of the jump cannot be determined analytically. In addition, practical complications arise from the general instability of the phenomenon and the difficulty of defining the beginning and the end sections of the jump. The length of the jump has been investigated experimentally by many hydraulicians and as a general statement it may be said that for a rectangular channel the length of the jump L_j varies between 5 and 7 times the height of the jump, that is,

$$L_j = (5 \text{ to } 7)\, h_j = (5 \text{ to } 7)\, (y_2 - y_1) \qquad ...(16.26)$$

(b) Types of Hydraulic Jump. Equation 16.23 emphasizes the importance of the Froude number Fr_1 of the incoming supercritical flow, as a parameter describing the phenomenon of hydraulic jump. As such according to the studies of U.S. Bureau of Reclamation, depending upon the value of Froude number Fr_1 of the incoming flow, there are five distinct types of the hydraulic jump which may occur on a horizontal floor. These different types of hydraulic jump are shown in Fig. 16.9 and are described below.

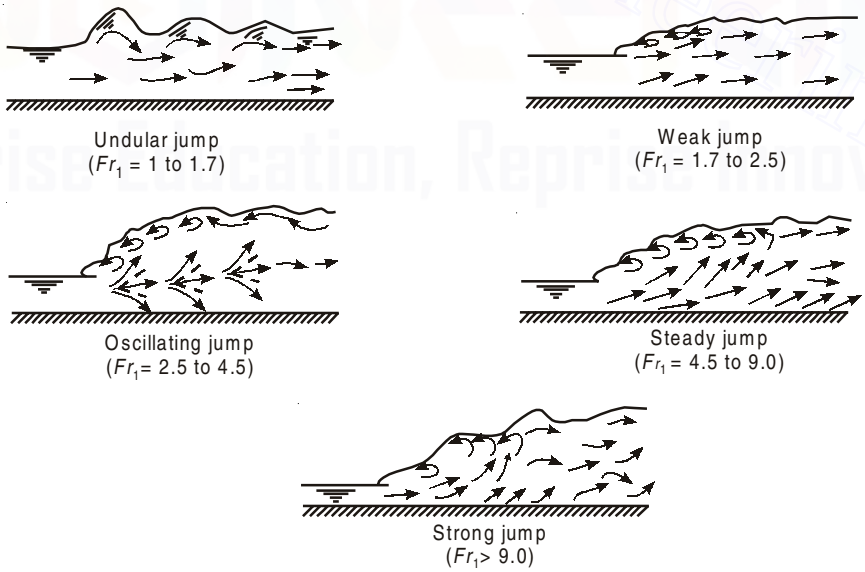


Figure 16.9 *Types of hydraulic jump*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(1) For Fr_1 = 1.0 to 1.7, the water surface shows undulations and the jump is called an *undular jump.*

(2) For Fr_1 = 1.7 to 2.5, the jump formed is called *weak jump,* as the velocity throughout is fairly uniform and only a small amount of energy is dissipated. In this case a series of small rollers form on the jump surface, but the downstream water surface remains quite smooth.

(3) For Fr_1 = 2.5 to 4.5, jump formed is known as an *oscillating jump.* In this case the entering jet of water oscillates back and forth from the bottom to the surface and back again.

(4) For Fr_1 = 4.5 to 9.0 the jump formed is well stabilized and is called a *steady jump.* For this jump the energy dissipation ranges from 45 to 70 per cent.

(5) For Fr_1 = 9.0 and larger the jump formed is called a *strong jump.* In this case a rough surface prevails which continues downstream for a long distance. The jump action is quite rough but is effective since the energy dissipation may reach 85 per cent.

The above described different types of hydraulic jump refer only to channels of rectangular section. In other channel sections the shape of the jump is often complicated additionally by cross currents.

In the above paragraphs only the hydraulic jump in rectangular channel has been discussed, but it may however be mentioned that by using Eq. 16.14 the hydraulic jump in prismatic channel of any shape can be analysed.

(c) Applications of Hydraulic Jump. The phenomenon of hydraulic jump has many practical applications as listed below.

(1) It is a useful means of dissipating excess energy of water flowing over spillways and other hydraulic structures or through sluices and thus preventing possible erosion on the downstream side of these structures.

(2) It raises the water level in the channels for irrigation etc.

(3) It increases the weight on an apron of a hydraulic structure due to increased depth of flow and hence the uplift pressure acting on the apron is considerably counterbalanced.

(4) It increases the discharge through a sluice by holding back the tail water.

(5) It may be used for mixing chemicals in water and other liquids, since it facilitates thorough mixing due to turbulence created in it.

16.8 LOCATION OF HYDRAULIC JUMP

Often it is required to locate the exact position of the hydraulic jump in a channel under different conditions of flow. As such the following three typical cases for the location of the exact position of the hydraulic jump are described below.

Case (1)

In this case a jump forms below a regulating sluice in a mild sloped channel, see Fig. 16.10 (a). The jet of water issuing from the sluice will contract upto vena contracta section at a distance L_e from the sluice which is taken approximately equal to the sluice opening h. Thereafter it will follow M_3 profile as indicated by DE in Fig.16.10 (a). The location of the jump in this case will be considerably affected by the length of the channel reach on the downstream side of the sluice. Thus if there exists a long reach of channel with same slope on the downstream side of the sluice, then after the formation of the jump, uniform flow with depth of flow equal to the normal depth of flow will be developed. Hence the depth of flow after the jump or the sequent depth will be equal to the normal depth of flow. As such from the vena contracta section the depth of flow will gradually increase, following M_3 profile, upto a certain section on the downstream side, where the depth of flow will be equal to the initial depth required for the formation of the jump corresponding to the sequent depth equal to the normal depth

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of flow. At this section the jump will be developed and it will extend from this section on the downstream side upto a distance equal to the length of the jump. Therefore, in order to locate the jump section, compute the initial depth corresponding to the sequent depth equal to the normal depth of flow in the channel and obtain the length of the M_3 profile between the vena contracta section and the section having the depth of flow equal to this initial depth. This method has been illustrated in Illustrative Example 16.6.

On the other hand if there exists a short reach of channel on the downstream side of the sluice, with a free overfall or an abrupt drop at its end, then there will be a M_2 profile developed as shown by *FG* in Fig. 16.10 (a). As such in this case after the formation of the jump the depth of flow will be gradually varying and the sequent depth will lie along the M_2 profile. However, from the vena contracta section the depth of flow will gradually increase following M_3 profile as shown by *DE* in Fig. 16.10 (a). The jump in this case will be developed at a section where the depth of flow along the M_3 profile will be

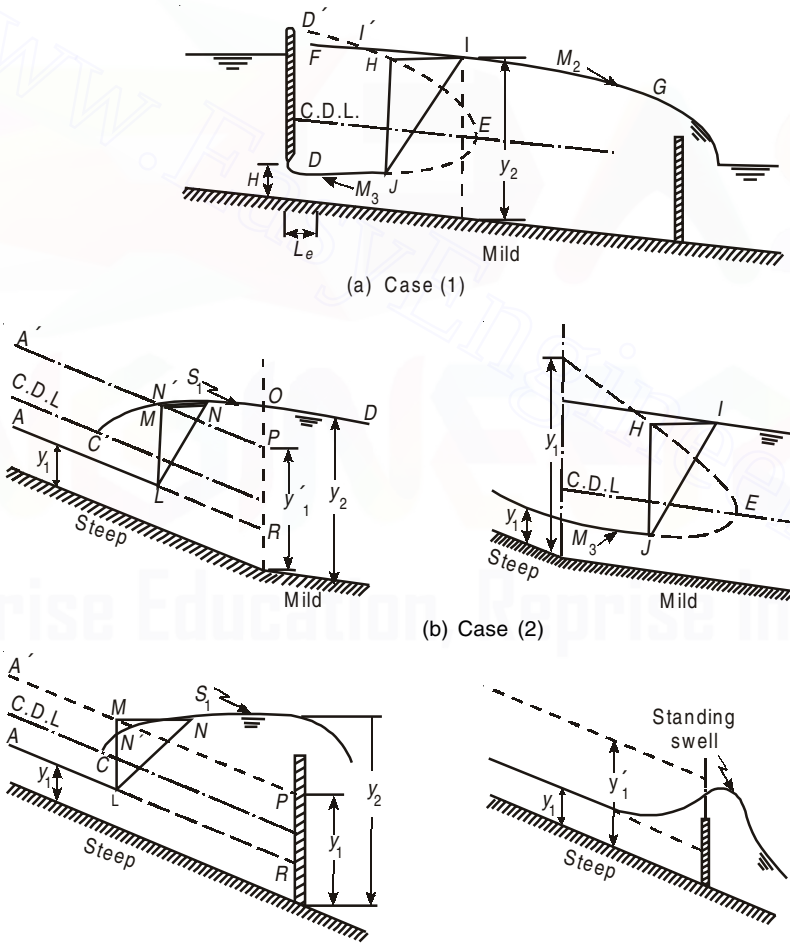


Figure 16.10 *Location of hydraulic jump*

equal to the initial depth required for the formation of the jump corresponding to a sequent depth along the M_2 profile. As such for locating the jump in this case, different values of initial depths are

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

assumed along the M_3 profile. Corresponding to these initial depths the sequent depths are computed and the same are plotted. The curve *D'E* shown in Fig. 16.10 (a) is a plot of the depth sequent to curve *DE* (i.e., M_3 profile). The intersection of M_2 profile i.e., curve *FG* and the curve *D'E* at point *I'* gives the approximate position of the jump if the length of the jump is not taken into account.

However, for taking the length of the jump into account find the initial and sequent depths corresponding to point *I'* and compute the length of the jump from Eq. 16.26. Now by trial locate a horizontal intercept *HI* equal to the length of the jump, between the curves *D'E* and *FG*. Locate a point *J* vertically below the point *H*, on the curve *DE*. Thus it is obvious that the jump will form between the points *J* and *I*, since the depth at *I* is equal to the sequent depth corresponding to the depth at *J* and the distance *HI* is equal to the length of the jump. Thus if the length of the jump is not taken into account, the error involved in the location of the jump is represented by *I'I*.

It can be seen from the above analysis that in the later case the jump can be moved upstream by increasing the downstream depth or raising the curve *FG*. On the other hand, decreasing the downstream depth or lowering the curve *FG* will move the jump downstream.

Furthermore, in both the cases discussed above if the depth of flow at the vena contracta section is equal to the initial depth required for the formation of the jump corresponding to the available sequent depth, then the M_3 profile will not be developed and the jump will be located just at the vena contracta section. On the other hand if the depth of flow at the vena contracta section is more than the required initial depth, then no clear jump will be formed but a submerged jump will be formed close to the sluice.

The above discussed method also applies in the case of the location of the jump formed at the foot of an overflow spillway or a weir.

Case (2)

In this case the jump forms in a channel having a break in the bottom slope that changes from steep to mild, see Fig. 16.10 (b). It is assumed that the flow is uniform in the channel section except in the reach between the jump and the change of slope.

Now if the normal depth in the mild channel is greater than the depth y_1' sequent to the normal depth in the steep channel, then the jump will form in the steep channel. Thus compute the profile S_1 in the steep channel as shown by *OC* in Fig. 16.10 (b). The S_1 profile starts from the point *O* at the section where the change in slope occurs. Compute the depth y_1' sequent to the normal depth in the steep channel, and the line *A'P* is drawn parallel to the bottom slope of the steep channel, at distance y_1'. Then the intersection of S_1 profile (i.e., curve *OC*) and the line *A'P* at point *N'* gives the approximate position of the jump if the length of the jump is not taken into account. On the other hand if the length of the jump is to be taken into account, then compute the length of the jump using Eq. 16.26, taking initial depth as equal to the normal depth in the steep channel and sequent depth as obtained above. Now by trial and error locate horizontal intercept *MN* equal to the length of the jump, between the line *A'P* and the curve *OC*. Locate the point *L* vertically below the point *M*, on the normal depth line in the steep channel. Thus it is obvious that the jump will be formed between the points *L* and *N* since the depth at *N* is equal to the sequent depth corresponding to the depth at *L* and the distance *MN* measures the length of the jump.

On the other hand if the normal depth in the mild channel is less than the depth y_1' sequent to the normal depth in the steep channel, then the jump will form in the mild channel. This case thus becomes similar to case (1) and hence the jump can be located by the same procedure as indicated in case (1).

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Case (3)

In this case the jump forms behind an overflow barrier provided in a steep channel. Theoretically, in this case the jump will form if the depth at the barrier is greater than the depth y_1' sequent to the approaching normal depth in the steep channel. The location of the jump in this case is similar to the case (2) when the jump forms in the steep portion. When the depth at barrier is less than the sequent depth y_1' there will be no jump and the barrier will be crossed by a standing swell.

16.9 SURGES IN OPEN CHANNELS

A surge or surge wave is a moving wave front which brings about an abrupt change in depth of flow. A surge is also often referred to as moving hydraulic jump and is caused by sudden increase or decrease in flow, such as that caused by sudden opening or closing of a gate fixed in the channel. Surges are usually classified as positive surges and negative surges. A positive surge is one which results in an increase in the depth of flow and a negative surge causes a decrease in the depth of flow. Figures 16.11 (a) and (b) shows two types of positive surges and Figs. 16.11 (c) and (d) shows two types of negative surges. Type A (Fig. 16.11 a) is a positive surge having an advancing wave front moving downstream. Type *B* (Fig. 16.11 b) is also a positive surge having an advancing wave front moving upstream.Type *C* (Fig. 16.11 c) is a negative surge having a retreating wave front moving downstream. Type *D* (Fig. 16.11 d) is also a negative surge having a retreating wave front moving upstream. The

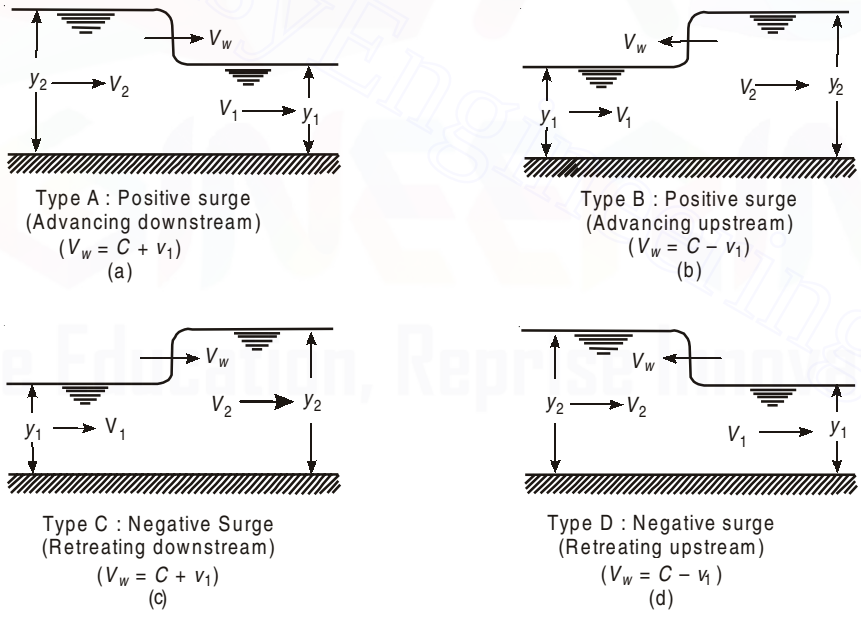


Figure 16.11 *Different types of surges.*

positive surge of type *A* may occur when a gate provided at the head of a channel is suddenly opened. The positive surge of the type *B* may occur when a gate provided at the tail end of a channel is suddenly closed. The negative surge of type *C* may occur when a gate provided at the head of a channel is suddenly closed. The negative surge of type *D* may occur when a gate provided at the tail end of a channel is suddenly opened.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Although the occurrence of a surge is an usteady flow phenomenon, but when the surge is moving at a constant velocity, it can be converted into a case of a steady flow by applying a velocity of the same magnitude but in opposite direction to the flowing stream as well as to the surge.

Positive Surges

Case (a) Surge due to sudden increase of flow

Consider a positive surge of type *A* created in a rectangular channel by suddenly opening a gate as shown in Fig. 16.12. Let V_w be the absolute velocity of the surge moving towards right and let V_1 and V_2 be the velocities and y_1 and y_2 be the corresponding depths at sections 1 and 2 respectively. In order to make it a case of steady flow, apply velocity V_w in opposite directions to the velocities V_1 and V_2 and the surge. Thus the velocities at sections 1 and 2 becomes $(V_1 - V_w)$ and $(V_2 - V_w)$ respectively. Then from continuity, we have

$$y_2(V_2 - V_w) = y_1(V_1 - V_w)$$

from which solving for V_w, we have

$$V_w = \frac{V_1 y_1 - V_2 y_2}{y_1 - y_2} \qquad ...(16.27)$$

and solving for V_2, we have

$$V_2 = \frac{y_1(V_1 - V_w) + y_2 V_w}{y_2} \qquad ...(16.28)$$

Further due to the surge developed there is a change in the momentum caused between the sections 1 and 2 and hence the momentum equation may be expressed as

$$\frac{1}{2}wy_2^2 - \frac{1}{2}wy_1^2 = \frac{w}{g}\, y_2(V_2 - V_w)\,[(V_1 - V_w) - (V_2 - V_w)]$$

or

$$\frac{1}{2}(y_2^2 - y_1^2) = \frac{1}{g}\, y_2(V_2 - V_w)(V_1 - V_2) \qquad ...(16.29)$$

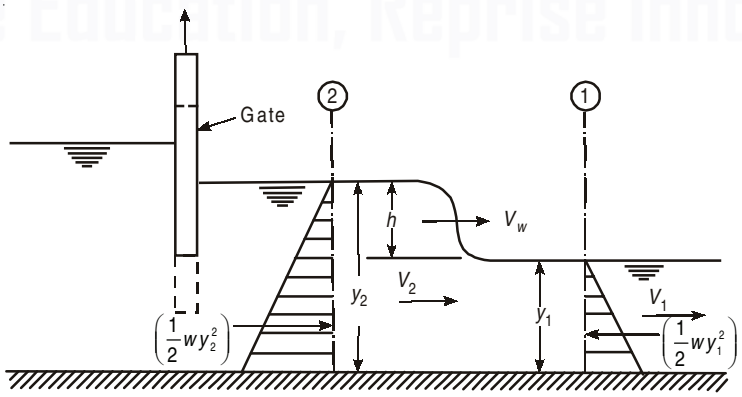


Figure 16.12 *Positive surge of type A in a rectangular channel*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the value of V_2 from Eq. 16.28 in Eq. 16.29 and simplifying, we get

$$(V_w - V_1)^2 = \frac{gy_2}{2y_1}(y_2 + y_1) \quad ...(16.30)$$

or

$$V_w = \left[\sqrt{\frac{gy_2}{2y_1}(y_2 + y_1)}\right] + V_1 \quad ...(16.31)$$

Eliminating V_w from Eqs 16.27 and 16.31 and simplifying, we have

$$(V_1 - V_2)^2 = (y_1 - y_2)^2 \frac{(y_1 + y_2)g}{2y_1y_2} \quad ...(16.32)$$

Equation 16.32 represents the relationship among the initial and final velocities and depth of the surge.

Further Eq. 16.31 represents the absolute velocity of the surge. The celerity of the wave C is defined as the velocity of the wave relative to the velocity of flow and hence $C = (V_w - V_1)$.

$$\therefore \quad C = \sqrt{\frac{gy_2}{2y_1}(y_2 + y_1)} \quad ...(16.33)$$

Multiplying Eq. 16.32 by the square of Eq. 16.33 and simplifying, we have

$$(V_1 - V_2) = \frac{(y_2 - y_1)}{C}\left(\frac{y_2 + y_1}{2y_1}\right)g$$

or

$$(V_1 - V_2) = \frac{h}{C}\left(\frac{y_2 + y_1}{2y_1}\right)g \quad ...(16.34)$$

where $h = (y_2 - y_1)$ is the height of the surge.

When the height of the surge is small compared with the depth of flow, then $y_1 \approx y_2$ and Eq. 16.34 may be written as

$$(V_1 - V_2) = \frac{hg}{C} \quad ...(16.35)$$

Equation 16.33 may also be expressed in terms of h as

$$C = \sqrt{\frac{g(y_1 + h)(2y_1 + h)}{2y_1}}$$

or

$$C = \sqrt{gy_1}\sqrt{1 + \frac{3}{2}\left(\frac{h}{y_1}\right) + \frac{1}{2}\left(\frac{h}{y_1}\right)^2} \quad ...(16.36)$$

If h is less than y_1, the term $\frac{1}{2}\left(\frac{h}{y_1}\right)^2$ becomes much small and hence it can be neglected. Equation 16.36 then simplifies to

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$C = \sqrt{gy_1}\sqrt{1+\frac{3}{2}\left(\frac{h}{y_1}\right)}$$

or
$$C = \sqrt{gy_1}\left(1+\frac{3}{4}\frac{h}{y_1}\right) \quad ...(16.37)$$

Again when the ratio $\left(\frac{h}{y_1}\right)$ is very small, the term $\left(\frac{3}{4}\frac{h}{y_1}\right)$ can also be neglected and then Eq. 16.33 reduces to

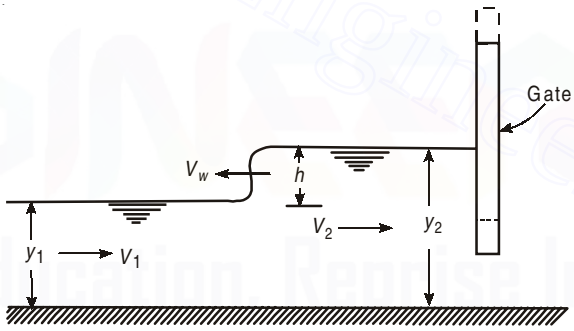
$$C = \sqrt{gy_1} \quad ...(16.38)$$

The above simplification was suggested by Lagrange and hence Eq. 16.38 is known as 'Lagrange's celerity equation'.

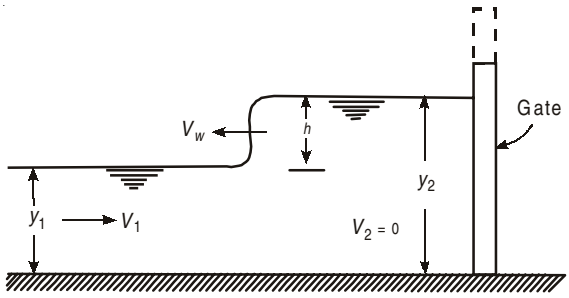
As indicated earlier a surge is often termed as moving hydraulic jump. However, in the case of a stationary hydraulic jump, the absolute velocity V_w is zero. Hence by substituting $V_w = 0$ in Eq. 16.30 and simplifying it, Eq. 16.16 may be obtained. As such a stationary hydraulic jump may be considered as a special case of a surge.

Case (b) Surge due to sudden reduction of flow

Consider a positive surge of type *B* created in a rectangular channel by sudden reduction of flow due to partial or complete closing of a gate. If V_w is the absolute velocity of the surge moving towards left



(a) Partial closure of the gate



(b) Total closure of the gate

Figure 16.13 *Positive surge of type B in a rectangular channel.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(Fig. 16.13 a) then in order to make it a case of steady flow apply a velocity V_w in the direction opposite to that of surge. The velocities at sections 1 and 2 will then be $(V_1 + V_w)$ and $(V_2 + V_w)$ respectively. The continuity equation in this case is

$$y_1(V_1 + V_w) = y_2(V_2 + V_w) \quad ...(16.39)$$

and the momentum equation is

$$\frac{1}{2}w\left(y_2^2 - y_1^2\right) = \frac{w}{g}\, y_2\,(V_2+V_w)\,[(V_1+V_w)-(V_2+V_w)] \quad ...(16.40)$$

If the flow is completely stopped (Fig. 16.13 b) the velocity $V_2 = 0$. The above equations will then be reduced to

$$y_1\,(V_1 + V_w) = y_2\,V_w \quad ...(16.39\text{ a})$$

and

$$\frac{1}{2}w\left(y_2^2 - y_1^2\right) = \frac{w}{g}\; y_2 \times V_w\, V_1 \quad ...(16.40\text{ a})$$

Eliminating V_w from Eqs 16.39 (a) and 16.40 (a) and simplifying, we have

$$V_1^2 = (y_2 - y_1)^2\, \frac{(y_1 + y_2)g}{2y_1y_2} \quad ...(16.41)$$

In this case the celerity of wave $C = (V_1 + V_w)$, and from Eq. 16.39 (a) and 16.41, we have

$$C = \sqrt{\frac{gy_2}{2y_1}(y_1 + y_2)} \quad ...(16.42)$$

Multiplying Eq. 16.41 by the square of Eq. 16.42 and simplifying, we have

$$V_1 = \frac{(y_2 - y_1)}{C}\left(\frac{y_2 + y_1}{2y_1}\right)g$$

or

$$V_1 = \frac{h}{C}\left(\frac{y_2 + y_1}{2y_1}\right)\, g \quad ...(16.43)$$

Thus the height of the surge is

$$h = \frac{C}{g}\left(\frac{2y_1}{y_1 + y_2}\right)V_1 \quad ...(16.44)$$

Again if the height of the surge is small retative to the depth of flow, then $y_1 \approx y_2$, and Eqs 16.43 and 16.44 may be approximately written as

$$V_1 = \frac{hg}{C} \quad ...(16.45)$$

and

$$h = \frac{V_1 C}{g} \quad ...(16.46)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Negative Surges

Negative surges are not stable in form because as indicated below the upper portions of the wave travel faster than the lower portions. As shown in Fig. 16.14 if y_2 is the depth below the top $a_1\ b_1$ of the wave and y_1 is the depth below the bottom or trough $c_1\ d_1$ of the wave, then since celerity $C = \sqrt{gy}$, its value at the top $a_1\ b_1$ of the wave is $C_2 = \sqrt{gy_2}$ and at the bottom $c_1\ d_1$ of the wave is $C_1 = \sqrt{gy_1}$; and C_2 is greater than C_1. Thus if the initial profile of the surge is assumed to have a steep front, it will soon flatten out as the surge moves through the channel. If the height of the surge is moderate or small compared with the depth of flow, the equations derived for a positive surge can be applied to determine approximately the propagation of the negative surge. However, if the height of the surge is relatively large, a more elaborate analysis as indicated below may be adopted.

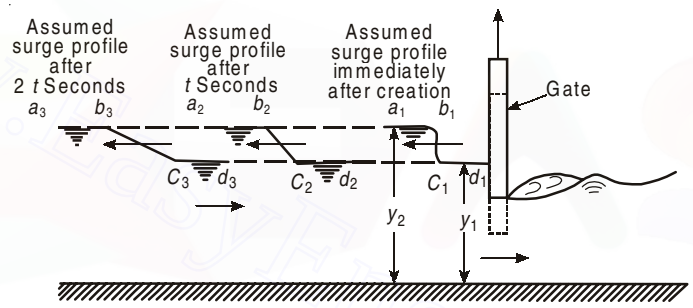


Figure 16.14 *Propagation of a negative surge due to sudden lift of a gate*

Figure 16.15 shows a negative surge of type *D* of relatively large height retreating in upstream direction in a rectangular channel. The surge is caused by the sudden lifting of a gate. In this case the velocity of the surge actually varies from point to point. Let V_w be the velocity of the surge at a point on its surface where the depth is y and the velocity of flow through the section is V. During a time interval dt, let the change in y be dy. The value of dy is positive for an increase of y and negative for a decrease of y. Assuming that the rate of change in y varies uniformly from O to dy in time dt, the average change is $\frac{dy}{2}$. The mass of water that moves per unit width of the channel is equal to $\frac{w}{g}\left(y+\frac{dy}{2}\right)\ [V_w-(-\mathrm{V})] = \frac{w}{g}\left(y+\frac{dy}{2}\right)(V_w+V)$, as V is in a direction opposite to V_w. If the change in velocity is dV, then the rate of change of momentum of the vertical element between y and $(y + dy)$ is equal to $\left[\frac{w}{g}\left(y+\frac{dy}{2}\right)(V_w+V)dV\right]$. By the momentum principle this rate of change of momentum is equal to the corresponding change in hydrostatic pressure which is equal to $\left[\frac{w}{2}y^2-\frac{w}{2}(y+dy)^2\right]$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

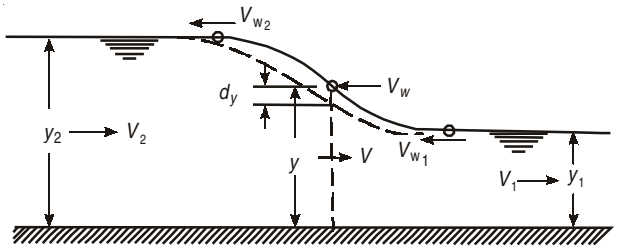


Figure 16.15 *Analysis of a negative surge*

Thus

$$\frac{w}{2}y^2 - \frac{w}{2}(y+dy)^2 = \frac{w}{g}\left(y+\frac{dy}{2}\right)(V_w+V)dV \qquad ...(16.47)$$

Simplifying the above equation and neglecting the differential terms of higher order, we get

$$dy = -\frac{(V+V_w)dV}{g} \qquad ...(16.48)$$

In this case the celerity of wave $C = V_w + V$, or

$$V_w = C - V$$

or
$$V_w = \sqrt{gy} - V \qquad ...(16.49)$$

Thus the velocity at the crest (or top) of the surge wave is

$$V_{w_2} = \sqrt{gy_2} - V_2 \qquad ...(16.50)$$

and at the trough (or bottom) of the surge wave is

$$V_{w_1} = \sqrt{gy_1} - V_1 \qquad ...(16.51)$$

When the surge is not too high, a straight-line relation between V_{w1} and V_{w2} may be assumed. Thus the mean velocity of the surge wave may be considered to be

$$\bar{V}_w = \frac{V_{w_1} + V_{w_2}}{2} \qquad ...(16.52)$$

Eliminating V_w between Eqs 16.48 and 16.49, we get

$$\frac{dy}{\sqrt{y}} = -\frac{dV}{\sqrt{g}} \qquad ...(16.53)$$

Integrating Eq. 16.53 from y_2 to y and V_2 to V, and solving for V, we get

$$V = V_2 + 2\sqrt{gy_2} - 2\sqrt{gy} \qquad ...(16.54)$$

From Eq. 16.49, we have

$$V_w = 3\sqrt{gy} - 2\sqrt{gy_2} - V_2 \qquad ...(16.55)$$

Thus the velocity V_{w1} at the trough (or bottom) of the surge wave is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_{w_1} = 3\sqrt{gy_1} - 2\sqrt{gy_2} - V_2 \quad ...(16.56)$$

The velocity V_{w2} at the crest (or top) of the surge wave is

$$V_{w_2} = \sqrt{gy_2} - V_2$$

which is same as given by Eq. 16.50.

Further let t be the time elapsed since the surge was created or, in this case since the gate was opened. At $t = 0$, the wavelength $\lambda = 0$. After t seconds the wavelength is equal to

$$\lambda = (V_{w_2} - V_{w_1})\, t \quad ...(16.57)$$

The above analysis can also be similarly applied to a negative surge of type *C*.

ILLUSTRATIVE EXAMPLES

Example 16.1. *A rectangular channel 7.5 m wide has a uniform depth of flow of 2.0 m and has a bed slope of 1 in 3000. If due to weir constructed at the downstream end of the channel, water surface at a section is raised by 0.75 m, determine the water surface slope with respect to horizontal at this section. Assume Manning's n = 0.02.*

Solution

According to Manning's formula, we have

$$Q = \frac{1}{n} AR^{2/3} S_0^{1/2}$$

or
$$Q = \frac{1}{0.02} (7.5 \times 2.0) \left[\frac{(7.5 \times 2.0)}{7.5 + (2 \times 2.0)}\right]^{2/3} \left(\frac{1}{3000}\right)^{1/2}$$

or
$$Q = 16.35 \text{ m}^3/\text{s}$$

From Eq. 16.1, we have

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2 T}{gA^3}}$$

$$S_0 = \frac{1}{3000} = 3.333 \times 10^{-4}$$

$$S_f = \frac{n^2 V^2}{R^{4/3}}$$

For
$$y = (2.0 + 0.75) = 2.75 \text{ m}$$
$$A = 20.625 \text{ m}^2$$
$$P = 13 \text{ m}$$
$$R = 1.5865 \text{ m}$$
$$V = 0.793 \text{ m/sec}$$

$\therefore$
$$S_f = 1.359\ 10^{-4}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{Q^2T}{gA^3} = \frac{(16.35)^2 \times 7.5}{9.81 \times (20.625)^3}$$

$$= 0.0233$$

Thus $$\frac{dy}{dx} = \frac{(3.333 - 1.359) \times 10^{-4}}{(1 - 0.0233)}$$

$$= 2.021 \times 10^{-4}$$

The water surface slope S_w with respect to horizontal is given by

$$S_w = S_0 - \frac{dy}{dx} = (3.333 - 2.021) \times 10^{-4}$$

$$= 1.312 \times 10^{-4}.$$

Example 16.2. *A rectangular channel 10 m wide carries a discharge of 30 m^3/s. It is laid at a slope of 0.0001. If at a section in this channel the depth is 1.6 m, how for (upstream or downstream) from the section will the depth be 2.0 m? Take Manning's n as 0.015.*

Solution

If y_n is the normal depth of flow then

$$A = 10y_n$$

and $$R = \frac{10y_n}{10 + 2y_n}$$

The velocity of flow is

$$V = \frac{30}{10y_n} = \frac{3}{y_n}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

or $$\frac{3}{y_n} = \frac{1}{0.015} \times \left(\frac{10y_n}{10 + 2y_n} \right)^{2/3} \times (0.0001)^{1/2}$$

or $$1.909(5 + y_n) = y_n^{5/2}$$

Solving the equation for y_n by trial

$$y_n = 2.97 \text{ m}$$

The critical depth y_c is given by

$$y_c = \left[\frac{q^2}{g} \right]^{1/3} = \left[\frac{(30/10)^2}{9.81} \right]^{1/3} = 0.972 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $y_n > y_c$, it is a mild-sloped channel. Further the given depths of 1.6 m and 2.0 m are between y_n and y_c hence the surface profile lies in zone (2) or the surface profile will be M_2 type curve. As such the depth of 2.0 m will be found upstream from that of 1.6 m. For the computation of the length of the surface profile the various steps are shown in Table below.

Table for Example 16.2

y m	A m^2	R m	V m/s	$\frac{V^2}{2g}$ m	E m	ΔE m	S_f	$\bar{S}_f$	$S_0 - \bar{S}_f$	Δx
1.6	16	1.21	1.88	0.179	1.779		0.000617			
						0.080		0.000562	0.000462	173.16
1.7	17	1.27	1.76	0.159	1.859		0.000507			
						0.083		0.000470	0.000370	224.32
1.8	18	1.32	1.67	0.142	1.942		0.000433			
						0.085		0.000400	0.000300	283.33
1.9	19	1.38	1.58	0.127	2.027		0.000366			
						0.088		0.000340	0.000240	366.67
2.0	20	1.43	1.50	0.115	2.115		0.000314			

The required distance between the two sections is

$$\Sigma(\Delta x) = (173.16 + 224.32 + 283.33 + 366.67) = 1\,047.48 \text{ m}$$

Example 16.3. *A trapezoidal channel having bottom width 6 m, side slope 2 horizontal to 1 vertical, Manning's roughness coefficient 0.025, and bottom slope 0.0016, carries a discharge of 10 m^3/s. Compute the back water profile created by a dam which backs up the water to a depth of 2.0 m immediately behind the dam. Use the direct step method for computation.*

Solution

If y_n is the normal depth of flow, then according to Manning's formula

$$10 = \frac{1}{0.025}(6 + 2y_n)\, y_n \times \left[\frac{(6 + 2y_n)y_n}{6 + (2\sqrt{5})y_n}\right]^{2/3} (0.0016)^{1/2}$$

or $$16.57 + 12.35y_n = [(3 + yn)y_n]^{5/2}$$

Solving by trial,

$$y_n = 0.963 \text{ m}$$

Further if y_c is the critical depth then

$$\frac{(10)^2}{9.81} = \frac{[(6 + 2y_c)y_c]^3}{6 + 4y_c}$$

or $$10.19\,(6 + 4y_c) = [(6 + 2y_c)y_c]^3$$

Solving by trial,

$$y_c = 0.611 \text{ m.}$$

Since in this case y_n is greater than y_c it is a mild sloped channel and hence the profile is of the M_1

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

type. For the computation of the length of the surface profile the various steps are shown in Table below

TABLE FOR EXAMPLE 16.3

y m	A m^2	R m	V m/s	$\frac{V^2}{2g}$ m	E m	ΔE m	S_f	$\bar{S}_f$	$S_0 - \bar{S}_f$	Δx
2.0	20.00	1.34	0.50	0.013	2.013		0.106×10^{-3}			
						0.196		0.133×10^{-3}	1.467×10^{-3}	133.61
1.80	17.28	1.23	0.58	0.017	1.817		0.160×10^{-3}			
						0.193		0.204×10^{-3}	1.396×10^{-3}	138.25
1.60	14.72	1.12	0.68	0.024	1.624		0.248×10^{-3}			
						0.191		0.329×10^{-3}	1.271×10^{-3}	150.28
1.40	12.32	1.00	0.81	0.033	1.433		0.410×10^{-3}			
						0.183		0.563×10^{-3}	1.037×10^{-3}	176.47
1.20	10.08	0.89	0.99	0.050	1.250		0.716×10^{-3}			
						0.170		1.062×10^{-3}	0.538×10^{-3}	315.99
1.00	8.00	0.76	1.25	0.080	1.080		1.408×10^{-3}			
						0.024		1.493×10^{-3}	0.107×10^{-3}	224.30
0.97	7.70	0.74	1.30	0.086	1.056		1.578×10^{-3}			

The length of the water surface profile is

$$\Sigma(\Delta x) = 1138.90 \text{ m.}$$

Note. The upstream end of the water surface profile is assumed at a depth equal to 1% greater than the normal depth and hence the depth at the upstream end of the profile has been considered as $(1.01 \times 0.963) = 0.97$ m.

Example 16.4. *Solve example 16.3 by Bresse's method.*

Solution

According to Bresse's method the length of the surface profile is given by Eq. 16.13 as

$$x = \frac{y_n}{S_0}\left[(u_2 - u_1) - \left(1 - \frac{C^2 S_0}{g}\right)\{\phi(u_2) - \phi(u_1)\}\right]$$

where $\phi(u) = \int \frac{du}{1-u^3}$ and $u = \left(\frac{y}{y_n}\right)$

$y_n = 0.963$ m ; and $y_c = 0.611$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The depth at the upstream end of the profile is (1.01× 0.963) = 0.97 m and at the downstream end the depth is 2.0 m.

As the depth of flow varies from 0.97 m to 2.0 m, an average value of $y = 1.485$ m may be assumed for the evaluation of Chezy's C.

Chezy's C may be expressed as

$$C = \frac{1}{n} R^{1/6}$$

For $y = 1.485$ m; $R = 1.054$ m

Thus $$C = \frac{1}{0.025}(1.054)^{1/6} = 40.35$$

Hence $$\frac{C^2 S_0}{g} = \frac{(40.35)^2(0.0016)}{9.81} = 0.266$$

and $$\left(1 - \frac{C^2 S_0}{g}\right) = 0.734$$

The values of u and $\phi(u)$ may be computed for the two extreme sections of the profile and the same are given in the following table.

y	u	$\phi(u)$
2.00	2.08	0.122
0.97	1.01	1.419

The values of the varied flow function $\phi(u)$ as given in the third column of the above table may be computed either by Eq. 16.12 or the same may be directly obtained from the Bakhmeteff's varied flow function tables.

By substituting the values in the above expression

$$x = \frac{0.963}{0.0016}[(2.08 - 1.01) - 0.734\,(0.122 - 1.419)]$$

or $$x = 1217 \text{ m}.$$

The length of the water surface profile obtained by the Bresse's method is about 7% more than the value obtained in Example 16.3. This is so because Bresse's method is derived primarily for a wide rectangular channel. As such when this method is applied to channels of other cross-sectional shapes, the solution is quite approximate.

Example 16.5. *A rectangular flume 2 m wide carries discharge at the rate of 2 m^3/s. The bed slope of the flume is 0.0004. At a certain section the depth of flow is 1 m. Calculate the distance of the section downstream where the depth of flow is 0.9 m. Solve by single step method. Assume rugosity coefficient as 0.014.*

Is the slope of the channel mild or steep? How is this type of surface profile classified?

Solution

If y_n is the normal depth of flow, then according to Manning's formula

$$2 = \frac{1}{0.014} \times (2 \times y_n) \times \left(\frac{2y_n}{2 + 2y_n}\right)^{2/3} (0.0004)^{2/3}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $0.5857\,(1 + y_n) = y_n^{5/2}$

Solving by trial

$$y_n = 1.082 \text{ m}$$

Further if y_c is the critical depth, then

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(2/2)^2}{9.81}\right]^{1/3} = 0.467 \text{ m.}$$

Since in this case y_n is greater than y_c, the bed slope of the flume is mild. Further the given depths of flow viz., 1.0 m and 0.90 m are between y_n and y_c, the surface profile lies in zone (2) and it is M_2 type curve.

The length of the surface profile between the sections where the depths of flow are 1.0 m and 0.90 m, may be computed by step method as follows.

At section 1 where the depth of flow is 1 m, the specific energy is

$$E_1 = 1.0 + \left[\frac{\{2/(2\times1)\}^2}{2\times9.81}\right] = 1.051 \text{ N.m/N}$$

Similarly at section 2 where the depth of flow is 0.90 m, the specific energy is

$$E_2 = 0.90 + \left[\frac{\{2/(2\times0.9)^2}{2\times9.81}\right] = 0.963 \text{ N.m/N}$$

$$\therefore \quad \Delta E = (E_2 - E_1) = (0.963 - 1.051)$$
$$= -0.088 \text{ N.m/N}$$

According to Manning's formula at section 1

$$S_{f1} = \frac{V^2 n^2}{R^{4/3}} = \frac{\{2/(2\times1)\}^2 \times (0.014)^2}{\left(\frac{2\times1}{2+2\times1}\right)^{4/3}} = 4.94 \times 10^{-4}$$

Similarly at section 2

$$S_{f2} = \frac{\{2/(2\times0.9)\}^2 \times (0.014)^2}{\left(\frac{2\times0.90}{2+2\times0.90}\right)^{4/3}} = 6.55 \times 10^{-4}$$

$$\therefore \quad \bar{S}_f = \frac{S_{f_1} + S_{f_2}}{2} = \frac{(4.94\times10^{-4} + 6.55\times10^{-4})}{2}$$
$$= 5.745 \times 10^{-4}$$

From Eq. 16.8 the distance between sections 1 and 2 is

$$x = \frac{\Delta E}{S_0 - \bar{S}_f} = \frac{-0.088}{(4\times10^{-4} - 5.745\times10^{-4})}$$
$$= 504.3 \text{ m.}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 16.6. *Water flows from an under sluice into a very wide rectangular channel. The channel has a bed slope of 1 in 1000. The sluice is regulated to discharge 6 m^3/s per m width of channel, the depth at venacontracta being 0.5 m. Will a hydraulic jump form? If so determine its location. Use a single step for the computation with Manning's n = 0.015.*

Solution

For a wide rectangular channel $R \approx y$

Thus if y_n is the normal depth of flow then according to Manning's formula

$$\frac{6}{y_n} = \frac{1}{0.015} \times y_n^{2/3} \times \left(\frac{1}{1000}\right)^{1/2}$$

or $$y_n = 1.873 \text{ m}$$

The critical depth y_c is given by

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left(\frac{36}{9.81}\right)^{1/3} = 1.542 \text{ m}$$

Since in this case $y_n > y_c$, the bed slope of the channel is mild. Further the depth of flow at vena contracta is 0.5 m, which is less than the critical depth. As such in this case hydraulic jump will form. The jump will be located at a section where the depth of flow will be conjugate to the normal depth of flow y_n equal to 1.873 m. For this the depth of flow will increase from 0.5 m at vena contracta to this conjugate depth along M_3 type surface profile.

From Eq. 16.20 the depth y_1 conjugate to the normal depth y_n equal to 1.873 m is obtained as

$$\frac{y_1}{1.873} = \frac{1}{2}\left[-1+\sqrt{1+\frac{8\times(6)^2}{9.81\times(1.873)^3}}\right]$$

or $$y_1 = 1.253 \text{ m}$$

The distance between vena contracta and the section where the jump will be located may be obtained as follows.

At vena contracta section the specific energy is

$$E_1 = 0.50 + \frac{(6/0.50)^2}{2\times 9.81} = 7.839 \text{ N.m/N}$$

Similarly at the section when the depth of flow is 1.253 m, the specific energy is

$$E_2 = 1.253 + \frac{(6/1.253)^2}{2\times 9.81} = 2.422 \text{ N.m/N}$$

$$\Delta E = (E_2 - E_1) = (2.422 - 7.839)$$
$$= -5.417 \text{ N.m/N}$$

According to Manning's formula at vena contracta section

$$S_{f1} = \frac{V^2 n^2}{R^{4/3}} = \frac{(6/0.50)^2 \times (0.015)^2}{(0.50)^{4/3}} = 8.164 \times 10^{-2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Similarly at the section where the depth of flow is 1.253 m

$$S_{f2} = \frac{(6/1.253)^2 \times (0.015)^2}{(1.253)^{4/3}} = 0.382 \times 10^{-2}$$

$$\therefore \quad \bar{S}_f = \frac{S_{f_1} + S_{f_2}}{2} = \frac{(8.164 + 0.382) \times 10^{-2}}{2}$$

$$= 4.273 \times 10^{-2}$$

From Eq. 16.8 the distance between the vena contracta section and the section where the jump will be located is

$$x = \frac{\Delta E}{S_0 - \bar{S}_f} = \frac{-5.417}{(0.10 \times 10^{-2} - 4.273 \times 10^{-2})}$$

$$= 130 \text{ m}$$

Note. It has been assumed that there exists a long reach of the channel with same slope on the downstream side of the sluice.

Example 16.7. *A horizontal rectangular channel 4 m wide carries a discharge of 16 m^3/s. Determine whether a jump may occur at an initial depth of 0.5 m or not. If a jump occurs, determine the sequent depth to this initial depth. Also determine the energy loss in the jump.*

Solution

The jump may occur at an initial depth of 0.5 m if the initial Froude number Fr_1 is greater than one.

$$Fr_1 = \frac{V_1}{\sqrt{g y_1}} = \frac{16}{(4 \times 0.5)\sqrt{9.81 \times 0.5}} = 3.61$$

Since $Fr_1 > 1$, the jump may occur at an initial depth of 0.5 m. From Eq. 16.23

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + 8Fr_1^2}\right]$$

or $$\frac{y_2}{0.5} = \frac{1}{2}\left[-1 + \sqrt{1 + 8(3.61)^2}\right]$$

or $$y_2 = 2.32 \text{ m}$$

From Eq. 16.25 the energy loss in the jump is given as

$$\Delta E = \frac{(y_2 - y_1)^3}{4 y_1 y_2} = \frac{(2.32 - 0.5)^3}{4(2.32 \times 0.5)} = 1.30 \text{ m.}$$

Example 16.8. *In a rectangular channel there occurs a jump corresponding to $Fr_1 = 2.5$. Determine the critical depth and head loss in terms of the initial depth y_1.*

Solution

For a rectangular channel

$$Fr_1^2 = \frac{q^2}{g y_1^3} = \left(\frac{y_c}{y_1}\right)^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$(2.5)^2 = \left(\frac{y_c}{y_1}\right)^3$$

or $$y_c = (2.5)^{2/3} y_1 = 1.84\, y_1$$

From Eq. 16.23

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1+\sqrt{1+8(2.5)^2}\right] = 3.07$$

From Eq. 16.25

$$\frac{\Delta E}{y_1} = \frac{[(y_2/y_1)-1]^3}{4(y_2/y_1)} = \frac{(2.07)^3}{4\times(3.07)} = 0.722$$

or $$\Delta E = 0.722\, y_1.$$

Example 16.9. *A trapezoidal channel having bottom width 8 m and side slope 1:1, carries a discharge of 80 m^3/s. Find the depth conjugate to initial depth of 0.75 m before the jump. Also determine the loss of energy in the jump.*

Solution

The area of flow section corresponding to $y_1 = 0.75$ m is

$$A_1 = (B + zy_1)y_1 = [8 + (0.75 \times 1)]0.75 = 6.56 \text{ m}^2$$

and $$\bar{z}_1 = \frac{y_1}{6}\frac{(3B+2zy_1)}{(B+zy_1)} = \frac{0.75(24+1.5)}{6(8+0.75)} = 0.364 \text{ m}$$

If y_2 is the conjugate depth after the jump then the corresponding values of A_2 and $\bar{z}_2$ are given by

$$A_2 = (8 + y_2)y_2$$

and $$\bar{z}_2 = \frac{y_2}{6}\frac{(24+2y_2)}{(8+y_2)}$$

By substituting the above obtained values in Eq. 16.14, it becomes

$$\frac{(80)^2}{(9.81\times6.56)} + (6.56 \times 0.364) = \frac{(80)^2}{9.81(8+y_2)y_2} + \frac{y_2^2}{6} \times (24 + 2y_2)$$

or $$101.84 = \frac{652.4}{(8+y_2)y_2} + \frac{y_2^2}{3}(12+y_2)$$

Solving the above equation for y_2 by trial and error

$$y_2 = 4.066 \text{ m}$$

The specific energies before and after the jump are

$$E_1 = y_1 + \frac{Q^2}{2gA_1^2} = 0.75 + \frac{(80)^2}{2\times9.81\times(6.56)^2}$$

$$= 8.33 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$E_2 = y_2 + \frac{Q^2}{2gA_2^2} = 4.066 + \frac{(80)^2}{2\times 9.81\times(49.06)^2}$$

$$= 4.20 \text{ m}$$

Thus, the loss of energy in the jump is

$$\Delta E = (E_1 - E_2)$$

$$= (8.33 - 4.20) = 4.13 \text{ m}.$$

Example 16.10. *A hydraulic jump occurs in a 90° triangular channel. Derive an equation relating the two depths and the flow rate. If the depths before and after the jump in the above channel are 0.5 m and 1.0 m, determine the flowrate and obtain the Froude numbers before and after the jump.*

Solution

From Eq. 16.14, we have

$$\frac{Q^2}{gA_1} + A_1\bar{z}_1 = \frac{Q^2}{gA_2} + A_2\bar{z}_2$$

If y_1 and y_2 are the depths before and after the jump, then in this case

$$A_1 = y_1^2;\ \bar{z}_1 = (y_1/3);\quad A_2 = y_2^2;\ \text{and}\ \bar{z}_2 = (y_2/3)$$

Thus by substitution, we get

$$\frac{Q^2}{gy_1^2} + y_1^2\times\frac{y_1}{3} = \frac{Q^2}{gy_2^2} + y_2^2\times\frac{y_2}{3}$$

or
$$\frac{Q^2}{g}\left(\frac{1}{y_1^2} - \frac{1}{y_2^2}\right) = \frac{1}{3}\left(y_2^3 - y_1^3\right)$$

or
$$\frac{Q^2}{g}\left(\frac{y_2^2 - y_1^2}{y_1^2y_2^2}\right) = \frac{1}{3}\left(y_2^3 - y_1^3\right)$$

or
$$\frac{Q^2}{gy_1^5}\left[\frac{(y_2/y_1)^2 - 1}{(y_2/y_1)^2}\right] = \frac{1}{3}\left[(y_2/y_1)^3 - 1\right] \quad ...(i)$$

But $\frac{Q^2}{gy_1^5} = \frac{V_1^2}{gy_1} = Fr_1^2$; and if $(y_2/y_1) = r$; then, we have

$$3Fr_1^2 = \frac{r^2(r^3-1)}{(r^2-1)} \quad ...(ii)$$

The depths before and after the jump are given as

$y_1 = 0.5$ m; and $y_2 = 1.0$ m

$\therefore \quad (y_2/y_1) = 2$

Introducing these values in Eq. (i), we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{Q^2}{9.81\times(0.5)^5}\left[\frac{(2)^2-1}{(2)^2}\right] = \frac{1}{3}\left[(2)^3-1\right]$$

$$\therefore \quad Q = 0.977 \text{ m}^3/\text{s}$$

Froude number before the jump is given by

$$Fr_1 = \frac{Q}{\sqrt{gy_1^5}}$$

$$= \frac{0.977}{\sqrt{9.81\times(0.5)^5}} = 1.764$$

Alternatively the value of F_{r_1} may be obtained by Eq. (ii) as

$$3Fr_1^2 = \frac{(2)^2(2^3-1)}{(2^2-1)}$$

$$\therefore \quad Fr_1 = 1.764$$

Froude number after the jump is given by

$$Fr_2 = \frac{Q}{\sqrt{gy_2^5}}$$

$$= \frac{0.977}{\sqrt{9.81\times(1)^5}} = 0.312$$

Example 16.11. *A rectangular channel carries a discharge of 2 m^3/s per metre width. If the loss of energy in the hydraulic jump is found to be 2.75 m, determine the conjugate depths before and after the jump.*

Solution

From Eq. 16.25 the loss of energy in the jump may be expressed as

$$\Delta E = \frac{(y_2-y_1)^3}{4y_1y_2}$$

or

$$\frac{\Delta E}{y_1} = \frac{\left[\left(\frac{y_2}{y_1}\right)-1\right]^3}{4\left(\frac{y_2}{y_1}\right)} = \frac{(r-1)^3}{4r}$$

where

$$\left(\frac{y_2}{y_1}\right) = r$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substitution

$$\frac{2.75}{y_1} = \frac{(r-1)^3}{4r} \quad \text{...(i)}$$

Further from Eq. 16.19, we have

$$\frac{y_2}{y_1} = \frac{1}{2}\left[1+\sqrt{1+\frac{8q^2}{gy_1^3}}\right]$$

or $$r = \frac{1}{2}\left[-1+\left\{1+\frac{8(2)^2}{9.81y_1^3}\right\}^{1/2}\right]$$

or $$r = \frac{1}{2}\left[-1+\left(1+\frac{3.26}{y_1^3}\right)^{1/2}\right] \quad \text{...(ii)}$$

From Eqs (i) and (ii) the values of y_1 and r can be obtained by trial and error. Thus we have

$$y_1 = 0.214 \text{ m}, r = 8.64, \text{ and } y_2 = 1.849 \text{ m}$$

∴ Conjugate depths are 0.214 m and 1.849 m.

Example 16.12. *Show that the head loss in a hydraulic jump formed in a rectangular channel may be expressed as*

$$\Delta E = \frac{(V_1 - V_2)^3}{2g(V_1+V_2)}$$

Solution

The head loss in a hydraulic jump may be expressed as

$$\Delta E = \left(y_1+\frac{V_1^2}{2g}\right)-\left(y_2+\frac{V_2^2}{2g}\right)$$

or $$\Delta E = \left(\frac{V_1^2-V_2^2}{2g}\right)-(y_2-y_1)$$

If q is the discharge per unit width of the channel, then

$$q = y_1V_1 = y_2V_2$$

or $$y_1 = \frac{q}{V_1} \text{ and } y_2 = \frac{q}{V_2}$$

By substitution, we get

$$\Delta E = \left(\frac{V_1^2-V_2^2}{2g}\right)-\left(\frac{q}{V_2}-\frac{q}{V_1}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\Delta E = \left(\frac{V_1^2 - V_2^2}{2g}\right) - q\left(\frac{V_1 - V_2}{V_1 V_2}\right) \quad ...(i)$$

From Eq. 16.16, we have

$$\frac{2q^2}{g} = y_1 y_2 (y_1 + y_2)$$

or $$\frac{2q^2}{g} = \frac{q^2}{V_1 V_2}\left(\frac{q}{V_1} + \frac{q}{V_2}\right)$$

or $$q = \frac{2(V_1 V_2)^2}{g(V_1 + V_2)} \quad ...(ii)$$

Introducing the value of q from Eq. (ii) in Eq. (i), we get

$$\Delta E = \left(\frac{V_1^2 - V_2^2}{2g}\right) - \frac{2V_1 V_2 (V_1 - V_2)}{g(V_1 + V_2)},$$

or $$\Delta E = \frac{(V_1^2 - V_2^2)(V_1 + V_2) - 4V_1 V_2 (V_1 - V_2)}{2g(V_1 + V_2)},$$

or $$\Delta E = \frac{(V_1 - V_2)^3}{2g(V_1 + V_2)}$$

Example 16.13. *A concrete lined trapezoidal irrigation canal has a bottom width of 10 m, side slopes of 1h : 1V and longitudinal bottom slope of 0.0005. If the channel is several kilometres long, what is the flow depth near the downstream end for a flow of 60 m^3/s under free fall condition ?*

Solution

Let y be the normal depth of flow.

Then

$$A = (B + zy)y = (10 + y)y$$

and $$P = B + 2y\sqrt{1 + 2^2} = \left(10 + 2y\sqrt{2}\right)$$

$$R = \frac{A}{P} = \frac{(10 + y)y}{\left(10 + 2y\sqrt{2}\right)}$$

For concrete lined channel value of Manning's n may be taken as 0.012. Using Manning's formula, we have

$$\frac{60}{(10 + y)y} = \frac{1}{0.012} \times \left[\frac{(10 + y)y}{\left(10 + 2y\sqrt{2}\right)}\right]^{2/3} \times (0.0005)^{1/2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $1827.14 + 516.79y = [(10 + y)y]^{5/2}$

Solving this equation for y by trial, we get

$$y = 2.0108 \text{ m}$$

Let y_c be the critical depth.

For critical flow, we have

$$\frac{Q^2}{g} = \left[\frac{A^3}{T}\right]_c$$

$$\frac{(60)^2}{9.81} = \frac{\left[(10 + y_c)y_c\right]^3}{(10 + 2y_c)}$$

or $366.97(10 + 2y_c) = [(10 + y_c)y_c]^3$

Solving this equation for y_c by trial, we get

$$y_c = 1.4657 \text{ m.}$$

Since normal depth $y >$ critical depth y_c, the flow is subcritical, and hence the depth of flow near the downstream end for free fall condition is equal to

$$y_c = 1.4657 \text{ m}$$

Example 16.14. *Water discharges at the rate of 298 m^3/s over a spillway 14 m wide into a stilling basin of the same width. The water level behind the spillway is 61 m over datum. The river water level in the downstream is 29.2 m over datum. The river bed is paved flush with the bed level to just adequate length to form the jump basin. Assuming that no energy is dissipated in the flow down the spillway, find the basin invert level required for a hydraulic jump to be formed within the basin.*

Solution

Let the basin invert be at a height z above the datum.

Further let y_1, y_2 and V_1, V_2 be the depth of flow and velocities of flow before and after the hydraulic jump respectively.

Then

$$V_1 = \frac{298}{14 \times y_1} = \frac{21.286}{y_1}$$

and

$$V_2 = \frac{298}{14 \times y_2} = \frac{21.286}{y_2}$$

If q is the discharge per unit width of the spillway, then

$$q = \frac{298}{14} = 21.286 \text{ m}^3/\text{s per m}$$

Applying Bernoulli's equation between the section behind the spillway where the hydraulic jump begins, and assuming that no energy is dissipated in the flow down the spillway, we get

$$61 = z + y_1 + \frac{V_1^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$61 = z + y_1 + \frac{(21.286)^2}{2g \times y_1^2}$$

or $$61 = z + y_1 + \frac{23.093}{y_1^2} \quad \text{...(i)}$$

The loss of energy in a hydraulic jump is given by Eq. 16.25 as

$$\Delta E = \frac{(y_2 - y_1)^3}{4y_1 y_2}$$

Thus applying Bernoulli's equation between the section behind the spillway and the section downstream of the hydraulic jump, we get

$$61 = 29.2 + \frac{V_2^2}{2g} + \frac{(y_2 - y_1)^3}{4y_1 y_2}$$

or $$61 = 29.2 + \frac{(21.286)^2}{2g \times y_2^2} + \frac{(y_2 - y_1)^3}{4y_1 y_2}$$

or $$31.8 = \frac{23.0.93}{(y_2 / y_1)^2 \times y_1^2} + \frac{\left[(y_2 / y_1) - 1\right]^3 \times y_1}{4(y_2 / y_1)}$$

or $$31.8 = \frac{23.093}{r^2 \times y_1^2} + \frac{(r-1)^3 \times y_1}{4r} \quad \text{...(ii)}$$

where $(y_2 / y_1) = r$

Further for hydraulic jump in a rectangular channel from Eq. 16.16, we have

$$\frac{2q^2}{g} = y_1 y_2 (y_1 + y_2)$$

or $$\frac{2 \times (21.286)^2}{g} = y_1 y_2 (y_1 + y_2)$$

or $$\frac{92.374}{y_1^3} = (y_2 / y_1)[(y_2 / y_1) + 1]$$

or $$\frac{92.374}{y_1^3} = (y_2 / y_1)\,[(y_2 / y_1) + 1)$$

or $$\frac{92.374}{y_1^3} = r\,(r + 1) \quad \text{...(iii)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equations (i), (ii) and (iii) are solved simultaneously by trial and error with the following steps.

1. Assume value of y_1 and obtain the value of r from Eq. (iii)
2. Repeat the trial till the values of $y_1$and r obtained in step 1 satisfy Eq. (ii).
3. Compute the value of z from Eq. (i).
4. As an alternative check, from the values of y_1 and r obtained in step 2, compute the value of y_2, and $(y_2 + z) = 29.2$.

The results obtained in this case are as follows :

$y_1 = 0.7429$ m ; $r = 14.5183$; $y_2 = 10.786$ m

$z = 18.414$ m ; and $(y_2 + z) = (10.786 + 18.414) = 29.2$ m.

Thus the basin invert level should be at a height of 18.414 m above the datum.

Example 16.15. *The depth and velocity of flow in a rectangular channel are 0.9 m and 1.5 m/s respectively. If a gate at the downstream end of the channel is abruptly closed, what will be the height and absolute velocity of the resulting surge? If the channel is 900 m long, how much time will be required for the surge to reach the upstream end of the channel?*

Solution

By closing the gate at the downstream end of the channel, a positive surge of type *B* is created. Thus from Eq. 16.39 (*a*), we have

$$y_1 (V_1 + V_w) = y_2 V_w$$

or $$0.9 (1.5 + V_w) = y_2 V_w \quad \text{...(i)}$$

Also from Eq. 16.40 (a), we have

$$\frac{1}{2}(y_2^2 - y_1^2) = \frac{y_2}{g} \times V_w V_1$$

or $$\frac{1}{2}\left(y_2^2 - 0.9^2\right) = \frac{y_2}{9.81} \times V_w \times 1.5 \quad \text{...(ii)}$$

Eliminating V_w between Eqs (i) and (ii) and simplifying, we get

$$y_2^3 - 0.9y_2^2 - 1.223y_2 + 0.729 = 0$$

Solving this Eq. by trial and error, we get

$$y_2 = 1.401 \text{ m}$$

From Eq. (i), we have

$$V_w = \frac{1.35}{y_2 - 0.9} = \frac{1.35}{1.401 - 0.9}$$

$$\therefore \quad V_w = 2.695 \text{ m/s}$$

Time required for the surge to reach the upstream end of the channel

$$= \frac{900}{2.695} = 334 \text{ s} = 5.57 \text{ min.}$$

Example 16.16. *The depth and velocity of flow in a rectangular channel are 1 m and 1.5 m/s respectively. If the rate of inflow at the upstream end is suddenly doubled, what will be the height and absolute velocity of the resulting surge and the celerity of the wave?*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

$$y_1 = 1 \text{ m} ; V_1 = 1.5 \text{ m/s}$$

$\therefore$ $$q1 = 1 \times 1.5 = 1.5 \text{ m}^3/\text{s per m}$$

When the discharge is doubled

$$q_2 = 1.5 \times 2 = 3.0 \text{ m}^3/\text{s per m}$$

But $$q_2 = y_2 \times V_2$$

$\therefore$ $$y_2 V_2 = 3$$

In this case a positive surge of type *A* will be created. Thus from Eq. 16.27, we have

$$V_w = \frac{V_1 y_1 V_2 y_2}{y_1 - y_2}$$

or $$V_w = \frac{(1 \times 1.5) - 3.0}{1 - y_2}$$

or $$V_w = \frac{1.5}{y_2 - 1}$$

Also from Eq. 16.31, we have

$$V_w = \left[\sqrt{\frac{g y_2}{2 y_1}(y_2 + y_1)}\right] + V_1$$

or $$\frac{1.5}{y_2 - 1} = \left[\sqrt{\frac{9.81 \times y_2}{2 \times 1}(y_2 + 1)}\right] + 1.5$$

or $$\frac{1.5}{y_2 - 1} - 1.5 = \sqrt{4.905 y_2 (y_2 + 1)}$$

or $$\frac{3 - 1.5 y_2}{y_2 - 1} = \sqrt{4.905 y_2 (y_2 + 1)}$$

Solving the above equation by trial and error, we get

$$y_2 = 1.284 \text{ m}$$

$\therefore$ Height of the surge

$$h = (y_2 - y_1)$$
$$= (1.284 - 1.0) = 0.284 \text{ m}$$

Absolute velocity of the surge

$$V_w = \frac{1.5}{1.284 - 1} = 5.282 \text{ m/s}$$

Celerity of the wave

$$C = (V_w - V_1)$$
$$= (5.282 - 1.5) = 3.782 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 16.17. *A horizontal rectangular channel 3 m wide and 2 m water depth conveys water at 18 m^3/s. If the flow rate is reduced to 2/3 of its original value, compute the magnitude and speed of the upstream surge.*

Assume that the front of the surge is rectangular and friction in the channel is neglected.

Solution

Due to reduction in the flow rate (which would be achieved by closing the gate) a positive surge of type *B* would be created.

Thus from Eq. 16.39, we have

$$y_1(V_1 + V_w) = y_2(V_2 + V_w)$$

$$y_1 = 2\text{ m}\,;\ V_1 = \frac{18}{3\times 2} = 3\text{ m/s}$$

$$V_2 = \frac{2}{3} \times \frac{18}{3\times y_2} = \frac{4}{y_2}$$

Thus by substitution, we get

$$2(3 + V_w) = y_2\left(\frac{4}{y_2} + V_w\right)$$

or $$6 + 2V_w = 4 + y_2 V_w$$

or $$V_w(y_2 - 2) = 2 \qquad \text{...(i)}$$

Further from Eq. 16.40, we have

$$\frac{1}{2}w(y_2^2 - y_1^2) = \frac{w}{g}y_2(V_2 + V_w)[(V_1 + V_w) - (V_2 + V_w)]$$

or $$\frac{1}{2}(y_2^2 - 4) = \frac{1}{g}y_2\left(\frac{4}{y_2} + V_w\right)\left[(3 + V_w) - \left(\frac{4}{y_2} + V_w\right)\right]$$

or $$\frac{1}{2}(y_2^2 - 4) = \frac{1}{9.81}y_2\left(\frac{4}{y_2} + V_w\right)\left(3 - \frac{4}{y_2}\right) \qquad \text{...(ii)}$$

Eliminating V_w between Eqs (i) and (ii) and simplifying we get

$y_2^4 - 2y_2^3 - 7.669y_2^2 + 1.786\,y_2 - 6.524 = 0$

Solving this equation by trial and error, we get

$$y_2 = 3.939\text{ m}$$

∴ Height of surge above the original water surface,

$$h = (3.939 - 2) = 1.939\text{ m}$$

Further from Eq. (i), we have

$$V_w = \frac{2}{(y_2 - 2)} = \frac{2}{(3.939 - 2)}$$

∴ $$V_w = 1.031\text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. The dynamic equation for gradually varied flow is given by

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2 T}{gA^3}} = \frac{S_0 - S_f}{1 - F_r^2}$$

where

$\frac{dy}{dx}$ = slope of the water surface with respect to the channel bottom;
S_0 = bed slope of the channel;
S_f = slope of the energy gradient; and
F_r = Froude number.

2. The dynamic equation for gradually varied flow in wide rectangular channel is given by

(i) For Manning's formula

$$\frac{dy}{dx} = \frac{1 - \left(\frac{y_n}{y}\right)^{10/3}}{1 - \left(\frac{y_c}{y}\right)^3}$$

(ii) For Chezy's formula

$$\frac{dy}{dx} = \frac{1 - \left(\frac{y_n}{y}\right)^3}{1 - \left(\frac{y_c}{y}\right)^3}$$

where

y_n = normal depth of flow or depth of flow for uniform flow for a given discharge Q;
y = actual depth of flow for non-uniform flow; and
y_c = critical depth of flow

3. The channel bottom slopes are classified as

(i) Critical slope—when $S_0 = S_c$ and $y_n = y_c$
(ii) Mild slope—when $S_0 < S_c$ and $y_n > y_c$
(iii) Steep slope—when $S_0 > S_c$ and $y_n < y_c$
(iv) Horizontal slope—when $S_0 = 0$ and $y_n = \infty$
(v) Adverse slope—when $S_0 < 0$ and y_n is imaginary or non-existent

where

S_0 = channel bottom slope; and
S_c = critical slope.

4. In a mild sloped channel there will be three zones viz., $y > y_n > y_c$ and $y_n > y > y_c$ and $y_n > y_c > y$ in which respectively M_1, M_2 and M_3 curves will be formed.
5. In a steep slopped channel also there will be three zones viz $y > y_c > y_n$, $y_c > y > y_n$ and $y_c > y_n > y$, in which respectively S_1, S_2 and S_3 curves will be formed.
6. In a critical slopped channel there will be only two zones viz., $y > y_n = y_c$ and $y < y_n = y_c$, in which $C_1$and C_3 curves will be formed.
7. In a horizontal channel also there will be two zones viz., $y_n > y > y_c$ and $y_n > y_c > y$ in which H_2 and H_3 curves will be formed.
8. In an adverse slopped channel also there will be only two zones viz., $y > y_c$ and $y < y_c$ in which A_2 and A_3 curves will be formed.
9. The general characteristics of the various water surface profiles are
   (i) The water surface profiles with subscript 1 and 3 i.e., M_1, M_3, S_1, S_3, C_1, C_3 H_3 and A_3 are back water or rising curves.
   (ii) The water surface profiles with subscript 2 i.e., M_2, S_2, H_2 and A_2 are drawdown or falling curves.
   (iii) The water surface profile meets the normal depth line asymptotically and the critical depth line normally.
10. The length of the water surface profile may be obtained by integrating the dynamic equation for gradually varied flow.

    In the step method of integration the entire length of the channel is divided into short reaches and computation is carried out step by step from one end of the reach to the other. The length of the small reach of the channel dx is given by

$$dx = \frac{E_1 - E_2}{S_0 - S_f} = \frac{\Delta E}{S_0 - S_f}$$

where

ΔE = change in specific energy between the two sections of the small reach of the channel. By adding the dx values for the different reaches the lenght of the water surface profile x is obtained.

11. The hydraulic jump is defined as the sudden and turbulent passage of water from a supercritical

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

state to subcritical state. It is classified as rapidly varied flow.

12. The depth of flow before the hydraulic jump is known as initial depth and the depth of flow after the hydraulic jump is known as sequent depth. The initial and sequent depths are commonly known as cojugate depths which indicate the same specific force.
13. For hydraulic jump in a rectangular channel the relation between the initial depth y_1 and the sequent depth y_2 is given as

$$y_2 = \frac{y_1}{2}\left[-1+\sqrt{1+\frac{8q^2}{gy_1^3}}\right]$$

or

$$y_2 = \frac{y_1}{2}\left[-1+\sqrt{1+8F_{r1}^2}\right]$$

where

q = discharge per unit width of the channel ; and

Fr_1 = froude number before the hydraulic jump.

14. The loss of energy ΔE in a hydraulic jump in rectangular channel is given by

$$\Delta E = \frac{(y_2 - y_1)^3}{4y_1y_2}$$

PROBLEMS

16.1 State and discuss the assumptions made in the derivation of the dynamic equation for gradually varied flow. Starting from first principles derive equations for the slope of the water surface in gradually varied flow with respect to (i) channel bed, (ii) horizontal.

16.2 Show that the water surface slope S_w of a gradually varied flow is equal to the sum of the energy slope S_f and the slope due to velocity change $\frac{d}{dx}\left(\frac{V^2}{2g}\right)$.

16.3 Show that the gradually varied flow equation for flow in a rectangular channel of variable width B may be expressed as

$$\frac{dy}{dx} = \frac{S_0 - S_f + \left(\frac{Q^2y}{gA^3}\right)\frac{dB}{dx}}{1-\frac{Q^2B}{gA^3}}$$

16.4 A trapezoidal channel having bottom width 6 m, side slopes 2 horizontal to 1 vertical and bottom slope 0.0016, carries a discharge of 10 m^3/s. A weir placed across the channel backs up the water to a depth of 2 m. Calculate how far upstream the depth is 1.5 m. Take Manning's $n = 0.025$. [*Ans.* 345 m]

16.5 A rectangular channel 10 m wide carries a discharge of 40 m^3/s. If at a section in this channel the depth is 1.5 m, how far (upstream and downstream) from this section will the depth be 2.0 m. Take $S_0 = 0.00009$ and $n = 0.017$. [*Ans.*367 m upstream]

16.6 A rectangular channel 10 m wide is laid with a break in its bottom slope from 0.01 to 0.0064. If it carries 125 m^3/s, determine the nature of the surface profile and compute its length. Take $n = 0.015$. [*Ans.* S_3 profile, 440 m]

16.7 In a horizontal rectangular channel 1.5 m wide, if the observed depths before and after the jump are 0.2 m and 1.0 m respectively, determine the discharge flowing through the channel. [*Ans.* 1.63 m^3/s]

16.8 A rectangular channel carries a discharge of 3 m^3/s per metre width. If the loss of energy in the hydraulic jump is found to be 3.2 m, determine the conjugate depths before and after the jump. [*Ans.* 0.293 m, 2.36 m]

16.9 In a horizontal triangular channel with sides sloping at 1:1, the depths before and after the jump are observed as 1 m and 1.25 m respectively. Determine the discharge flowing through the channel. [*Ans.* 2.943 m^3/s]

16.10 A discharge of 600 m^3/s flows over spillway and then passes over a horizontal concrete apron (n = 0.015). The velocity of water at the bottom of the spillway is 10 m/s and the width of the apron is 40 m. A hydraulic jump forms on the apron due to a tail water depth of 4.5 m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in the channel below. In order that the jump be contained on the apron, how long should the apron be built? Also determine the loss of energy from the foot of the spillway to the downstream side of the jump.

[*Ans.* 81.5 m; 90.06×10^5 W]

16.11 For a hydraulic jump in a horizontal triangular channel show that

$$3Fr_1^2 = \frac{r^2(r^3-1)}{r^2-1}$$

where $Fr_1^2 = \left(V_1^2 / gy_1\right) =$ and $r = (y_2/y_1)$

16.12 The normal depth of flow of water in a rectangular channel 1.5 m wide is 1 m. The bed slope of the channel is 0.0006 and Manning's n = 0.012. Find the critical depth.

At a certain section of the same channel the depth is 0.92 m while at second section the depth is 0.86 m. Find the distance between the two sections. Also find whether the second section is located upstream of the first section.

[*Ans.* 0.516 m; 222 m downstream]

16.13 Determine the approximate water surface slope (m/km) at a point in a rectangular channel in which the discharge is 0.84 cumec, the bed-width is 3 m, the depth of flow is 0.75 m and the bed slope is 0.15 m/km. Assume C in Chezy's formula as 55 and take energy coeffiient α equal to unity. [*Ans.* 0.091 m/km]

16.14 The depth and velocity of flow in a rectangular channel are 1 m and 1.5 m/s respectively. If the rate of inflow at the upstream end of the channel is increased to the extent that the depth is doubled in magnitude, determine the absolute velocity of the resulting surge.

[*Ans.* 6.925 m/s]

16.15 A gate is to be suddenly dropped into place closing a rectangular channel 2 m deep and 3 m wide in which 6 cumec of water is flowing at a depth of 1.2 m. Will the water spill over the sides? What will be the velocity and height of the surge produced?

[*Ans.* y_2 = 1.842 m, hence water will not spill over the sides ; 3.115 m/s ; 0.642 m]

16.16 Water flows from a lake into a steep rectangular channel 3 m wide. Determine the lake level above the channel bed at the outfall if the discharge is 40 m^3/s. [*Ans.* 2.626 m]

16.17 A sluice acros a channel 6 m wide discharges a stream 1 m deep. What is the flow rate when the depth upstream of the sluice is 7 m? On downstream side, concrete blocks have been placed to create conditions for hydraulic jump to occur. Determine the force on the blocks if the downstream depth is 3 m.

[*Ans.* 65.77 m^3/s; 24,996 kg(or 245.21 kN)]

16.18 Water emerges from a spillway with a velocity of 20 m/s and a depth 0.5 m at the toe. Estimate the distance of the hydraulic jump from the spillway toe if no specific measures are taken for the formation of jump at the toe. Assume for the downstream channel Manning's n = 0.020 and S_o = 0.001. Calculations may be performed by direct step method taking only one step. Assume channel to be a wide rectangular channel.

[**Hint :** y_c = 2.168 m; y_n = 3.026 m; Since $y_n > y_c$, it is mild slopped channel. For sequent depth y_2 = *3.26* m, the initial depth y_1 = 1.493 m. Thus M_3 type surface profile will be formed along which the depth of flow will increase from 0.5 m at the toe of the spillway to 1.493 m at the section where hydraulic jump will be formed. The distance of the hydraulic jump from the toe = 83.14 m which is equal to the length of the M_3 type surface profile.]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Dimensional Analysis, Hydraulic Similitude and Model Investigation

Chapter **17**

17.1 INTRODUCTION

Dimensional Analysis is a mathematical technique which makes use of the study of dimensions as an aid to the solution of several engineering problems. Each physical phenomenon can be expressed by an equation, composed of variable (or physical quantities) which may be dimensional and non-dimensional quantities. Dimensional analysis helps in determining a systematic arrangement of the variables in the physical relationship and combining dimensional variables to form non-dimensional parameters.

In the study of fluid mechanics the dimensional analysis has been found to be useful in both analytical and experimental investigations. Some of the uses of dimensional analysis are as narrated below:

(i) Testing the dimensional homogeneity of any equation of fluid motion.

(ii) Deriving equations expressed in terms of non-dimensional parameters to show the relative significance of each parameter.

(iii) Planning model tests and presenting experimental results in a systematic manner in terms of non-dimensional parameters; thus making it possible to analyse the complex fluid flow phenomenon.

In order to facilitate the study of dimensional analysis and its application to practical problems a preliminary discussion about dimensions is given below.

17.2 DIMENSIONS

The various physical quantities used by the engineers and the scientists, to describe a given phenomenon can be described by a set of quantities which are in a sense independent of each other. These quantities are known as *fundamental* or *primary quantities*. The primary quantities are mass, length, time and temperature, designated by the letters M, L, T and θ respectively. Temperature is specially useful in compressible flow. All other quantities such as area, volume, velocity, acceleration, force, energy, power, etc., are termed as *derived quantities* or *secondary quantities*, because they can be expressed in terms of the primary quantities. The expression for a derived quantity in terms of the primary quantities is called the *dimension* of the physical quantity. For example the dimension of force can be expressed as indicated below.

$$[\text{Force}] = [\text{Mass} \times \text{Acceleration}]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since, [Acceleration] $= \left[\frac{L}{T^2}\right]$

$$[\text{Force}] = \left[\frac{ML}{T^2}\right] = [MLT^2]$$

The rectangular bracket signifies that the dimensions of the quantity are being considered.

Some engineers prefer to use force instead of mass as fundamental quantity because the former is easy to measure. The system is then represented by symbols F-L-T instead of M-L-T. The dimensional form of any quantity is independent of the system of units (*SI*, metric or English) chosen and enables us to convert the units readily from one system of measurements to the other.

Table 17.1 gives the dimensions and units of measurement of measurement for the various physical quantities involved in the problems concerning fluid flow.

TABLE 17.1

Quantities	*Symbol*	*Dimensions in terms of*		*Units of Measurement*		
		M–L–T	*F–L–T*	*SI*	*Metric*	*English*
1	2	3	4	5	6	7
(A) Geometric						
Length (any linear measurement)	L, l	L	L	m	m	ft
Area	A	L^2	L^2	m^2	m^2	ft^2
Volume	V	L^3	L^3	m^3	m^3	ft^3
Curvature	C	1/L	1/L	m^{-1}	m^{-1}	ft^{-1}
Slope	S, *i*	$M^\circ L^\circ T^\circ$	$F^\circ L^\circ T^\circ$	m/m	m/m	ft/ft
Angle	α, θ	$M^\circ L^\circ T^\circ$	$M^\circ L^\circ T^\circ$	radians or degrees	radians or degrees	radians or degrees
Shape factor	η	$M^\circ L^\circ T^\circ$	$M^\circ L^\circ T^\circ$	—	—	—
(B) Kinematic						
Time	T, *t*	T	T	s	s	s
Velocity (linear)	V, v	L/T	L/T	m/s	m/s	ft/s
Angular velocity	ω	1/T	1/T	rad/s	rad/s	rad/s
Frequency	*n*	1/T	1/T	1/s	1/s	1/s
Acceleration (linear)	*a*	L/T^2	L/T^2	m/s^2	m/s^2	ft/s^2
Angular acceleration	α	$1/T^2$	$1/T^2$	rad/s^2	rad/s^2	rad/s^2
Gravitational acceleration	*g*	L/T^2	L/T^2	m/s^2	m/s^2	ft/s^2
Discharge	Q	L^3/T	L^3/T	m^3/s (cumec)	m^3/s (cumec)	ft^3/s (cusec)
Discharge per unit width	*q*	L^2/T	L^2/T	$[m^3/s/m]$	$[m^3/s/m]$	$[ft^3/s/ft]$
Kinematic Viscosity (μ/ρ)	υ	L^2/T	L^2/T	m^2/s	m^2/s	ft^2/s
Circulation	Γ	L^2/T	L^2/T	m^2/s	m^2/s	m^2/s

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 17.1 Contd.

1	*2*	*3*	*4*	*5*	*6*	*7*
(C) Dynamic						
Mass	M, *m*	M	FT^2/L	kg	metric slug(msl) or gm(mass)	slug or lb (mass)
Force	F	ML/T^2	F	N	kg(f)	lb(f)
Weight	W	ML/T^2	F	N	kg(wt.)	lb(wt)
Mass density	ρ	M/L^3	FT^2/L^4	kg/m^3	$ms\ell/m^3$ or gm/cm^3	$slug/ft^3$ or $lb(m)/ft^3$
Specific weight	ω, γ	M/L^2T^2	F/L^3	N/m^3	$kg(f)/m^3$	$lb(f)/m^3$
Specific gravity	*s*	$M°L°T°$	$F°L°T°$	—	—	—
Pressure intensity	*p*	M/LT^2	F/L^2	N/m^2 or pascal (Pa)	$kg(f)/m^2$ or $kg(f)\ cm^2$	$lb(f)/ft^2$ or $lb(f)\ in^2$
Shear stress	τ	M/LT^2	F/L^2	N/m^2 or pascal (Pa)	$kg(f)/m^2$ or $kg(f)\ cm^2$	$lb(f)/ft^2$ or $lb(f)\ in^2$
Dynamic viscosity	μ	M/LT	FT/L^2	$N.s/m^2$ or pa.s	$\frac{kg(f)\text{-}s}{m^2}$	$\frac{lb(f)\text{–}s}{ft^2}$
Surface Tension	σ	M/T^2	F/L	N/m	$kg(f)/m$	$lb(f)/ft$
Modulus of Elasticity	E	M/LT^2	F/L^2	N/m^2 or pascal(Pa)	$kg(f)/m^2$ or $kg(f)/cm^2$	$lb(f)/ft^2$ or $lb(f)/in^2$
Compressibility	1/E	LT^2/M	L^2/F	m^2/N or I/(Pa)	$m^2/kg(f)$ or $cm^2/kg(f)$	$ft^2/lb(f)$ or $in^2/lb(f)$

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 17.1 Contd.

1	*2*	*3*	*4*	*5*	*6*	*7*
Impulse, Momentum	I, M	ML/T	FT	N.s	kg(f)-s	lb(f)-s
Work, Energy	W, E	ML^2/T^2	FL	N.m or joule (J)	kg(f)-m	ft-lb(f)
Torque	T	ML^2/T^2	FL	N.m or joule (J)	kg(f)-m	ft-lb(f)
Power	P	ML^2/T^3	FL/T	N.m/s or watt(W)	$\frac{kg(f)\text{-}m}{s}$	$\frac{ft\text{–}lb(f)}{s}$
Weight rate of flow	W	ML/T^3	F/T	N/s	kg(f)/s	lb(f)/s
(D) General						
Moment of a force	$F \times l$	ML^2/T^2	FL	N.m	kg(f)–m	ft–lb(f)
Moment of Inertia of area	I	L^4	L^4	m^4	m^4	ft^4
Moment of Inertia of mass	Ml^2	ML^2	FLT^2	$kg.m^2$	$\left\{\begin{matrix}msl\text{-}m^2 \\ or \\ gm\text{-}cm^2\end{matrix}\right\}$	$\left\{\begin{matrix}slug\text{-}ft^2 \\ or \\ lb(m)\text{-}ft^2\end{matrix}\right\}$
Thermodynamic temperature	T	θ	θ	K (kelvin)	°C (abs) (degree centigrade or celsius absolute)	°F (abs) (degree Fahrenheit absolute)
Temperature	$t°$	θ	θ	°C (degree Celsius)	°C (degree centigrade or Celsius)	°F (degree Fahrenheit)
Heat	ϕ	ML^2/T^2	FL	joule(J)	kilo-calories	B.Th.U or C.H.U
Specific heat	ϕ_s	$L^2/T^2\theta$	$L^2/T^2\theta$	joule per kilogram kelvin (J/kg.K)	kg(f)-m per msl°C(abs)	lb(f)–ft per slug °F(abs)

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 17.1 Contd.

1	2	3	4	5	6	7
Thermal conductivity or Heat conduction	ϕ_c	$ML/T^3\theta$	$F/T\theta$	watt per metre kelvin (W/m.K)	[kg(f)-m/s] per m °C (abs)	[ft-lb(f)/s] per ft °F(abs)
(E) Dimensionless						
Reynolds, number	Re,N_R	M°L°T°	F°L°T°	—	—	—
Froude number	*Fr*,N_F	M°L°T°	F°L°T°	—	—	—
Euler number	E*u*,N_E	M°L°T°	F°L°T°	—	—	—
Weber number	W*e*,N_W	M°L°T°	F°L°T°	––	––	––
Mach number	Ma,N_M	M°L°T°	F°L°T°	—	—	—
Friction factor	*f*	M°L°T°	F°L°T°	—	—	—
Kinetic Energy correction factor	α	M°L°T°	F°L°T°	—	—	—
Momentum correction factor	β	M°L°T°	F°L°T°	—	—	—
Efficiency	η	M°L°T°	F°L°T°	—	—	—

17.3 DIMENSIONAL HOMOGENEITY

Fourier's principle of dimensional homogeneity states that an equation which expresses a physical phenomenon of fluid flow must be algebraically correct and dimensionally homogeneous.

An equation is said to be dimensionally homogeneous, if the dimensions of the terms on its left hand side are same as the dimensions of the terms on its right hand side. To illustrate this consider the equation for the time of swing of a simple pendulum,

$$t = 2\pi\sqrt{\frac{l}{g}}$$

The dimensions of the left hand side are $[T]$. Since 2π is a pure number, i.e., it has no dimensions, the dimensions of $\sqrt{(l/g)}$ are $\left[\frac{L}{(L/T^2)}\right]^{1/2} = [T]$. Thus the dimensions of the terms on both sides of the equation are same and hence it is dimensionally homogeneous. Some of the other examples of the dimensionally homogeneous equations are as noted below:

$$Q = \frac{2}{3} C_d \sqrt{2g}\, LH^{3/2} \quad \text{... (Flow over rectangular weir)}$$

$$Q = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \quad \text{... (Flow over triangular weir)}$$

$$\Delta p = \frac{32\mu VL}{D^2} \quad \text{... (Laminar flow in pipes)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\Delta p = f \frac{l}{D} \frac{\rho V^2}{2} \quad \text{... (Turbulent flow in pipes)}$$

A dimensionally homogeneous equation has the unique characteristic of being independent of the units chosen for measurement i.e., if an equation is dimensionally homogeneous, it can be used without any modification with either system of units.

It is always possible to reduce a dimensionally homogeneous equation to a non-dimensional form as explaind below:

$$\left[\frac{t}{\sqrt{\frac{l}{g}}}\right] = 2\pi; \left[\frac{Q}{\sqrt{g}LH^{3/2}} = \frac{2\sqrt{2}}{3} C_d\right]; \left[\frac{(\Delta p)D}{\mu V}\right] = 32\left[\frac{L}{D}\right]$$

and

$$\left[\frac{\Delta p}{\frac{\rho V^2}{2}} = f \frac{l}{D}\right]$$

Further from the study of these equations it will be observed that in general the number of dimensionless groups that can be made is equal to the number of variables constituting the equation minus the number of primary quantities involved.

In addition to these, there are several equations in hydraulics which are dimensionally non-homogeneous, but still well applicable to flow system within their limited ranges. For example

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

is Manning's equation (for SI or metric system of units) which relates mean velocity V, in channels to the hydraulic mean depth R, bed slope S and the roughness coefficient n. If n is assumed to be a dimensionless factor, then on analysing the dimensions of both sides of this equation it will be found that

$$\left[\frac{\mathrm{L}}{\mathrm{T}}\right] \neq \left[\mathrm{L}^{2/3}\right]$$

Since the equation is dimensionally non-homogeneous, it can be used only in SI or metric system of units, with V in m/s and R in metres. However, if this equation is to be dimensionally homogeneous, then n must have the dimension $\left[\frac{\mathrm{T}}{\mathrm{L}^{1/3}}\right]$

Some of the other examples of the dimensionally non-homogeneous equations are

$$V = C\sqrt{RS} \quad \text{... (Chezy's formula)}$$

$$V = 0.84y^{0.64} \quad \text{... (Kennedy's equation)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} \quad \text{... (Specific speed of turbines)}$$

$$Q = 1.84\,(L - 0.1\,nH)\,H^{3/2} \quad \text{... (Francis formula)}$$

17.4 METHODS OF DIMENSIONAL ANALYSIS

The following two methods of dimensional analysis are generally used:

(a) Rayleigh Method.

(b) Buckingham π-Method.

(a) Rayleigh Method. This method of dimensional analysis was proposed by Lord Rayleigh in 1899, for determining the effect of temperature on the viscosity of a gas. In this method a functional relationship of some variables is expressed in the form of an exponential equation which must be dimensionally homogeneous. Thus if X is some function of variables $X_1, X_2, X_3..X_n$; the functional equation can be written in the following general form

$$X = f\,(X_1, X_2, X_3.....X_n) \quad \ldots(17.1)$$

In this equation X is a dependent variable, while $X_1, X_2, X_3.....X_n$ are independent variables. A dependent variable is the one about which information is required while independent variables are those which govern the variation of dependent variable.

The above noted equation may be expressed as

$$X = C\left(X_1^a X_2^b X_3^c .....X_n^n\right) \quad \ldots(17.2)$$

in which C is a dimensionless constant which may be determined either from the physical characteristics of the problem or from experimental measurements. The exponents $a, b, c,....n$ are then evaluated on the basis that the equation is dimensionally homogeneous. The dimensionless parameters are then formed by grouping together the variables with like powers. The method is now illustrated by means of an example.

Consider the problem of fluid flow through a small orifice discharging freely into atmosphere under a constant head. Let Q be the discharge passing through a small orifice of diameter d, under a constant head H. Also let ρ be the mass density and μ the dynamic viscosity of the fluid flowing through the orifice. The discharge Q may be assumed to depend on these variables viz., d, H, ρ, μ and the gravitational acceleration g since the flow is freely into atmosphere. Therefore the functional relationship for Q may be written as

$$Q = f(\mu, \rho, d, H, g)$$

which by Rayleigh's method may be expressed in an exponential form

$$Q = C(\mu^a \rho^b d^c H^d g^e)$$

where C is a dimensionless constant.

Substituting the proper dimensions for each variable in this exponential equation in M-L-T system,

$$(L^3/T) = (M^\circ L^\circ T^\circ)\,(M/LT)^a\,(M/L^3)^b\,(L)^c\,(L)^d\,(L/T^2)^e$$

For dimensional homogeneity the exponents of each dimension on the both sides of the equation must be indentical. Thus

for M : $0 = a + b$

for L : $3 = -a - 3b + c + d + e$

for T : $-1 = -a - 2e$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since there are five unknowns in three equations, three of the unknowns must be expressed in terms of the other two:

$$b = -a$$

$$e = \frac{1}{2} - \frac{a}{2}$$

$$c = \frac{5}{2} - \frac{3a}{2} - d$$

Therefore,

$$Q = C\left[\mu^a \rho^{-a} d^{\left(\frac{5}{2}-\frac{3a}{2}-d\right)} H^d g^{\left(\frac{1}{2}-\frac{a}{2}\right)}\right]$$

$$= C\left[\left(d^{\frac{5}{2}} g^{\frac{1}{2}}\right)\left(\mu^a \rho^{-a} d^{\frac{-3a}{2}} g^{\frac{-a}{2}}\right), \left(H^d d^{-d}\right)\right]$$

$$= C\left[\left(d^2 d^{\frac{1}{2}} g^{\frac{1}{2}}\right)\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^d\right]$$

$\therefore$ $$Q = C\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^{d-\frac{1}{2}} d^2 H^{1/2} g^{1/2}$$

$$= \frac{C}{\frac{\pi}{4}\sqrt{2}}\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^{d-\frac{1}{2}} \frac{\pi}{4} d^2 \sqrt{2gH}$$

$$= \left(a\sqrt{2gH}\right) f_1\left[\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right), \left(\frac{H}{d}\right)\right]$$

This expression may be written in the usual form, viz.,

$$Q = C_d\, a \sqrt{2gH}$$

where C_d is the coefficient of discharge of the orifice which can be expressed as

$$C_d = f_1\left[\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right), \left(\frac{H}{d}\right)\right]$$

It may be pointed out that both the terms in the bracket are dimensionless and C_d is also a non-dimensional factor.

(b) Buckingham π-Method. The Buckingham's π-theorem states that if there are n dimensional variables involved in a phenomenon, which can be completely described by m fundamental quantities

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or dimensions (such as mass, length, time etc.), and are related by a dimensionally homogeneous equation, then the relationship among the n quantities can always be expressed in terms of exactly $(n - m)$ dimensionless and independent π terms.

Mathematically, if any variable Q_1 depends on the independent variables, $Q_2, Q_3, Q_4 \ldots\ldots Q_n$; the functional equation may be written as

$$Q_1 = f(Q_2, Q_3, Q_4 \ldots\ldots Q_n) \qquad \ldots(17.3)$$

which can be transformed to another functional relationship

$$f_1(Q_1, Q_2, Q_3, \ldots\ldots Q_n) = C \qquad \ldots(17.4)$$

where C is a dimensionless constant. This is as if $x = f(y) = y^2 + C$; whence $x - y^2 = f_1(x, y) = C$. In accordance with the π-theorem, a non-dimensional equation can thus be obtained in the following form

$$f_2(\pi_1, \pi_2, \pi_3, \ldots\ldots \pi_{n-m}) = C_1 \qquad \ldots(17.5)$$

wherein each dimensionless π-term is formed by combining m variables out of the total n variables with one of the remaining $(n - m)$ variables. These m variables which appear repeatedly in each of the π-terms, are consequently called *repeating variables* and are chosen from among the variables such that they together involve all the m fundamental quantities (or dimensions) and they themselves do not form a dimensionless parameter. Thus the different π-terms may be established as shown below:

$$\left.\begin{aligned} \pi_1 &= Q_1^{a_1} Q_2^{b_1} Q_3^{c_1} \ldots\ldots\ldots Q_m^{m_1} \quad Q_{m+1} \\ \pi_2 &= Q_1^{a_2} Q_2^{b_2} Q_3^{c_2} \ldots\ldots\ldots Q_m^{m_2} \quad Q_{m+2} \\ &\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \\ \pi_{n-m} &= Q_1^{a_{n-m}} Q_2^{b_{n-m}} Q_3^{c_{n-m}} \ldots Q_m^{m_{n-m}} \quad Q_n \end{aligned}\right\} \qquad \ldots(17.6)$$

In Eq. 17.6 each individual equation is dimensionless and the exponents $a, b, c, d \ldots\ldots m$ etc., are determined by considering dimensional homogeneity for each equation so that each π-term is dimensionless.

The final general equation for the phenomenon may then be obtained by expressing any one of the π-terms as a function of the others as

$$\pi_1 = f_1(\pi_2, \pi_3, \pi_4, \ldots\ldots \pi_{n-m})$$
$$\pi_2 = f_2(\pi_1, \pi_3, \pi_4, \ldots\ldots \pi_{n-m})$$

or any other desired relationship may be obtained.

The use of Buckingham's π-method may now be illustrated by means of an example.

Considering the same problem of flow through small orifice (as discussed under Rayleigh's method) the following steps may be adopted to solve it by Buckingham's π-method.

Step 1. The physical quantities involved in the phenomenon are, Q, d, H, g, μ and ρ. The functional equation for the discharge Q may be expressed as

$$Q = f(d, H, g, \mu, \rho)$$

which in its most general form may be written as

$$f_1(Q, d, H, g, \mu, \rho) = C$$

Thus the total number of variables $n = 6$ and all these variables may be completely described by the three fundamental dimensions of either the M-L-T or F-L-T systems. Hence m equals 3. Therefore there are $(n-m) = 3$ dimensionless π-terms, so that

$$f_2(\pi_1, \pi_2, \pi_3) = C_1$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Step 2. In order to form these π-terms, we have to choose 3 repeating variables, since in this case $m = 3$. As stated earlier these variables should be such that they, among them, contain all the three fundamental dimensions and they themselves do not form a dimensionless parameter. Thus let us choose, $\rho(M/L^3)$, $d(L)$ and $g(L/T^2)$ as repeating variables, since the above noted requirements are fulfilled by these.

Step 3. Since physical quantities of dissimilar dimensions can neither be added nor subtracted the terms are expressed as product as follows:

$$\left.\begin{aligned}\pi_1 &= \rho^{a_1} d^{b_1} g^{c_1} Q \\ \pi_2 &= \rho^{a_2} d^{b_2} g^{c_2} \mu \\ \pi_3 &= \rho^{a_3} d^{b_3} g^{c_3} H\end{aligned}\right\}$$

Step 4. Expressing π_1 dimensionally in terms of M-L-T system

$$\pi_1 = M^\circ L^\circ T^\circ = \left(M/L^3\right)^{a_1} (L)^{b_1} \left(L/T^2\right)^{c_1} \left(L^3/T\right)$$

Equating the exponents of M, L and T, we get

for $M : 0 = a_1$

for $L : 0 = -3a_1 + b_1 + c_1 + 3$

for $T : 0 = -2c_1 - 1$

From these equations, we get

$$a_1 = 0; b_1 = -\frac{5}{2}; c_1 = -\frac{1}{2}$$

Hence $$\pi_1 = \left(\frac{Q}{g^{1/2} d^{5/2}}\right)$$

Similarly, we have

$$\pi_2 = \rho^{a_2} d^{b_2} g^{c_2} \mu$$

or $$M^\circ L^\circ T^\circ = \left(M/L^3\right)^{a_2} (L)^{b_2} \left(L/T^2\right)^{c_2} (M/LT)$$

Equating the exponents of M, L, and T, we get

for $M : 0 = a_2 + 1$

for $L : 0 = -3a_2 + b_2 + c_2 - 1$

for $T : 0 = -2c_2 - 1$

from which

$$a_2 = -1; b_2 = -\frac{3}{2}; c_2 = -\frac{1}{2}$$

Hence $$\pi_2 = \left(\frac{\mu}{\rho g^{1/2} d^{3/2}}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Lastly, we have

$$\pi_3 = \rho^{a_3} d^{b_3} g^{c_3} H$$

or $$M^\circ L^\circ T^\circ = \left(M/L^3\right)^{a_3} (L)^{b_3} \left(L/T^2\right)^{c_3} (L)$$

Equating the exponents of M, L and T, we get

for $$M: 0 = a_3$$

for $$L: 0 = -3a_3 + b_3 + c_3 + 1$$

for $$T: 0 = -2c_3$$

from which

$$a_3 = 0; b_3 = -1; c_3 = 0$$

Hence $$\pi_3 = \frac{H}{d}$$

Step 5. Since

$$f_2(\pi_1, \pi_2, \pi_3) = C_1$$

By substitution, we have

$$f_2\left(\frac{Q}{g^{1/2}d^{5/2}}, \frac{\mu}{\rho g^{1/2} d^{3/2}}, \frac{H}{d}\right) = C_1$$

or $$\frac{Q}{g^{1/2}d^{5/2}} = C_2 f_3\left(\frac{\mu}{\rho g^{1/2} d^{3/2}}, \frac{H}{d}\right)$$

from which the expression in its usual form may be obtained as explained earlier.

17.5 OUTLINE OF PROCEDURE FOR BUCKINGHAM METHOD

The following is the outline of the procedure to be followed when the dimensional analysis is carried out by the Buckingham π-method.

(1) List all the n physical quantities or variables involved in the phenomenon. Note their dimensions and the number m of the fundamental dimensions comprised in them. So that there will be $(n - m)$ π-terms.

(2) Select m variables out of these which are to serve as repeating variables. These variables should be such that none of them is dimensionless, no two variables have the same dimensions, they themselves do not form a dimensionless parameter and all the m fundamental dimensions are included collectively in them. Moreover, as far as possible the dependent variable should not be taken as a repeating variable as otherwise it will not be possible to obtain an explicit relationship. In fluid flow problems, usually a characteristic linear dimension, a characteristic velocity and a characteristic fluid property (usually fluid density) are chosen as repeating variables, so that familiar non-dimensional parameters as discussed later are formed.

(3) Write the general equations for different π-terms. These may be expressed as the product of the repeating variables each raised to an unknown exponent and one of the remaining variables, taken in turn, with a known power (usually taken as one).

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(4) Write the dimensional equations for the equations of the π-terms obtained in the step (3) above.

Compute the values of the unknown exponents by equating the exponents of the respective fundamental dimensions on both the sides of each of the dimensional equation and obtain the different dimensionless groups or π-terms.

(5) Write the final general equation for the phenomenon in terms of the π-terms.

In order to obtain the final expression in the desired manner the following useful suggestions may be noted:

(i) If a quantity is dimensionless, it is a π-term without going through the above procedure.

(ii) If any two physical quantities have the same dimensions, their ratio will be one of the π-terms. For example (H/d) is dimensionless and hence it is a π-term which may be directly written without going through the above procedure.

(iii) Any π-term may be replaced by any power of that term, including negative as well as fractional powers. For example π_1 may be replaced by π_1^{-1}, or π_2 may be replaced by π_2^2, π_3 may be replaced by $(1/\pi_3^{1/2})$ etc.

(iv) Any π-term may be replaced by multiplying it by a numerical constant. For example π_1 may be replaced by $3\pi_1$ or so.

(v) Any π-term may be replaced by another π-term obtained by adding or subtracting an absolute numeral from it.

(vi) Any π-terms may be replaced by multiplying or dividing it by another π-term. For example π_1 may be replaced by $(\pi_1 \times \pi_2)$ or (π_1/π_2), or π_1 and π_2 may be replaced by $(\pi_1 \times \pi_2)$ or (π_1/π_2).

A critical study of the methods of the dimensional analysis discussed above reveals that the two methods are essentially the same. But when the number of variables is considerably increased, the Rayleigh's method becomes tedious. This difficulty in case of more variables is however, avoided by using Buckingham's π-method instead of Rayleigh's method.

17.6 NUMBER OF DIMENSIONLESS GROUPS IN A COMPLETE SET OF VARIABLES

According to Buckingham's π-theorem if there are n variables involved in a phenomenon then these n variables can be grouped together into $(n–m)$ dimensionless groups (or π-terms) when m is the number of the fundamental dimensions required to express the n variables. This has been found to be true not only for the problems of fluid mechanics, but also for the problems relating to several other fields. However, in some cases the above rule does not hold good and therefore it cannot be considered as a general rule for determining the number of the dimensionless groups in a complete set of variables. As such in order to obtain the number of dimensionless groups into which a complete set of variables may be grouped, another method based on *dimensional matrix approach* may be adopted which is as follows:

(i) List all the variables involved in the phenomenon and prepare a *dimensional matrix* of the variables, which is nothing but display of the exponents of the dimensions of the variables by a tabular arrangement.

(ii) Find the rank of the dimensional matrix. The rank of a matrix is said to be r, if it contains a non-zero determinant of order r and if all determinants of order greater than r that the matrix contains are equal to zero.

(iii) The number of dimensionless groups of π-terms is equal to $(n–r)$, that is, total number of variables n minus the rank of the dimensional matrix r.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As an example let it be assumed that in a particular problem the variables involved are velocity V, length L, force F, mass density ρ, dynamic viscosity μ, and gravitational acceleration g. The dimensional matrix is formed by writing the exponents (or powers) of M, L and T which appear in the dimensional expression of the variable in the column below the variable itself. As such dimensional matrix of these variables is as given below:

	V	L	F	ρ	μ	g
M	0	0	1	1	1	0
L	1	1	1	–3	–1	1
T	–1	0	–2	0	–1	–2

In order to find the rank of the matrix, a third order determinant formed from any three columns of the matrix may be considered and evaluated. Thus the determinant formed from the last three columns in the above dimensional matrix is

$$\begin{vmatrix} 1 & 1 & 0 \\ -3 & -1 & 1 \\ 0 & -1 & -2 \end{vmatrix} = -3$$

Since in this case a third order determinant of the dimensional matrix is non-zero (i.e., it has a value different from zero) the rank of the matrix is 3. However, if this determinant would have been equal to zero, then other third order determinants should be considered and evaluated one after the other till any one of these is non-zero. Furthermore if none of the third order determinants of the dimensional matrix is non-zero, then the second order determinants should be considered and evaluated till a non-zero determinant of second order is found.

Since the rank of the above dimensional matrix is 3 and the number of variables is 6, the number of dimensionless groups in the present case is therefore (6 – 3) = 3. The actual dimensionless groups may however be formed by using Buckingham's method.

Another example that may be considered is that of capillary rise h which depends on the radius r of the tube, surface tension σ and specific weight w of the liquid. Thus in this case there are 4 variables which involve 3 fundamental dimensions. As such according to Buckingham's π-theorem these four variables can be grouped into (4 – 3) i.e., only 1 dimensionless group. However, it is not true and by adopting the above method based on dimensional matrix approach, as indicated below, it may be shown that in this case the number of dimensionless groups will be 2.

The dimensional matrix of these four variables is as given below:

	h	r	σ	w
M	0	0	1	1
L	1	1	0	–2
T	0	0	–2	–2

In order to find the rank of the above matrix, a third order determinant formed from any three columns of the matrix may be considered and evaluated. But in this case it will be observed that none of the third order determinant is non-zero. As such the rank of the matrix is not three. Then a second order matrix as indicated below may be considered and evaluated:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\begin{vmatrix} 1 & 1 \\ 0 & -2 \end{vmatrix} = -2$$

Since in this case a second order determinant of the dimensional matrix is non-zero, the rank of the matrix is 2 and the number of variables is 4, the number of dimensionless groups is therefore (4 – 2)= 2. The actual dimensionless groups may however be formed by using Buckingham's method.

17.7 SUPERFLUOUS AND OMITTED VARIABLES

The first step in the dimensional analysis of a problem is to decide about the variables that govern the flow phenomenon. The various variables that may occur in the problems concerning fluid flow are categorically listed in Table 17.1. Any phenomenon under consideration may involve only a few of these variables. There is however, no definite rule which may govern the proper selection of variables to be included in any problem, and it has to be carried out purely as guess work though supported by logic. But even for this, one must have enough insight into the mechanism of the problem, so as to explain why and how any of the variables, otherwise considered suitable, influences the phenomenon.

Often variables may be included that really do not have any effect on the phenomenon. Such variables are known as *superfluous variables.* The inclusion of these variables will result in the appearance of too many terms in the final equation, thereby making the whole analysis unnecessarily complicated. These superfluous variables can be eliminated from the analysis only on the basis of the results obtained from the experimental investigation of the problems. This may be made clear by means of an example. In the problem of flow of fluid under pressure through a pipe, if the acceleration due to gravity, g, is taken as one of the variables in addition to those considered in the Illustrative Example 17.1 for the pressure drop Δp, one more π-term may be formed, as indicated below:

$$\pi_5 = V^{a_5} D^{b_5} \rho^{c_5} g$$

which by the usual procedure will be grouped as

$$\pi_5 = \left(\frac{gD}{V^2}\right) \text{ or } \left(\frac{V^2}{gD}\right)$$

Thus the final expression for Δp will have to be modified as

$$\left(\frac{\Delta p}{\rho V^2}\right) = \phi\left(\frac{l}{D}, \frac{k}{D}, \frac{\rho DV}{\mu}, \frac{V^2}{gD}\right)$$

A set of experiments can now be carried out to determine if g really affects the phenomenon. For this purpose the other parameters namely $\left(\frac{l}{D}\right)$, $\left(\frac{k}{D}\right)$ and $\left(\frac{\rho DV}{\mu}\right)$ must be kept constant while $\left(\frac{V^2}{gD}\right)$ is varied. A plot of $\left(\frac{\Delta p}{\rho V^2}\right)$ against $\left(\frac{V^2}{gD}\right)$ prepared from the experimental observations would then indicate a straight line parallel to the axis of $\left(\frac{V^2}{gD}\right)$ showing thereby that g is irrelevant to this problem.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Alternatively, a number of experiments are carried out and all the variables, V, D, ρ, π, l and k changed, then a plot of $\left(\frac{\Delta p}{\rho V^2}\right)$ against $\left(\frac{\rho DV}{\mu}\right)$ for constant values of $\left(\frac{l}{D}\right)$ and $\left(\frac{k}{D}\right)$, would show all the points on a single curve on graph, no matter how $\left(\frac{V^2}{gD}\right)$ changes. Thus it can be concluded that the acceleration due to gravity g, considered additionally in the present problem is a superfluous variable. However if g would have been really significant, a plot of $\left(\frac{\Delta p}{\rho V^2}\right)$ against $\left(\frac{\rho DV}{\mu}\right)$ for constant value of $\left(\frac{l}{D}\right)$ and $\left(\frac{k}{D}\right)$, would show several curves on the graph, one curve for each value of $\left(\frac{V^2}{gD}\right)$.

If, on the contrary, any of the pertinent variable that may actually influence the phenomenon, is omitted at the beginning, the analysis may lead to an incomplete, or even erroneous conclusions. Thus for instance in the same problem of fluid flow through a pipe if viscosity of the flowing fluid is not included in the analysis, the π-term $\left(\frac{\rho DV}{\mu}\right)$ will be eliminated from the general relationship shown above. If the viscosity of the fluid has a significant effect on the phenomenon, the inferences drawn on the basis of the modified relationship viz., $\left(\frac{\Delta p}{\rho V^2}\right) = C_1 f_1\left(\frac{l}{D}, \frac{k}{D}\right)$ will be altogether incorrect. This will however be indicated by the final experimental plot of π-terms which will show a scatter of the points which is not due to experimental error. In such cases a search must then be made for the omitted variable, a new dimensional analysis made and a revised plotting carried out.

A common mistake may be on account of the omission of certain variables that have a practically constant value. These variables are sometimes essential because they combine with other active variables to form dimensionless parameters. The example of gravitational acceleration g, may again be quoted in this context. In combination with certain characteristic length dimension, this variable having almost a constant value, develops a parameter, which has the dimensions of those of $(\text{velocity})^2$. In certain problems, wherein the effect of gravity prevails, this new dimensional parameter combines with other variables to form an important non-dimensional parameter.

It is thus evident that the method of dimensional analysis does not give any clue as to the correctness of the selection of the relevant variables of a particular phenomenon. It merely tells how the variables should be grouped so that from the experimental investigation it may be decided which of the variables are important ones. This is however, the main *limitation* of the dimensional analysis.

17.8 USE OF DIMENSIONAL ANALYSIS IN PRESENTING EXPERIMENTAL DATA

If there are only two variables involved in any problem, a single curve is sufficient to describe their functional relationship. An addition of one more variable to a problem will require a family of curves relating any two variables with constant values of the third variable as varying parameter. Likewise

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

as the number of variables involved in any problem increases, more and more number of families of curves are required to describe their relative effect on the phenomenon. It is thus obvious that with a large number of variables involved in any problem, it may be rather very difficult to carry out the tests in an orderly manner and to present the experimental data in suitable form, unless the number of variables is reduced. Dimensional analysis is however, extremely useful in reducing the number of variables in a problem by formulating them into non-dimensional parameters. The reduction of the number of variables in a problem provides a systematic scheme for planning laboratory tests and also permits the presentation of experimental results in a more concise and useful form.

Suppose it has been found by the dimensional analysis that some of the variables A, B, C, D etc., influencing any phenomenon, are interconnected by say four non-dimensional groups π_1, π_2, π_3, and π_4 such that

$$\pi_1 = f(\pi_2, \pi_3, \pi_4)$$

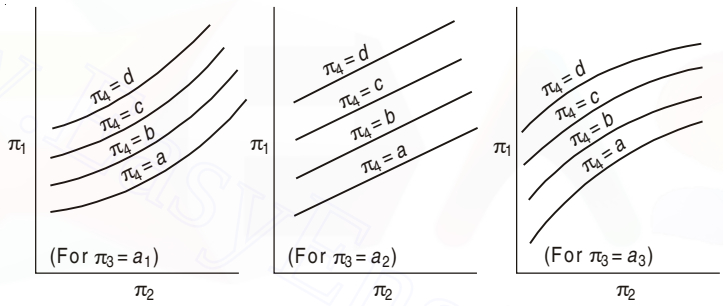


FIGURE 17.1 *Family of curves plotted between non-dimensional groups*

An experimental investigation will provide certain data which may be presented as shown in Fig.17.1, as a set of groups of π_1 against π_2 with a number of curves for different values of π_4 and each graph being for a certain value of π_3. In this way the effect of every group is presented separately. There will of course be many sets of graphs if the number of relevant non-dimensional π groups is large. If one group, say π_4 is irrelevant, then all the curves for different values of π_4 in each graph will coincide leaving only π_1, π_2 and π_3 as relevant groups.

17.9 MODEL INVESTIGATION

The engineers associated with the design, construction and efficient working of the various types of hydraulic structures (such as dams, spillways etc.) or hydraulic machines (such as turbines, pumps etc.) usually try to find out, in advance, how the structure or the machine would behave when it is actually constructed. For this purpose the engineers have to resort to experimental investigation. Such experiments are also necessitated in the case of the problems which cannot be solved completely simply by theoretical analysis. Obviously the experiments can not be carried out on the full size hydraulic structures or machines, which are proposed to be erected. It is then essential to construct a small scale replica of the structure or the machine and the tests are performed on it to obtain the desired information. The small scale replica of the actual structure or the machine is known as its *model* while the actual structure or machine is called *prototype*. Figure 17.2 shows a prototype of overflow spillway and its model. Mostly the models are much smaller than the corresponding prototypes, but in some cases the models may be larger than the prototypes.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

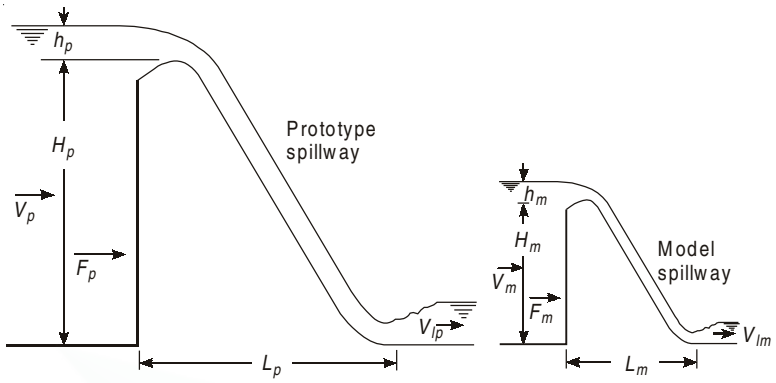


Figure 17.2 *Prototype spillway and its model*

The model tests are quite economical and convenient, because the design, construction and operation of the model may be altered several times if necessary (without incurring much expenditure), till all the defects of the model are eliminated and the most suitable design is obtained. On the basis of the final results obtained from the model tests the design of the prototype may be modified and also it may be possible to predict the behaviour of the prototype. However, the model test results can be utilized to obtain in advance the useful information about the performance of the prototype only if there exists a complete similarity between the model and prototype, which may be achieved as indicated below.

17.10 SIMILITUDE–TYPES OF SIMILARTIES

There are in general three types of similarities to be established for complete similarity to exist between the model and its prototype. These are:

(a) Geometric Similarity

(b) Kinematic Similarity

(*c*) Dynamic Similarity.

(a) Geometric Similarity. Geometric similarity exists between the model and the prototype if the ratios of corresponding length dimensions in the model and the prototype are equal. Such a ratio is defined as scale ratio and may be written as:

Length scale ratio $$= L_r = \frac{L_m}{L_p} = \frac{b_m}{b_p} = \frac{d_m}{d_p} \text{ etc.}$$

Area scale ratio $$= A_r = \frac{A_m}{A_p} = \left(\frac{L_m \times b_m}{L_p \times b_p}\right) = L_r^2$$

Volume scale ratio $$= V_r = \frac{V_m}{V_p} = \left(\frac{L_m \times b_m \times d_m}{L_p \times b_p \times d_p}\right) = L_r^3$$

in which subscripts m and p correspond to model and prototype respectively.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

It will thus be observed that if the model and the prototype are geometrically similar, by mere change of scale they can be superimposed.

(b) Kinematic Similarity. Kinematic similarity exists between the model and the prototype if (1) the paths of the homologous moving particles are geometrically similar, and (2) if the ratios of the velocities as well as acceleration of the homologous particles are equal. Homologous point means corresponding point in the model and the prototype. Since velocity and acceleration have magnitude and direction, kinematic similarity implies that velocities and accelerations at corresponding points are parallel and have a constant ratio of magnitude between all the corresponding sets of points. A few useful scale ratios are as follows:

Time scale ratio $= T_r = \dfrac{T_m}{T_p}$

Velocity scale ratio $= V_r = \dfrac{V_m}{V_p} = \dfrac{\dfrac{L_m}{T_m}}{\dfrac{L_p}{T_p}} = \dfrac{L_r}{T_r}$

Acceleration scale ratio

$$= a_r = \frac{a_m}{a_p} = \frac{\dfrac{L_m}{(T_m)^2}}{\dfrac{L_p}{(T_p)^2}} = \frac{L_r}{T_r^2}$$

Discharge scale ratio

$$= Q_r = \frac{Q_m}{Q_p} = \frac{\dfrac{(L_m)^3}{T_m}}{\dfrac{(L_p)^3}{T_p}} = \frac{L_{r^3}}{T_r}$$

Kinematic similarity can be attained if flownets for the model and the prototype are geometrically similar, which in turn means that by mere change in scale the two flownets—one for the model and other for the prototype—can be superimposed.

(c) Dynamic Similarity. Dynamic similarity exists between the model and the prototype which are geometrically and kinematically similar if the ratio of all the forces acting at homologous points in the two systems viz., the model and the prototype, are equal. Thus for flows to be dynamically similar, the ratios of the various forces acting on the fluid particles in one flow system should be equal to the ratios of similar forces at corresponding points in the other flow system.

In the problems concerning fluid flow, the forces acting may be any one, or a combination of the several of the following forces:

(i) Inertia forces, F_i

(ii) Friction or viscous forces, F_v.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) Gravity forces, F_g
(iv) Pressure forces, F_p
(v) Elastic forces, F_e
(vi) Surface tension forces, F_s

Inertia force is the force of resistance offered by an inert mass to acceleration. According to Newton's second law of motion, the magnitude of inertia force is equal to the product of the particle mass and particle acceleration and its direction is opposite to the direction of the acceleration of the particle. The conditions required for complete dynamic similarity are developed from the Newton's second law of motion. If in a certain system of flowing fluid, a fluid particle of mass M is subjected to acceleration a, the inertia force F_i of the particle equals Ma. Again, if all the above noted forces exist in the system under consideration, then the resultant force ΣF, acting on the particle, which is the vectorial sum of all the forces acting on the particle, will be equal to the inertia force of the particle i.e.,

$$\Sigma F = F_v + \rightarrow F_g + \rightarrow F_p + \rightarrow F_e + \rightarrow F_s = (Ma)$$

For complete dynamic similarity to exist between the model and its prototype, the ratio of inertia forces of the two systems must be equal to the ratio of the resultant forces. Thus the following relation between the forces acting on model and prototype develops:

$$\frac{(\Sigma F)_m}{(\Sigma F)_p} = \frac{(F_v + \rightarrow F_g + \rightarrow F_p + \rightarrow F_e + \rightarrow F_s)_m}{(F_v + \rightarrow F_g + \rightarrow F_p + \rightarrow F_e + \rightarrow F_s)_p} = \frac{(Ma)_m}{(Ma)_p} \quad ...(17.7)$$

In addition to the above noted condition for complete dynamic similarity, the ratio of the inertia forces of the two systems must also be equal to the ratio of individual component forces i.e., the following relationships will be developed:

(i) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_v)_m}{(F_v)_p} \text{ or } \left(\frac{Ma}{F_v}\right)_m = \left(\frac{Ma}{F_v}\right)_p \quad ...(17.8)$$

(ii) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_g)_m}{(F_g)_p} \text{ or } \left(\frac{Ma}{F_g}\right)_m = \left(\frac{Ma}{F_g}\right)_p \quad ...(17.9)$$

(iii) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_p)_m}{(F_p)_p} \text{ or } \left(\frac{Ma}{Fp}\right)_m = \left(\frac{Ma}{F_p}\right)_p \quad ...(17.10)$$

(iv) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_e)_m}{(F_e)_p} \text{ or } \left(\frac{Ma}{F_e}\right)_m = \left(\frac{Ma}{F_e}\right)_p \quad ... (17.11)$$

(v) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_s)_m}{(F_s)_p} \text{ or } \left(\frac{Ma}{F_s}\right)_m = \left(\frac{Ma}{F_s}\right)_p \quad ... (17.12)$$

It may thus be mentioned that when the two systems are geometrically, kinematically and dynamically similar, then they are said to be *completely similar* or *complete similitude* exists between the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

two systems. However, as stated earlier dynamic similarity implies geometric and kinematic similarities and hence if two systems are dynamically similar, they may be said to be completely similar. Moreover, as indicated later, for complete similitude to exist between the two systems viz., model and prototype, the dimensionless or π terms, formed out of the complete set of variables involved in that phenomenon, must be equal.

17.11 FORCE RATIOS–DIMENSIONLESS NUMBERS

Since inertia force always exists when any mass is in motion, the conditions for dynamic similarity are developed by considering the ratio of the inertia force and any one of the remaining forces noted above. Each of these ratios will obviously be a non-dimensional factor. These ratios are discussed below:

(a) The Inertia Force Ratio

$$\text{Inertia force} = \text{mass} \times \text{acceleration}$$

$$= \rho \times \text{volume} \times \frac{\text{velocity}}{\text{time}}$$

$$= \rho \times \frac{\text{volume}}{\text{time}} \times \text{velocity}$$

$$= \rho \times AV \times V$$

$$= \rho L^2 V^2$$

Since area A has the dimensions as L^2

$$\therefore \quad (F_i)_r = \frac{(F_i)_m}{(F_i)_p} = \frac{\rho_m L_m^2 V_m^2}{\rho_p L_p^2 V_p^2} = \rho_r L_r^2 V_r^2.$$

(b) Inertia-Viscous Force Ratio–Reynolds Number

Since $F_i = \rho L^2 V^2$

and $$F_v = \tau A = \left(\mu \frac{dv}{dy}\right) A = \left(\mu \frac{V}{L}\right) L^2 = (\mu VL)$$

where τ and μ are shear stress and absolute viscosity respectively.

$$\therefore \quad \frac{F_i}{F_v} = \frac{\rho L^2 V^2}{\mu VL} = \left[\frac{\rho VL}{\mu}\right] = \left[\frac{VL}{\upsilon}\right]$$

where $(\mu/\rho) = \upsilon$ the kinematic viscosity of the fluid.

This non-dimensional ratio $(\rho VL/\mu)$ or (VL/υ) is called 'Reynolds number' (Re or N_R) in honour of O. Reynolds, a British physicist. Thus Reynolds number signifies the relative predominance of the inertia to the viscous forces occurring in the flow system. The larger the Reynolds number, the greater will be the relative contribution of inertia effects. The smaller the Reynolds number, the greater will be the relative magnitude of viscous force.

(c) Inertia–Gravity Force Ratio–Froude Number

$$F_i = \rho L^2 V^2$$

$$F_g = \text{mass} \times \text{acceleration due to gravity}$$

$$= (\rho \times \text{volume}) \times g$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \rho L^3 g$$

$$\therefore \quad \frac{F_i}{F_g} = \frac{\rho L^2 V^2}{\rho L^3 g} = \frac{V^2}{Lg}$$

The square root of this ratio i.e., $\left(\frac{V}{\sqrt{gL}}\right)$ is known as 'Froude number' (Fr or N_F) after W. Froude, a British naval architect who first applied it to the practical problem of investigating the resistance to ships. Froude number may also be interpreted as a ratio of mean velocity to the velocity of a small wave (or celerity) in quiet fluid. Its value equal to one is considered as critical value.

(d) Inertia–Pressure force Ratio–Euler Number

$$F_i = \rho L^2 V^2$$
$$F_p = \text{pressure intensity} \times \text{area}$$
$$= p \times A = p \times L^2$$

where p represents pressure intensity.

$$\therefore \quad \frac{F_i}{F_p} = \frac{\rho L^2 V^2}{pL^2} = \frac{\rho V^2}{p} = \frac{V^2}{p/\rho}$$

The square root of this ratio i.e., $\frac{V}{\sqrt{p/\rho}}$ is designated as 'Euler number', named after Swiss mathematician and is represented by the symbol Eu or N_E. The reciprocal of this number viz., $\frac{1}{Eu}$ or $\left(\frac{\sqrt{p/\rho}}{V}\right)$ is sometimes known as 'Newton number'. Further ratio of pressure force to inertial force is also known as 'pressure coefficient'.

(e) Inertia–Elasticity Force Ratio–Mach Number

$$F_i = \rho L^2 V^2$$
$$F_e = \text{Bulk modulus of elasticity} \times \text{Area}$$
$$= (K \times A) = K \times L^2$$

where K is the bulk (or volume) modulus of elasticity of the flowing fluid.

$$\frac{F_i}{F_e} = \frac{\rho L^2 V^2}{KL^2} = \frac{V^2}{K/\rho} = \frac{V^2}{C^2}$$

where $C = \sqrt{K/\rho}$, which represents the velocity of sound in that fluid medium whose K and ρ are being considered.

The ratio $\left(\frac{V^2}{C^2}\right)$ or $\frac{V^2}{K/\rho}$ is known as 'Cauchy number'. The square root of this ratio i.e, $\frac{V}{C}$ or

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\frac{V}{\sqrt{K/\rho}}$ is known as 'Mach number' (*Ma* or N_M) in honour of *E*. Mach an Austrian philosopher. As Mach number represents a ratio of two velocities, its value equal to one, is considered as critical value. This is so because it amounts to the characteristic velocity of the phenomenon becoming equal to the velocity of sound in that fluid medium. It becomes still more significant when $Ma > 1$ i.e., $V > C$, in which case the flow is designated as *supersonic*. For $Ma < 1$ i.e., $V < C$ the flow is termed as *subsonic*, and if $Ma = 1$ i.e., $V = C$ flow is considered to be *sonic*. When $Ma >> 1$ i.e., $V >> C$ then the flow is sometimes termed as *hypersonic*. A higher Mach number signifies the predominance of the effect of compressibility of the fluid. However, when Mach number is relatively small say $Ma < 0.4$, the effect of compressibility of the fluid can be altogether neglected.

(f) Inertia–Surface Tension Force Ratio–Weber Number

$$F_i = \rho L^2 V^2$$
$$F_s = \sigma L$$

where σ represents surface tension

$$\therefore \quad \frac{F_i}{F_s} = \frac{\rho L^2 V^2}{\sigma L} = \frac{\rho V^2 L}{\sigma} = \frac{V^2}{\sigma/\rho L}$$

The square root of this ratio viz., $\frac{V}{\sqrt{\sigma/\rho L}}$ is known as 'Weber number' in honour of W.E. Weber, a German physicist. In analogy with other numbers, a smaller Weber number signifies larger predominance of surface tension force and vice versa. It is represented by *We* or N_W.

17.12 SIMILARITY LAWS OR MODEL LAWS

The results obtained from the model tests may be transferred to the prototype by the use of model laws which may be developed from the principles of dynamic similarity. The conditions for the existence of complete dynamic similarity between a model and its prototype are denoted by Eqs 17.7 to 17.12. It may, however, be pointed out that, in the case of almost all the hydraulic problems, for which model studies are required to be carried out, it is quite rare that all the above noted forces are simultaneously predominant in the phenomenon. Moreover, debarring certain exceptions, in most of the problems only one force in addition to the inertia force is relatively more significant than the rest of the forces, which may either not exist or may be of negligible magnitude. Under the circumstances the various model laws have been developed depending upon the significant influence of each of the forces on the different phenomena. In the derivation of these model laws, it has been assumed that for equal values of the dimensionless parameters the corresponding flow pattern in model and its prototype are similar.

(a) Reynolds Model Law. For the flows where in addition to inertia, viscous force is the only other predominant force, the similarity of flow in the model and its prototype can be estabilised if the Reynolds number is same for both the systems. This is known as Reynolds model law, according to which

$$(\mathrm{Re})_{\text{model}} = (\mathrm{Re})_{\text{prototype}}$$

or

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p} \qquad ...(17.13)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{\rho_r V_r L_r}{\mu_r} = 1 \quad \text{... (17.14)}$$

or $$\frac{V_r L_r}{\upsilon_r} = 1 \quad \text{... (17.15)}$$

where the various quantities with subscript r represent the corresponding scale ratios.

Equation 17.14 or 17.15 may be used to obtain the scale ratios for various other physical quantities on the basis of Reynolds model law. Some of these scale ratios are shown in Table 17.2.

TABLE 17.2 *Scale Ratios for Models Governed by Reynolds and Froude Model Laws Scale Ratios*

Decription of Quantities	*Reynolds Law*	*Froude Law*
Length	L_r	L_r
Velocity	$\mu_r / L_r \rho_r$	$L_r^{1/2} g_r^{1/2}$
Time	$L_r^2 \rho_r / \mu_r$	$L_r^{1/2} / g_r^{1/2}$
Acceleration	$\mu_r^2 / \rho_r^2 L_r^3$	g_r
Discharge	$L_r \mu_r / \rho_r$	$L_r^{5/2} g_r^{1/2}$
Force	μ_r^2 / ρ_r	$\rho_r L_r^3 g_r$
Work, Energy and Torque	$\mu_r^2 L_r / \rho_r$	$\rho_r L_r^4 g_r$
Pressure intensity	$\mu_r^2 / L_r^2 \rho_r$	$\rho_r g_r L_r$
Power	$\mu_r^3 / \rho_r^2 L_r$	$\rho_r L_r^{7/2} g_r^{3/2}$

Some of the phenomena for which the Reynolds model law can be a sufficient criterion for similarity of flow in the model and the prototype are—flow of incompressible fluid in closed pipes, motion of submarines completely under water, motion of air planes, and flow around structures and other bodies immersed completely under moving fluids.

(b) Froude Model Law. When the force of gravity can be considered to be the only predominant force which controls the motion in addition to the force of inertia, the similarity of the flow in any two such systems can be established if the Froude number for both the systems is the same. This is known as Froude model law according to which

$$(Fr)_{\text{model}} = (Fr)_{\text{prototype}}$$

or $$\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_p}{\sqrt{g_p L_p}} \quad \text{...(17.16)}$$

or $$\frac{V_r}{\sqrt{g_r L_r}} = 1; \text{ or } V_r = \sqrt{g_r L_r} \quad \text{...(17.17)}$$

Since in most of the cases $g_r = 1$, as the value of g at the site of model testing will be practically the same as at the site of the proposed prototype, Eq. 17.17 then becomes

$$\frac{V_r}{\sqrt{L_r}} = 1; \text{ or } V_r = \sqrt{L_r} \quad \text{...(17.18)}$$

Equation 17.17 or 17.18 may be used to obtain the scale ratios for various other physical quantities on the basis of Froude model law. Some of these scale ratios are also shown in Table 17.2.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Some of the phenomena for which the Froude model law can be a sufficient criterion for dynamic similarity to be established in the model and the prototype are—free surface flows such as flow over spillways, sluices etc., in which gravity is a motivating force, flow of jet from an orifice or nozzle, problems in which waves are likely to be formed on the surface and problems in which fluids of different mass densities flow over one another.

(c) Euler Model Law. In a fluid system where supplied pressures are the controlling forces in addition to the inertia force and the other forces are either entirely absent or are insignificant, the dynamic similarity is obtained by equating the Euler number for both the model and its prototype. This is known as Euler model law according to which

$$(Eu)_{\text{model}} = (Eu)_{\text{prototype}}$$

or
$$\frac{V_m}{(p_m/\rho_m)^{1/2}} = \frac{V_p}{(p_p/\rho_p)^{1/2}} \quad \text{... (17.19)}$$

or
$$\frac{V_r}{(p_r/\rho_r)^{1/2}} = 1 \quad \text{... (17.20)}$$

Equation 17.20 represents the primary relationship for the Euler model law which may be used to evaluate the scale ratios for various other physical quantities in accordance with the Euler model law.

Euler model law may be contemplated as an essential requirement for establishing dynamic similarity in an enclosed fluid system where the turbulence is fully developed, so that the viscous forces are insignificant, and also the forces of gravity and surface tension are entirely absent.

(d) Mach Model Law. If in any phenomenon only the forces resulting from elastic compression are significant in addition to inertia and all other forces may be neglected, then the dynamic similarity between the model and its prototype may be achieved by equating the Mach number for both the systems. This is known as Mach model law, according to which

$$(Ma)_{\text{model}} = (Ma)_{\text{prototype}}$$

or
$$\frac{V_m}{\sqrt{K_m/\rho_m}} = \frac{V_p}{\sqrt{K_p/\rho_p}} \quad \text{... (17.21)}$$

or
$$\frac{V_r}{\sqrt{K_r/\rho_r}} = 1 \quad \text{... (17.22)}$$

The expression represented by Eq. 17.22 may be considered as the basic relationship for the Mach model law from which the scale ratios for the other physical quantities may be derived.

The similitude based on Mach model law finds extensive application in aerodynamic testing and in phenomena involving velocities exceeding the speed of sound. In addition to this, it is also applied in hydraulic model testing for the cases of unsteady flow, especially water hammer problems.

(e) Weber Model Law. When surface tension effects predominate in addition to inertia force the pertinent similitude law is obtained by equating the Weber number for the model and its prototype, which is known as Weber-model law. Thus according to this model law

$$(We)_{\text{model}} = (We)_{\text{prototype}}$$

or
$$\frac{V_m}{\sqrt{(\sigma_m/\rho_m L_m)}} = \frac{V_p}{\sqrt{(\sigma_p/\rho_p L_p)}} \quad \text{... (17.23)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{V_r}{\sqrt{\sigma_r / \rho_r L_r}} = 1 \quad \text{... (17.24)}$$

Equation 17.24 may be considered as the basic equation for the Weber model law, and the scale ratios for the various physical quantities may be derived with the help of this equation.

Some of the practical cases where surface tension forces dominate and accordingly the Weber model law can be the sufficient criterion for the dynamic similarity to be established between the model and the prototype are—flow over weirs involving very low heads, very thin sheet of liquid flowing over a surface, capillary waves in channels etc.

17.13 TYPES OF MODELS

In general hydraulic models can be classified under two broad categories:

(i) Undistorted Models

(ii) Distorted Models.

(i) Undistorted Models. An undistorted model is that which is geometrically similar to its prototype, that is, the scale ratios for corresponding linear dimensions of the model and its prototype are same. Since the basic condition of perfect similitude is satisfied, prediction in the case of such models is relatively easy and many of the results obtained from the model tests can be transferred directly to the prototype.

(ii) Distorted Models. Distorted models are those in which one or more terms of the model are not identical with their counterparts in the prototype. Since the basic condition of perfect similitude is not satisfied, the results obtained with the help of a distorted model are liable to distortion and have more qualitative value only.

A distorted model may have either geometrical distortion, or material distortion, or distortion of hydraulic quantities or a combination of these. In geometrical distortion, the distortion can be either of dimension or that of configuration. When different scale ratios are adopted for the longitudinal, transverse and vertical dimensions then it is said to be a distortion of dimensions. Distortion of dimensions is frequently adopted in river models where a different scale ratio for depth is adopted. Such models with vertical scales greater than horizontal scales are called 'vertically exaggerated' models. The distortion of configuration results when the general configuration of the model does not bear a resemblance with its prototype. For example a river model will have distortion of configuration if it is constructed with a bed-slope different from the one given by vertical exaggeration. The material distortion takes place when the physical properties of the corresponding materials in the model and in the prototype do not satisfy the similitude conditions. The material distortion may have to be adopted in the river models constructed for the studies of sediment transport. Also it may not be possible to obtain similitude in respect of certain uncontrollable hydraulic quantities such as time, discharge etc., which may lead to distortion of hydraulic quantities.

Some of the typical examples for which distorted models are required to be prepared are rivers, dams across very wide rivers, harbours, estuaries etc., for which the horizontal dimensions are large in proportion to the vertical ones. Obviously, it is on account of certain limitations imposed, it becomes necessary to design distorted models. The following are some of the reasons for adopting distorted models:

(1) To maintain accuracy in vertical measurements.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(2) To maintain turbulent flow.

(3) To obtain suitable bed material and its adequate movement.

(4) To obtain suitable roughness condition.

(5) To accommodate the available facilities such as space, money, water supply and time.

17.14 MERITS AND LIMITATIONS OF DISTORTED MODELS

The merits of distorted models may be summed up as follows:

(a) The vertical exaggeration results in steeper water surface slopes and magnification of wave heights in models, which can therefore be measured easily and accurately.

(b) Due to exaggerated slopes, the Reynolds numbers of a model is considerably increased and the surface resistance is lowered. This assists in simulation of the flow conditions in the model and the prototype.

(c) In case of distorted models sufficient tractive force can be developed to produce adequate bed movement with a reasonably small model.

(d) Model size can be sufficiently reduced by its distortion, thereby its operation is simplified and also cost is lowered considerably.

Besides the advantages accruing from distortion as indicated above, there are certain *limitations* of distorted models, which are as listed below:

(a) The magnitude and distribution of velocities are incorrectly reproduced because vertical exaggeration causes distortion of lateral distribution of velocity and kinetic energy.

(b) The pressures may not be correctly reproduced in magnitude and direction.

(c) Some of the flow details may not be correctly reproduced because distortion increases longitudinal slopes of model streams thus tending to upset flow regime at a point where artificial model roughness is required to restore it.

(d) Slopes of river bends, earth cuts and dikes are often so steep that they cannot be moulded satisfactorily in sand or other erodible material.

(e) A model wave may differ in type and possibly in action from that of the prototype.

(f) There is an unfavourable psychological effect on the observer.

Although distorted models have a number of limitations, yet if judicious allowances are made in the interpretation of the results obtained from such models, useful information can be obtained, which is not possible otherwise.

17.15 SCALE EFFECT IN MODELS

If complete similitude does not exist between a model and its prototype there will be some discrepancy between the results obtained from the model tests and those which will be indicated by the prototype after its construction. This discrepancy or disturbing influence is called *scale effect*. Some of the factors which may lead to the departure from complete similarity and hence result in the development of the scale effect are as indicated below.

In the case of certain problems if several forces have predominance, the complete similitude will be ensured only if all the pertinent model laws are simultaneously satisfied. However, as indicated below, it is quite difficult to satisfy all the model laws involved in the phenomenon and hence in such cases complete similarity cannot be achieved. Under such circumstances the variables which may be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

considered to have secondary influence on the phenomenon are neglected, so that the number of the model laws to be satisfied is reduced. But by neglecting these variables some discrepancy or scale effect would be developed between the results obtained from the model tests and those of the prototype. For instance if in a particular problem, both gravity and viscous forces are predominant, then for complete similarity to exist between the model and prototype, both Froude as well as Reynolds model laws should be satisfied simultaneously. For this the necessary condition to be satisfied may be obtained by combining Eqs 17.15 and 17.18 as

$$(L_r)^{3/2} = \upsilon_r \qquad \text{... (17.25)}$$

Equation 17.25 indicates that once the model scale L_r is selected, a liquid of appropriate viscosity as required by Eq. 17.25 must be found for the model testing; or if a particular liquid is to be used for the model testing then the model scale L_r has to be in accordance with Eq. 17.25. Further if for the model testing the liquid same as that for the prototype is to be used then $L_r = 1$, i.e., the model must be as large as the prototype. It is, however, impossible to find liquids of any desired viscosity and also models with odd scales or full scale models are quite impractical. Under such circumstances the investigator is forced to construct and operate the model in accordance with only one of two model laws, say Froude model law, believed to be more significant. But it would however result in developing the scale effect, which will invalidate the transferring of the results of the model tests to the prototype, without correcting the same.

The scale effect may also be developed in cases where the forces which have practically no effect on the behaviour of the prototype, significantly affected the behaviour of its model. Such a situation may arise in the case of flow over weir models wherein it is apparent that if the head is less than about 15 mm, surface tension plays a proportionately greater part than would be the case in the full size weir having a larger head than in the model.

Often it may not be possible to correctly simulate all the conditions (e.g., roughness) in the model, as that of the prototype. This may also result in developing scale effect if any of these conditions has a pronounced effect on the phenomenon.

In order to detect the presence of such disturbing influences the proposed work may be tried in models with different scales and the resulting scale effects judged from the comparative results so obtained. Besides this the observations collected on models constructed to different scales will also provide an empirical relationship between scale effect and size of model, which may be utilized to correct the results of the model tests.

17.16 APPLICATION OF DYNAMIC SIMILARITY TO SPECIFIC MODEL INVESTIGATIONS

1. Submerged Objects. As an example consider the testing of an aeroplane model in a wind tunnel in order to find the drag F (that is the resistance to its motion) on the prototype. In the case of this phenomenon only viscous and inertia forces are involved. Gravity forces may be disregarded because the fluid concerned being air has very low density and the vertical movements of the air particles as they flow past the aeroplane are small, so that the work done by gravity is negligible. Surface tension is not involved because there is no interface between a liquid and another fluid. Further if it is assumed that it is a low-speed type aeroplane then the effects of compressibility are also negligible. Thus the following functional relationship is obtained for the drag F.

$$F = \phi(V, \rho, \mu, L)$$

where ρ and μ respectively represent the density and viscosity of the fluid used, V represents the relative velocity between the aeroplane (prototype or model) and the fluid and L is some characteristic length.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By adopting the usual method of dimensional analysis the above functional relationship can be expressed in terms of dimensionless parameters as

$$\frac{F}{\rho L^2 V^2} = \phi\left(\frac{\rho VL}{\mu}\right) = \phi(\text{Re}) \qquad \ldots(17.26)$$

Equation 17.26 is true for both model and prototype. For true dynamic similarity between model and prototype (which are of course geometrically similar) the ratio of force magnitudes at corresponding points must be the same for both, and so, for this case the Reynolds number must be same. Consequently the function ϕ (Re) has the same value for both prototype and model and so $\frac{F}{\rho L^2 V^2}$ is the same in each case. Thus using suffix m for the model and suffix p for the prototype, we may write

$$\frac{F_m}{\rho_m L_m^2 V_m^2} = \frac{F_p}{\rho_p L_p^2 V_p^2} \qquad \ldots(17.27)$$

This expression is valid only if the test on the model is carried out under such conditions that Reynolds number is the same as for the prototype. Then

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_m V_p L_p}{\mu_p}$$

or

$$V_m = V_p\left(\frac{L_p}{L_m}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{\mu_m}{\mu_p}\right) \qquad \ldots(17.28)$$

The velocity V_m represented by Eq. 17.28 is known as the *corresponding velocity*. Only when the model is tested at the corresponding velocity as given by Eq. 17.28, the drag F_p for the prototype aeroplane can be predicted from the measured drag F_m for the model aeroplane by using Eq. 17.27.

It may however be stated that if in such problems the effects of compressibility are also important for the prototype, then the Mach numbers must also be indentical in addition to the Reynolds numbers for complete similarity to be established between the model and the prototype. Thus, we have

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

and

$$\frac{V_m}{\sqrt{K_m/\rho_m}} = \frac{V_p}{\sqrt{K_p/\rho_p}}$$

For both the above noted conditions to be satisfied simultaneously

$$\left(\frac{L_p}{L_m}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{\mu_m}{\mu_p}\right) = \frac{V_m}{V_p} = \left(\frac{K_m/\rho_m}{K_p/\rho_p}\right)^{1/2}$$

However, on account of the range of ρ, μ and K for available fluids being limited it is not possible to satisfy the above noted condition. In such circumstances one of the forces is regarded as having secondary effect, so that it may be neglected and the model tests may be carried out in accordance with

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the remaining model law. This would however result in introducing the scale effect which will have to be accounted for before making any predictions about the prototype behaviour.

2. Partially Submerged Objects. Another interesting example of model testing which is also of wide engineering interest is to find the drag or the resistance experienced by a ship. The resistance experienced by a ship moving through water is generally due to three causes:

(i) Viscosity,

(ii) Surface waves, and

(iii) Eddies formed in the wake.

Due to the viscosity of the fluid viscous forces are set up between the fluid layers close to the ship and those farther away, which results in causing a frictional resistance to the motion of the ship. This part of the total drag is usually termed as *skin friction*. Further a ship being only partly immersed in water, its motion through water gives rise to waves on the surface. The formation of these waves requires energy and, since this energy must be derived from the motion, the ship experiences an increased resistance to its motion through water due to the surface waves. In addition, as the ship moves forward, part of the flow towards the rear of the ship breaks away from its surface to form a 'wake' of eddying motion. The eddies so formed considerably modify the distribution of pressure round the moving ship, which results in developing an additional resistance to its motion through water. Thus for a ship moving through water the total drag is the sum of friction drag, drag due to surface waves and pressure or form drag. Since the ships are generally streamlined, the pressure or form drag is relatively small and within the usual range of Reynolds number, it can be assumed to remain constant. Moreover, it is usual practice to consider this portion of the total drag along with the drag due to surface waves. The formation of surface waves is associated with gravitational action. As such the resistance to the motion of a ship is affected by viscosity as well as gravity.

The total resistance R encountered by a ship will thus be a function of the velocity V of the ship, the viscosity of water μ and its density ρ, some characteristic length L to specify the size of the ship and the gravitational acceleration g. In this problem the compressibility of liquid is however insignificant and the surface tension forces may also be neglected if the model is not too small. The functional relationship for R may thus be written as follows :

$$R = \phi(V, \mu, \rho, L, g)$$

which according to the method of dimensional analysis can be expressed in terms of dimensionless parameters as

$$\frac{R}{\rho L^2 V^2} = \phi_1\left[\left(\frac{\rho VL}{\mu}\right), \frac{V^2}{gL}\right]$$

that is

$$\frac{R}{\rho L^2 V^2} = \phi_1[\text{Re}, F_r^2] \qquad ...(17.29)$$

The total resistance therefore, depends both on the Reynolds number and on the Froude number. Thus, in this case for complete similarity between the prototype and its model the Reynolds number as well as Froude number must be the same for each, that is

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_p}{\sqrt{g_p L_p}}$$

Since in practice $g_m = g_p$, the combination of the above equations gives the necessary conditions to be satisfied for complete similarity to be obtained in this case as $\left(\frac{L_m}{L_p}\right)^{3/2} = \left(\frac{\upsilon_m}{\upsilon_p}\right)$ or $L_r^{3/2} = \upsilon_r$. However, as stated earlier this condition cannot be satisfied in actual practice. Therefore, a method suggested by W. Froude may be adopted to overcome the difficulty.

The total resistance R experienced by a ship may be assumed to be consisting of two portions viz., (i) the wave resistance R_w due to the action of the waves; and (ii) the frictional resistance R_f due to frictional effects on the wetted surface of the ship. That is

$$R = R_w + R_f$$

A similar subdivision can also be made for the total resistance r of the model. Thus if r_w and r_f represent the wave and frictional resistance respectively, for the model, then

$$r = r_w + r_f$$

The values of the frictional resistances for the ship and the model can be computed by the expressions noted below :

$$R_f = f_p A_p (V_p)^n \quad \text{... (17.30)}$$

$$r_f = f_m A_m (V_m)^n \quad \text{... (17.31)}$$

where f_p and f_m represent the frictional resistance per unit area of a given surface at unit velocity for the ship and the model respectively, n is an arbitrary index, A_p and A_m are the wetted surface areas and V_p and V_m are velocities of the ship and the model respectively. The values of f_p and f_m can be found experimentally by Froude's experiment as discussed in Chapter on 'Flow Through Pipes'.

A geometrically similar model of the ship is then tested by towing it in water at a corresponding speed given by Froude model law viz., $V_m = V_p \left(\frac{L_m}{L_p}\right)^{1/2}$; and total drag r is measured. The wave drag of the model r_w is then found as

$$r_w = (r - r_f)$$

since r_f for the model can be obtained as indicated above. Now for dynamic similarity established between the model and the prototype for wave resistance only (since Froude number is same for both), we have

$$\frac{R_w}{\rho_p L_p^2 V_p^2} = \frac{r_w}{\rho_m L_m^2 V_m^2} \quad \text{... (17.32)}$$

The wave drag R_w for the prototype ship can thus be obtained from Eq. 17.32 as

$$R_w = \frac{\rho_p}{\rho_m}\left(\frac{L_p}{L_m}\right)^2 \left(\frac{V_p}{V_m}\right)^2 r_w$$

But from Froude model law

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{V_p}{V_m} = \left(\frac{L_p}{L_m}\right)^{1/2}$$

$$\therefore \quad R_w = \frac{\rho_p}{\rho_m}\left(\frac{L_p}{L_m}\right)^3 r_w \qquad \ldots(17.33)$$

Thus R_w for the ship may be computed by Eq. 17.33. Similarly R_f for the ship may be computed by Eq. 17.30. The total drag R for the ship may then be obtained by adding the two as

$$R = R_w + R_f$$

(c) Models of Rivers. River models are frequently used to predict the behaviour of the rivers when some works are proposed to be undertaken on the rivers. In such models the Froude law is the basic similitude criteria, but in addition to this the surface roughness must also be given careful consideration. Mostly the river models are of distorted type, which may have either vertical exaggeration or they may have slope exaggeration. In a model with vertical exaggeration the depth scale is exaggerated so that the model has a depth distortion or vertical distortion. On the other hand in a model with slope exaggeration the model is otherwise geometrically similar but it is provided with a relatively larger slope by tilting it. The various scale ratios for river models with such exaggeration may be obtained as indicated below.

The scale relationships for river models are usually based on Manning's formula, according to which

$$V_r = \frac{R_r^{2/3} S_r^{1/2}}{n_r}$$

For a model with vertical exaggeration since the depth scale is exaggerated, let L_r and D_r represent the horizontal length scale ratio and vertical length (or depth) scale ratio respectively. The hydraulic radius R is dependent upon both horizontal and vertical dimensions, so that R_r cannot be uniquely represented by a simple scale ratio. However, as an approximation for wide rivers $R_r \approx D_r$. Also the slope scale ratio $S_r = (D_r/L_r)$. Thus velocity scale ratio may be expressed as

$$V_r = \frac{D_r^{7/6}}{n_r L_r^{1/2}}$$

Further according to Froude law

$$V_r = D_r^{1/2}$$

Thus scale ratio for rugosity coefficient n_r becomes

$$n_r = \frac{D_r^{2/3}}{L_r^{1/2}}$$

The value of n_r can thus be controlled by suitably fixing the scale ratios D_r and L_r. However, for an undistorted model the above expression reduces to

$$n_r = L_r^{1/6}$$

which is rather difficult to be satisfied in actual practice.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The value of the scale ratio for rugosity coefficient n_r can also be controlled by tilting the model which is otherwise geometrically similar. Such models are known as tilted models or the models with slope distortion or slope exaggeration. For such models according to Froude Law and Manning's formula

$$V_r = L_r^{1/2} = \frac{R_r^{2/3} S_r^{1/2}}{n_r} = \frac{L_r^{2/3} S_r^{1/2}}{n_r}$$

or $$n_r = L_r^{1/6} S_r^{1/2}$$

Thus by suitably selecting the slope and length scale ratios the corresponding value of n_r can be obtained from the above expression.

The river models may be classified as (i) *fixed bed models*; and (ii) *movable bed models*. The fixed bed models are those which are built with rigid beds and banks which cannot be eroded. Such models are useful in investigating the problems related to the stretches of rivers having relatively rigid bed and banks so that the movement of sediment is almost negligible or the same is not of much significance. For designing such models the above derived expressions may be used to compute the relevant dimensions of the models.

The movable bed models have both the beds and banks erodible. These models are useful in investigating the problems involving scour, transportation and deposition of the sediment through the action of flowing water. These are the most difficult type of models to design and operate. Moreover, in order to obtain satisfactory results from such models a lot of experience is needed.

The movement of the sediment depends on the tractive force exerted on the river bed by the flowing water. In an undistorted model the tractive force is extremely small. Therefore movable bed models are invariable distorted to develop steep water surface slopes, thereby providing the required tractive force. This requirement may also be accomplished by adopting an exaggerated vertical scale in conjunction with the exaggerated slope scale as indicated earlier. For this purpose different scales are examined in the model with a given material and the one which gives a general bed movement for the model discharge corresponding to which bed movement occurs in the prototype is adopted. The necessary tractive force may however be reduced by using light weight, easily transportable materials with specific gravity only slightly in excess of unity. Some of the substances used in movable bed models for this purpose are coal dust, lignite, saw dust, pumice stone, plexi glass or plastic sand, lac and lac products and powdered bakelite. The effect of these lighter materials is to lessen the need for extreme distortion of the model and also to decrease the time required to complete the experimental investigation.

In addition to this, since the flow in the prototype rivers is mostly turbulent the model scale should be such that in the model also the flow is turbulent even for the lowest discharge. This can be ensured if the Reynolds number for the flow in the model is sufficiently high. A simplified form of this criterion as specified by the U.S. Waterways Experiment Station, Vicksburg is that the flow in the model may be considered as turbulent as long as the value of the parameter VR is more than 0.007, where V is the velocity of flow in m/s and R is hydraulic mean depth in metres. The flow will however be laminar if VR is less than 0.002 and for the value of VR between 0.002 and 0.007 the flow will be in transition.

Moreover, the state of flow prevailing in a model can also be detected by means of another parameter called Karman number or shear (or friction or roughness) Reynolds number which is defined as

$$R_k = \frac{V_* k}{\upsilon} = \frac{k\sqrt{gRS}}{\upsilon}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where R is hydraulic mean depth, S is bed slope, $V_* = \sqrt{gRS}$ is shear velocity, k is height of equivalent sand grain roughness and υ is kinematic viscosity. If the value of Karman number is more than 50 or 60 for the sand grain type of roughness and more than 100 for any other type of roughness, rough turbulent flow will prevail in the model.

ILLUSTRATIVE EXAMPLES

Example 17.1. *The pressure drop 'Δp' in a pipe of diameter D and length l depends on mass density ρ and viscosity μ of the flowing fluid, mean velocity of flow V and average height k of roughness projections on the pipe surface. Obtain a dimensionless expression for Δp. Hence show that*

$$h_f = \frac{flV^2}{2gD}$$

where h_f is the head loss due to friction $\left(= \frac{\Delta p}{w}\right)$, w is the specific weight of the fluid and f is coefficient of friction.

Solution

The functional equation for the pressure drop Δp can be expressed as

$$\Delta p = \phi (V, l, D, \mu, k, \rho)$$

which in its general form may be written as

$$\phi(\Delta p, V, l, D, \mu, k, \rho) = C$$

Thus the number of variables $n = 7$, and all the variables may be completely described by 3 primary dimensions. Hence $m = 3$. Therefore, the functional relationship can be expressed in terms of $(7 - 3) = 4$ non-dimensional π terms, i.e.,

$$\phi_1 (\pi_1, \pi_2, \pi_3, \pi_4) = C_1$$

Let V, D and ρ be the repeating variables chosen in this case, then the π-terms are grouped as follows:

$$\left.\begin{aligned} \pi_1 &= V^{a_1} D^{b_1} \rho^{c_1} \Delta p \\ \pi_2 &= V^{a_2} D^{b_2} \rho^{c_2} l \\ \pi_3 &= V^{a_3} D^{b_3} \rho^{c_3} k \\ \pi_4 &= V^{a_4} D^{b_4} \rho^{c_4} \mu \end{aligned}\right\}$$

Expressing dimensionally in terms of M-L-T system

$$\pi_1 = (\mathrm{M^\circ L^\circ T^\circ}) = \left(\frac{\mathrm{L}}{\mathrm{T}}\right)^{a_1} (\mathrm{L})^{b_1} \left(\frac{\mathrm{M}}{\mathrm{L}^3}\right)^{c_1} \left(\frac{\mathrm{M}}{\mathrm{LT}^2}\right)$$

Equating the exponents of M, L and T on both sides, we have

for $\mathrm{M}: 0 = c_1 + 1$

for $\mathrm{L}: 0 = a_1 + b_1 - 3c_1 - 1$

for $\mathrm{T}: 0 = -a_1 - 2$

From these equations we get

$$a_1 = -2, b_1 = 0, c_1 = -1$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence $$\pi_1 = \left(\frac{\Delta p}{\rho V^2}\right)$$

By adopting the same procedure, we have

$$\pi_2 = \left(\frac{l}{D}\right)$$

$$\pi_3 = \left(\frac{k}{D}\right)$$

and $$\pi_4 = \left(\frac{\mu}{\rho VD}\right)$$

Thus the following expression for Δp in terms of non-dimensional parameters is obtained

$$\frac{\Delta p}{\rho V^2} = \phi\left(\frac{l}{D}, \frac{k}{D}, \frac{\mu}{\rho VD}\right)$$

which may also be written as

$$\Delta p = (\rho V^2)\phi\left(\frac{l}{D}, \frac{k}{D}, \frac{\rho VD}{\mu}\right)$$

From experimental observations it is indicated that the drop in pressure Δp is proportional to the length l. Therefore

$$\Delta p = \left(\rho V^2 \times \frac{l}{D}\right)\phi\left(\frac{k}{D}, \frac{\rho VD}{\mu}\right)$$

or $$\frac{\Delta p}{w} = \left(\frac{\rho V^2 l}{wD}\right)\phi\left(\frac{k}{D}, Re\right)$$

or $$\frac{\Delta p}{w} = \left(\frac{\rho V^2 l}{2\rho gD}\right)\left[2\phi\left(\frac{k}{D}, \text{Re}\right)\right]$$

or $$\frac{\Delta p}{w} = h_f = \frac{flV^2}{2gD}$$

where f is friction factor which may be expressed as

$$f = \phi\left(\frac{k}{D}, \text{Re}\right)$$

Example 17.2. *(a) Show by method of dimensional analysis that the resistance R to the motion of a sphere of diameter D moving with uniform velocity V through a fluid having density ρ and viscosity μ may be expressed as*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$R = (\rho D^2 V^2)\,\phi\left(\frac{\mu}{\rho VD}\right)$$

Also show that the above expression reduces to $R = k\mu VD$ when the motion is through viscous fluid at low velocity, where k is a dimensionless constant.

(b) Find the viscosity in poise of a liquid through with a steel ball of diameter 1 mm falls, with a uniform velocity of 20 mm/s. The specific gravity of the liquid is 0.91 and that of the steel is 7.8. Given that $k = 3\pi$.

Solution

(a) The functional relationship for R may be expressed as

$$R = f(\mu, \rho, D, V)$$

which by Rayleigh's method may be expressed in an exponential form

$$R = k(\mu^a \rho^b D^c V^d)$$

where k is a dimensionless constant.

Now considering the dimensions for each variable on both the sides of the equation in M-L-T system, we get

$$\left[\frac{ML}{T^2}\right] = [M^\circ L^\circ T^\circ]\left[\frac{M}{LT}\right]^a \left[\frac{M}{L^3}\right]^b [L]^c \left[\frac{L}{T}\right]^d$$

For dimensional homogeneity the exponents of each dimension of both sides of the equation must be identical. Thus

for $\quad M: 1 = a + b \quad \therefore\ b = (1 - a)$

for $\quad T: -2 = -a - d \quad \therefore\ d = (2 - a)$

for $\quad L: 1 = -a - 3b + c + d \quad \therefore\ c = (2 - a)$

Therefore by substitution, we get

$$R = k\left[\mu^a \rho^{(1-a)} D^{(2-a)} V^{(2-a)}\right]$$

$$= k(\rho D^2 V^2)\left(\frac{\mu}{\rho DV}\right)^a$$

This may be expressed in the functional form as

$$R = (\rho D^2 V^2)\,\phi\left(\frac{\mu}{\rho DV}\right)$$

For viscous flow, $R \sim V$, hence taking $a = 1$, we get

$$R = k(\rho D^2 V^2)\left(\frac{\mu}{\rho DV}\right) = k\mu VD.$$

(b) Given $D = 1$ mm; $V = 20$ mm/s; $k = 3\pi$

For uniform velocity of the sphere, resistance R is equal to the gravitational force.

$\therefore$ By substitution, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$3\pi \times \mu \times 20 \times 10^{-3} \times 1 \times 10^{-3} = \pi \left(\frac{1}{2} \times 10^{-3}\right)^3 (7.8 - 0.91)\ 9810$$

$$\therefore \quad \mu = \frac{4(0.5)^3 (6.89 \times 9.81)}{3 \times 3 \times 20}$$

$$= 0.187\ 8\ \text{N.s/m}^2$$

$$= 1.878 \text{ poise.}$$

Example 17.3. *Find the form of the equation for discharge Q through a sharp-edged triangular notch assuming Q depends on the central angle α of the notch, head H, gravitational acceleration g and on the density ρ, viscosity μ and surface tension σ of the fluid.*

Solution

The functional relationship for the discharge Q may be written as

$$Q = \phi(\rho, H, g, \mu, \sigma, \alpha)$$

which in its most general form may be written as

$$\phi(Q, \rho, H, g, \mu, \sigma, \alpha) = C$$

Thus in the present case the number of variables $n = 7$ and these can be completely described by three fundamental dimensions. Hence $m = 3$ and $(n–m) = 4$.

Thus there are 4 dimensionless π-groups in terms of which the above functional relationship may be expressed.

Choosing ρ, g and H as repeating variables, we have

$$\pi_1 = \rho^{a_1}\ g^{b_1}\ H^{c_1}\ Q; \quad \pi_2 = \rho^{a_2}\ g^{b_2}\ H^{c_2}\ \mu$$

$$\pi_3 = \rho^{a_3}\ g^{b_3}\ H^{c_3}\ \sigma; \quad \pi_4 = \alpha$$

since α itself is a dimensionless quantity

$$\pi_1 = \rho^{a_1}\ g^{b_1}\ H^{c_1}\ Q$$

Inserting fundamental dimensions

$$[\text{M}^\circ\ \text{L}^\circ\ \text{T}^\circ] = \left[\frac{\text{M}}{\text{L}^3}\right]^{a_1} \left[\frac{\text{L}}{\text{T}^2}\right]^{b_1} [\text{L}]^{c_1} \left[\frac{\text{L}^3}{\text{T}}\right]$$

Equating the exponents of M, L and T, we get

for $\quad \text{M}: 0 = a_1$

for $\quad \text{L}: 0 = -3a_1 + b_1 + c_1 + 3$

for $\quad \text{T}: 0 = -2b_1 - 1$

from which $\quad a_1 = 0, b_1 = -\frac{1}{2}, c_1 = -\frac{5}{2}$

$$\therefore \quad \pi_1 = \left(\frac{Q}{g^{1/2} H^{5/2}}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Similarly, we have

$$\pi_2 = \rho^{a_2} \; g^{b_2} \; H^{c_2} \; \mu$$

or $$[M^\circ L^\circ T^\circ] = \left(\frac{M}{L^3}\right)^{a_2} \left(\frac{L}{T^2}\right)^{b_2} (L)^{c_2} \left(\frac{M}{LT}\right)$$

Equating the exponents of M, L, and T, we get

for $$M: 0 = a_2 + 1$$

for $$L: 0 = -3a_2 + b_2 + c_2 - 1$$

for $$T: 0 = -2b_2 - 1$$

from which $$a_2 = -1, b_2 = -\frac{1}{2}, c_2 = -\frac{3}{2}$$

$$\therefore \quad \pi_2 = \left(\frac{\mu}{\rho g^{1/2} H^{3/2}}\right)$$

Again, we have

$$\pi_3 = \rho^{a_3} g^{b_3} H^{c_3} \sigma$$

or $$[M^\circ L^\circ T^\circ] = \left[\frac{M}{L^3}\right]^{a_3} \left[\frac{L}{T^2}\right]^{b_3} [L]^{c_3} \left[\frac{M}{T^2}\right]$$

Equating the exponents of M, L and T, we get

for $$M: 0 = a_3 + 1$$

for $$L: 0 = -3a_3 + b_3 + c_3$$

for $$T: 0 = -2b_3 - 2$$

from which $$a_3 = -1, b_3 = -1, c_3 = -2$$

$$\therefore \quad \pi_3 = \left(\frac{\sigma}{\rho g H^2}\right)$$

Thus, we have

$$\left(\frac{Q}{g^{1/2} H^{5/2}}\right) = \left[\left(\frac{\mu}{\rho g^{1/2} H^{3/2}}\right), \left(\frac{\sigma}{\rho g H^2}\right), \alpha\right]$$

or $$Q = CH^{5/2}$$

where $$C = (g^{1/2})\, \phi \left[\left(\frac{\mu}{\rho g^{1/2} H^{3/2}}\right), \left(\frac{\sigma}{\rho g H^2}\right), \alpha\right].$$

Example 17.4. *Assuming that the rate of discharge Q of a centrifugal pump is dependent upon the mass density ρ of fluid, pump speed N (rpm), the diameter of impeller D, the pressure p and the viscosity of fluid μ, show using the Buckingham's π-theorem that it can be represented by*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q = (ND^3)\phi\left[\left(\frac{gH}{N^2D^2}\right),\left(\frac{\upsilon}{ND^2}\right)\right]$$

where H = *head and* υ = *kinematic viscosity of the fluid.*

Solution

From the given data, we have

$$Q = \phi(\rho, N, D, p, \mu)$$

Thus there are 6 variables involved in the phenomenon which are expressible in terms of three fundamental dimensions. Hence the above fundamental relationship may be replaced by another one involving only 3 dimensionless π-terms.

Thus taking ρ, N and D as repeating variables, we have

$$\pi_1 = \rho^{a_1} N^{b_1} D^{c_1} Q$$

$$\pi_2 = \rho^{a_2} N^{b_2} D^{c_2} p$$

and $$\pi_3 = \rho^{a_3} N^{b_3} D^{C_3} \mu$$

Now, $$\pi_1 = \rho^{a_4} N^{b_4} D^{c_4} Q$$

Inserting fundamental dimensions

$$[M^\circ L^\circ T^\circ] = \left[\frac{M}{L^3}\right]^{a_1}\left[\frac{l}{T}\right]^{b_1}[L]^{c_1}\left[\frac{L^3}{T}\right]$$

Equating the exponents of M, L, and T, we obtain

for $M: 0 = a_1$

for $L: 0 = -3a_1 + c_1 + 3$

for $T: 0 = -b_1 - 1$

from which $a_1 = 0, b_1 = -1, c_1 = -3$

$$\therefore \quad \pi_1 = \left(\frac{Q}{ND^3}\right)$$

By adopting the same procedure, we obtain

$$\pi_2 = \left(\frac{p}{\rho N^2 D^2}\right)$$

and $$\pi_3 = \left(\frac{\mu}{\rho ND^2}\right)$$

Therefore

$$\frac{Q}{ND^3} = \phi\left[\left(\frac{p}{\rho N^2 D^2}\right)\left(\frac{\mu}{\rho ND^2}\right)\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

since $p = wH = \rho gH$ and $\frac{\mu}{\rho} = \upsilon$, by substituting these expressions, we get

$$Q = (ND^3)\,\phi\left[\left(\frac{gH}{N^2D^2}\right)\left(\frac{\upsilon}{ND^2}\right)\right]$$

Example 17.5. *The equation for specific speed for a turbine is given by*

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

By π-theorem and using variables such as power P, speed N, head H, diameter D of the turbine, density ρ of the fluid and acceleration due to gravity g, deduce the above expression for N_s.

Solution

From the data given, we have

$$N = f(\rho, gH, P, D)$$

wherein gH called shaft work is considered to be one variable only.

Thus in all there are 5 variables involved in the phenomenon which can be completely described by 3 fundamental dimensions. As such the above functional relationship may be replaced by another one involving only 2 dimensionless π-terms.

Choosing ρ, gH and P as repeating variables, we have

$$\pi_1 = \rho^{a_1}(gH)^{b_1} P^{c_1} N$$

and

$$\pi_2 = \rho^{a_2}(gH)^{b_2} P^{c_2} D$$

Now,

$$\pi_1 = \rho^{a_1}(gH)^{b_1} P^{c_1} N$$

Inserting fundamental dimensions

$$[M^\circ L^\circ T^\circ] = \left[\frac{M}{L^3}\right]^{a_1}\left[\frac{L^2}{T^2}\right]^{b_1}\left[\frac{ML^2}{T^3}\right]^{c_1}\left[\frac{1}{T}\right]$$

Equating the powers of M, L and T, we get

for $M: 0 = a_1 + c_1$

for $L: 0 = -3a_1 + 2b_1 + 2c_1$

for $T: 0 = -2b_1 - 3c_1 - 1$

from which $a_1 = -\frac{1}{2}, b_1 = -\frac{5}{4}, c_1 = \frac{1}{2}$

$$\therefore \quad \pi_1 = \left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right]$$

By adopting the same procedure, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\pi_2 = \left[\frac{D(gH)^{3/4}\rho^{1/2}}{P^{1/2}}\right]$$

Thus, we have

$$\left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right] = \phi\left[\frac{D(gH)^{3/4}\rho^{1/2}}{P^{1/2}}\right]$$

Now if a geometrically similar model of the turbine is considered, then complete similarity between the two will exist when the value of function ϕ is same for both. According to which the value of the parameter on the L.H.S. must be same for the model and its prototype. That is

$$\left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right]_{\text{model}} = \left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right]_{\text{prototype}}$$

The specific speed N_s is defined as the speed of a geometrically similar turbine which when working under a unit head develops unit power.

Thus substituting $P = 1$, $H = 1$ and $N = N_s$ for the model and assuming g and ρ to be same for the model and the prototype, we get

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

Example 17.6. *Water at 15°C flows at 4 m/s in a 150 mm pipe. At what velocity must oil at 30°C flow in a 75 mm pipe for the two flows to be dynamically similar? Take υ for water at 15°C as 1.145×10^{-6} m^2/s and that for oil at 30°C as 3.0×10^{-6} m^2/s.*

Solution

Since for the flow of fluids through pipes only viscous and inertia forces are predominent, Reynolds model law is the criterion for similarity. Thus

$$\left(\frac{Vd}{\upsilon}\right)_m = \left(\frac{Vd}{\upsilon}\right)_p$$

By substitution, we get

$$\frac{4\times150\times10^{-3}}{1.145\times10^{-6}} = \frac{V\times75\times10^{-3}}{3.0\times10^{-6}}$$

$$\therefore \quad V = 20.96 \text{ m/s}$$

Example 17.7. *(a) Show that the total resistance R of a floating ship is given by the expression*

$$R = \frac{1}{L_{r^3}}\left[r + r_f\left(\frac{f_s}{f_m} - 1\right)\right]$$

where L_r is the length scale ratio, r is total resistance for the model, r_f is the frictional resistance of the model, and f_s and f_m are frictional coefficients for the ship and its model respectively. It may be assumed that the velocity index is 2 and the ship and model are towed in the same water.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*b*) *Hence determine the power required to drive a ship 85 m long, having a wetted surface 1024 m*^{2} *and moving at 15 m/s, if the resistance of its 1:16 model at the corresponding speed is 276.64 N and* $f_s = 3.53$ *and* $f_m = 3.92$. *It may also be assumed that both are towed in the same water and only 60% of power is available for propulsion.*

Solution

We have

$$R = R_w + R_f$$

and

$$r = r_w + r_f$$

Also $R_f = f_s A_p V_p^2$ and $r_f = f_m A_m V_m^2$

since $n = 2$.

Thus, we have

$$\frac{R_f}{r_f} = \frac{f_s}{f_m}\left(\frac{A_p}{A_m}\right)\left(\frac{V_p}{V_m}\right)^2$$

According to Froude model law

$$\frac{V_m}{V_p} = \sqrt{\frac{L_m}{L_p}} = (L_r)^{1/2} \text{ and } \frac{A_m}{A_p} = (L_r)^2$$

$\therefore$ By substitution, we get

$$\frac{R_f}{r_f} = \frac{f_s}{f_m} \times \frac{1}{L_r^3}; R_f = r_f\left(\frac{f_s}{f_m} \cdot \frac{1}{L_r^3}\right)$$

The wave drag of the ship R_w and that of the model r_w can thus be expressed as

$$R_w = R - R_f = R - \left(r_f \frac{f_s}{f_m} \cdot \frac{1}{L_r^3}\right)$$

and

$$r_w = (r - r_f)$$

From Eq. 17.32

$$\frac{R_w}{\rho_p L_p^2 V_p^2} = \frac{r_w}{\rho_m L_m^2 V_m^2}$$

$\therefore$

$$R_w = \frac{\rho_p}{\rho_m}\frac{L_p^2}{L_m^2}\frac{V_p^2}{V_m^2} r_w = \frac{r_w}{L_r^3} = \frac{r - r_f}{L_r^3}$$

since

$$\left(\frac{\rho_p}{\rho_m}\right) = 1$$

Thus equating both the values of R_w obtained above, we get

$$R - \left(r_f \frac{f_s}{f_m}\frac{1}{L_r^3}\right) = \frac{r - r_f}{L_r^3}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \qquad R = \frac{1}{L_r^3}\left[r + r_f\left(\frac{f_s}{f_m} - 1\right)\right]$$

(b) Given: $L_p = 85$ m; $A_p = 1024$ m^2; $L_r = \dfrac{1}{16}$

$V_p = 15$ m/s; $f_s = 3.53$; $f_m = 3.92$

$r = 276.64$ N

$r_f = f_m A_m V_m^2$

$$A_m = A_p L_r^2 = 1024 \times \frac{1}{(16)^2} = 4 \text{ m}^2$$

From Froude model law, we have

$$V_m = V_p (L_r)^{1/2} = 15 \times \left(\frac{1}{16}\right)^{1/2} = 3.75 \text{ m/s}$$

$$\therefore \qquad r_f = 3.92 \times 4 \times (3.75)^2 = 220.5 \text{ N}$$

Thus by substitution in the above expression, we get

$$R = (16)^3 \left[276.64 + 220.5\left(\frac{3.53}{3.92} - 1\right)\right]$$

$$= 1.043 \times 10^6 \text{ N}$$

$\therefore$ Power required to drive the ship

$$= RV \times \frac{100}{60}$$

$$= \frac{1.043 \times 10^6 \times 15 \times 100}{60}$$

$$= 26.075 \times 10^6 \text{ W}$$

$$= 26.075 \times 10^3 \text{ kW}$$

Example 17.8. *A model with length scale ratio, model to prototype, equal to x, of a Mach 2 supersonic aircraft is tested in a wind tunnel, wherein air is maintained at atmospheric temperature and a pressure of y times atmospheric pressure. Determine the speed of the model in the tunnel. Given that the velocity of sound in atmospheric air = z.*

Solution

In the present case

$$Ma = 2 = \frac{V}{C} = \frac{V}{z}; \therefore V = 2z$$

where V is the velocity of the prototype aircraft.

Further according to Boyle's law

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{1}{\rho} = \frac{y}{\rho_m}; \text{ or } \rho_m = y\rho$$

where ρ_m represents the mass density of the compressed air of the wind tunnel.

Dynamic similarity between the model and the prototype will be ensured if Reynolds number is same for both. Thus

$$\left(\frac{\rho VL}{\mu}\right)_m = \left(\frac{\rho VL}{\mu}\right)_p$$

Since absolute viscosity is not affected by pressure changes, $\mu_m = \mu_p = \mu$. Also $L_m = xL_p$; $\rho_p = \rho$ and $\rho_m = y\rho$.

Thus by substitution, we get

$$\frac{(y\rho)(V_m)(xL_p)}{\mu} = \frac{(\rho)(2z)L_p}{\mu}$$

$$\therefore \quad V_m = \left(\frac{2z}{xy}\right)$$

Example 17.9. *A solid sphere of diameter 100 mm moves in water at 5 m/s. It experiences a drag of magnitude 19.62 N (2 kg(f). What would be the velocity of 5 m diameter sphere moving in air in order to ensure similarity? What will be the drag experienced by it? State which law governs the similarity.*

Take $\rho_{water} = 1000\ kg/m^3$; $\rho_{air} = 1.2\ kg/m^3$; $\upsilon_{air} = 13\upsilon_{water}$

Solution

Since the spheres in both the cases are moving in a submerged state, Reynolds model law governs the similarity, according to which

$$\left(\frac{VL}{\upsilon}\right)_m = \left(\frac{VL}{\upsilon}\right)_p$$

$$\therefore \quad V_m = V_p \times \frac{L_p}{L_m} \times \frac{\upsilon_m}{\upsilon_p}$$

$$= 5 \times \frac{5 \times 1000}{100} \times \frac{1}{13}$$

$$= 19.23 \text{ m/s}$$

From Eq. 17.27 the ratio of the drag for the model and the prototype is

$$\frac{F_m}{F_p} = \frac{\rho_m}{\rho_p} \times \left(\frac{L_m}{L_p}\right)^2 \times \left(\frac{V_m}{V_p}\right)^2$$

$$= \frac{1000}{1.2} \times \left(\frac{100}{5 \times 1000}\right)^2 \times \left(\frac{5}{19.23}\right)^2$$

$$= 0.0225$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad F_p = \frac{19.62}{0.0225} = 872 \text{ N}$$

or $$F_p = \frac{2}{0.0225} = 88.89 \text{ kg(f)}$$

Example 17.10. *A proposed ship is to have a length of 125 m at the water line and a wetted surface of 1600 m^2. Its speed is to be 36 km/hour. A 1:25 model of the proposed ship is tested in a towing tank at a velocity corresponding to the wave resistance. The total drag (resistance) of the model was found to be 27 N [2.75 kg(f)]. Calculate the total drag of the prototype. The friction drag (resistance) may be computed by the relation*

$$R_f = \frac{1}{2} C_f \rho A V^2$$

where A is area of wetted surface, C_f is average friction drag coefficient given by the following relations

$$C_f = \frac{0.074}{(Re)^{1/5}} \text{ for } Re < 2 \times 10^7$$

and $$C_f = \frac{0.01}{(Re)^{1/5}} \text{ for } Re > 2 \times 10^7$$

$$\rho_m = 1\,000 \text{ kg/m}^3 \text{ (102 msl/m}^3\text{)}$$
$$\rho_p = 1027 \text{ kg/m}^3 \text{ (104.7 msl/m}^3\text{)}$$
$$\upsilon_m = 1.115 \times 10^{-6} \text{ m}^2\text{/s}; \; \upsilon_p = 1.121 \times 10^{-6} \text{ m}^2\text{/s}$$

Solution

Since the model is tested at a velocity corresponding to the wave resistance, the similarity is based on Froude model law, according to which

$$\left(\frac{V}{\sqrt{gL}}\right)_m = \left(\frac{V}{\sqrt{gL}}\right)_p$$

But $$V_p = \frac{36 \times 1000}{60 \times 60} = 10 \text{ m/s}$$

$$\therefore \quad V_m = V_p \sqrt{\left(\frac{L_m}{L_p}\right)}$$

$$= 10 \sqrt{\left(\frac{1}{25}\right)} = 2 \text{ m/s}$$

The length of the model corresponding to 125 m of the prototype is

$$L_m = \frac{125}{25} = 5 \text{ m}$$

The Reynolds number for the model is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(\text{Re})_m = \frac{V_m L_m}{\upsilon_m}$$

$$= \frac{2 \times 5}{1.115 \times 10^{-6}} = 8.97 \times 10^6$$

Since $(\text{Re})_m < 2 \times 10^7$, the friction drag coefficient for the model is given by

$$(C_f)_m = \frac{0.074}{(Re_m)^{1/5}}$$

$$= \frac{0.074}{(8.97 \times 10^5)^{1/5}} = 0.003\ 01$$

Similarly the Reynolds number for the prototype is

$$(\text{Re})_p = \frac{V_p L_p}{\upsilon_p}$$

$$= \frac{10 \times 125}{1.121 \times 10^{-6}} = 1.115 \times 10^9$$

Since $(\text{Re})_p > 2 \times 10^7$, the friction drag coefficient for the prototype is given by

$$(C_f)_p = \frac{0.01}{(Re_p)^{1/5}}$$

$$= \frac{0.01}{(1.115 \times 10^9)^{1/5}} = 0.000\ 155$$

SI units

The friction drag for the model is given by

$$(\text{Re})_m = \frac{1}{2}(C_f \rho A V^2)_m$$

$$= \frac{1}{2} \times 0.003\ 01 \times 1000 \times \frac{1600}{(25)^2} \times (2)^2$$

$$= 15.411 \text{ N}$$

Therefore, wave resistance for the model is

$$(R_w)_m = (27 - 15.411) = 11.589 \text{ N}$$

From Eq. 17.32, we have

$$\left(\frac{R_w}{\rho L^2 V^2}\right)_m = \left(\frac{R_w}{\rho L^2 V^2}\right)_p$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad (R_w)_p = (R_w)_m \times \frac{\rho_p L_p^2 V_p^2}{\rho_m L_m^2 V_m^2}$$

$$= \frac{11.589 \times 1027 \times (25)^3}{1000}$$

$$= 18.597 \times 10^4 \text{ N}$$

The friction drag for the prototype is given by

$$(R_f)_p = \frac{1}{2}(C_f \rho A V^2)_p$$

$$= \frac{1}{2} \times 0.000\,155 \times 1027 \times 1600 \times (10)^2$$

$$= 1.273 \times 10^4 \text{ N}$$

Hence the total drag (or resistance) for the prototype is

$$R_p = (R_f)_p + (R_w)_p$$

$$= 1.273 \times 10^4 + 18.597 \times 10^4$$

$$= 19.87 \times 10^4 \text{ N}$$

Metric units

The friction drag for the model is given by

$$(R_f)_m = \frac{1}{2}(C_f \rho A V^2)_m$$

$$= \frac{1}{2} \times 0.003\,01 \times 102 \times \frac{1600}{(25)^2} \times (2)^2$$

$$= 1.572 \text{ kg(f)}$$

Therefore, wave resistance for the model is

$$(R_w)_m = (2.75 - 1.572) = 1.178 \text{ kg(f)}$$

From Eq. 17.32, we have

$$\left(\frac{R_w}{\rho L^2 V^2}\right)_m = \left(\frac{R_w}{\rho L^2 V^2}\right)_p$$

$$\therefore \quad (R_w)_p = (R_w)_m \times \frac{\rho_p L_p^2 V_p^2}{\rho_m L_m^2 V_m^2}$$

$$= \frac{1.178 \times 104.7 \times (25)^3}{102}$$

$$= 1.889 \times 10^4 \text{ kg(f)}$$

The friction drag for the prototype is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(R_f)_p = \frac{1}{2}\left(C_f \rho A V^2\right)_p$$

$$= \frac{1}{2} \times 0.000\,155 \times 104.7 \times 1600 \times (10)^2$$

$$= 0.1298 \times 10^4 \text{ kg(f)}$$

Hence the total drag (or resistance) for the prototype is

$$R_p = (R_f)_p + (R_w)_p$$
$$= 0.1298 \times 10^4 + 1.889 \times 10^4$$
$$= 2.0188 \times 10^4 \text{ kg(f)}$$

Example 17.11. *A 1:10 scale model of a submarine moving far below the surface of water is tested in a water tunnel. If the speed of the prototype is 8 m/s, determine the corresponding velocity of water in the tunnel. Also determine the ratio of the drag for the model and the prototype.* $\upsilon_{sea\ water} = 1.121 \times 10^{-6}\ m^2/s$; $\upsilon_{water} = 1.00 \times 10^{-6}\ m^2/s$; $\rho_{sea\ water} = 1027\ kg/m^3$; $\rho_{water} = 1000\ kg/m^3$.

Would this result be true if the prototype were to move close to the surface? Explain.

Solution

Since the submarine is moving far below the surface of water, it is totally submerged and hence the similarity between the model and its prototype can be established on the basis of Reynolds model law according to which

$$\left(\frac{VL}{\upsilon}\right)_m = \left(\frac{VL}{\upsilon}\right)_p$$

$$\therefore \quad V_m = V_p \times \frac{L_p}{L_m} \times \frac{\upsilon_m}{\upsilon_p}$$

$$= 8 \times 10 \times \frac{1.00 \times 10^{-6}}{1.121 \times 10^{-6}}$$

$$= 71.365 \text{ m/s}$$

From Eq. 17.27 the ratio of the drag for the model and the prototype is

$$\frac{F_m}{F_p} = \frac{\rho_m}{\rho_p} \times \left(\frac{L_m}{L_p}\right)^2 \times \left(\frac{V_m}{V_p}\right)^2$$

$$= \frac{1000}{1027} \times \left(\frac{1}{10}\right) \times \left(\frac{71.365}{8}\right)^2$$

$$= 0.775$$

If the prototype were to move close to the surface then it will be subjected to both viscous and wave resistance. Hence for complete similarity between the model and the prototype, the model will have to be tested in accordance with both Reynolds as well as Froude model laws. As such the results obtained above on the basis of model testing done in accordance with only the Reynolds model law will not be true for this case.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 17.12. *In order to estimate the energy loss in a pipeline of 1 m diameter through which kerosene of specific gravity 0.80 and dynamic viscosity 0.02 poise is to be transported at the rate of 2000 litres per second, model tests were conducted on a 0.1 m diameter pipe using water at 20° C. Calculate the discharge required for the model pipe. If the energy head loss in 30 m length of the model pipe is measured as 4.8 m of water, determine the corresponding head loss in the prototype. Also determine the value of Darcy's friction factor for the prototype pipe. Take absolute viscosity of water at 20°C as 1.00 × 10^{-2} poise.*

Solution

In this case the dynamic similarity will be achieved if Reynolds number in the model and the prototype are equal. Thus

$$\left(\frac{\rho VD}{\mu}\right)_m = \left(\frac{\rho VD}{\mu}\right)_p$$

or
$$\frac{V_m}{V_p} = \frac{\rho_p}{\rho_m} \times \frac{\mu_m}{\mu_p} \times \frac{D_p}{D_m}$$

$$= \frac{1000 \times 0.8}{1000} \times \frac{1.0 \times 10^{-2}}{0.02} \times \frac{1}{0.1}$$

$$= 4.0$$

but
$$V_p = \frac{2000 \times 10^{-3}}{\frac{\pi}{4} \times (1)^2} = 2.546 \text{ m/s}$$

$\therefore$ $V_m = (2.546 \times 4) = 10.184$ m/s

Therefore, model discharge

$$Q_m = A_m V_m$$

$$= \frac{\pi}{4} \times (0.1)^2 \times 10.184$$

$$= 0.08 \text{ m}^3/\text{s or } 80 \text{ } l/\text{s}$$

Resistance to flow

$$R = \rho L^2 V^2 \phi \left(\frac{\rho VD}{\mu}\right)$$

As the Reynolds numbers for the model and the prototype are equal the value of the function ϕ will be same for the both. Hence

$$\left(\frac{R}{\rho L^2 V^2}\right)_m = \left(\frac{R}{\rho L^2 V^2}\right)_p$$

But
$$R = \text{pressure drop due to head loss} \times \text{cross-sectional area}$$

$$= (\rho g h_f) \left(\frac{\pi D^2}{4}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where h_f is the head loss. Thus, we have

$$\left(\frac{\rho g h_f \times \frac{\pi D^2}{4}}{\rho L^2 V^2}\right)_m = \left(\frac{\rho g h_f \times \frac{\pi D^2}{4}}{\rho L^2 V^2}\right)_p$$

or
$$\frac{(h_f)_m}{(hf)_p} = \frac{D_p^2}{D_{m^2}} \times \frac{L_m^2}{L_p^2} \times \frac{V_m^2}{V_p^2}$$

As
$$\frac{L_m}{L_p} = \frac{D_m}{D_p} = \frac{0.1}{1}$$

$\therefore$
$$\frac{(h_f)_m}{(h_f)_p} = \left(\frac{1}{0.1}\right)^2 \times \left(\frac{0.1}{1}\right)^2 \times \left(\frac{10.184}{2.546}\right)^2$$
$$= 16.0$$

But $(h_f)_m = 4.8$ m of water for a length of 30 m

$\therefore$
$$(h_f)_p = \frac{4.8}{16} = 0.30 \text{ m of kerosene}$$

The corresponding length of the prototype pipe is

$$L_p = \frac{30 \times 1}{0.1} = 300 \text{ m}$$

$\therefore$ The head loss per metre length of the prototype pipe is

$$\frac{(h_f)_p}{L_p} = \frac{0.30}{300} = 0.001 \text{ m of kerosene per m}$$

According to Darcy's formula, the head loss due to friction for the prototype is given by

$$(h_f)_p = \frac{f_p L_p V_p^2}{2 g D_p}$$

$\therefore$
$$f_p = \frac{2 g (h_f)_p D_p}{L_p V_p^2}$$
$$= \frac{2 \times 9.81 \times 0.30 \times 1}{300 \times (2.546)^2}$$
$$= 0.003\ 027$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In the same manner the friction factor f_m for the model pipe may also be computed and it will be found that $f_m = f_p$. This is so because for smooth pipes (which is an implied assumption in this case) the friction factor depends only on Reynolds number, which is same for both the model and the prototype pipes.

Example 17.13. *An overflow structure 550 m long is designed to pass a flow of 4400 m^3/s. A 1: 20 model of the cross-section of the structure is built in a laboratory channel 0.5 m wide. Calculate the corresponding flow rate for the model if the action of viscosity and surface tension may be neglected. When the model is tested at this flow rate the pressure at a point on the model is observed to be 50 mm of mercury vacuum, how should this be interpreted.*

Solution

This model has been built in a laboratory channel which is 0.5 m wide and has a scale of 1:20 for its cross-section. Obviously in this case the model is a distorted one, with horizontal length scale ratio $L_r = \frac{0.5}{550} = \frac{1}{1100}$, and vertical and transverse length scale ratios being $\frac{1}{20}$(as given).

Further in this case since gravity is the predominant force, according to Froude model law, we have

$$V_r = (D_r)^{1/2} = \left(\frac{1}{20}\right)^{1/2}$$

Now
$$Q_r = \frac{Q_m}{Q_p}$$
$$= A_r V_r$$
$$= (L_r D_r) \times (D_r)^{1/2}$$
$$= \frac{1}{1100} \times \left(\frac{1}{20}\right)^{3/2}$$

But
$$Q_p = 4400 \text{ m}^3/\text{s}$$

$\therefore$
$$Q_m = \frac{4400}{1100 \times (20)^{3/2}}$$
$$= 0.0447 \text{ m}^3/\text{s} = 44.7 \text{ } l/\text{s}$$

If h_m and h_p are the pressure heads at the corresponding points in the model and the prototype, then

$$\frac{h_m}{h_p} = \frac{1}{20}$$

Since at a certain point in the model, $h_m = 50$ mm of mercury vacuum, at the corresponding point in the prototype

$$h_p = (50 \times 20)$$
$$= 1000 \text{ mm of mercury vacuum}$$

From this it may be inferred that if a pressure of such a low magnitude will be allowed to be developed at any point of the prototype structure cavitation will be inevitable (as cavitation may occur even at 580 mm of mercury vacuum). Thus in order to avoid this happening the profile of the model overflow structure should be modified to such an extent that the corresponding pressure at any point on the prototype overflow structure will not drop below the permissible limit.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 17.14. *The performance of a spillway of a power project is to be studied by means of a model constructed to a scale of 1:9. Neglecting the viscous and surface tension effect, determine:*

(*i*) *rate of flow in the model for a prototype discharge of 1000 m^3/s; and*

(*ii*) *the dissipation of energy in the prototype hydraulic jump, if the jump in the model studies dissipates 294.2 watts (0.4 metric horse power).*

Solution

In this case since gravity is the predominant force, according to Froude model law, we have

$$Q_r = L_r^{5/2} g_r^{1/2}$$

$$L_r = \frac{1}{9}; \text{ and } g_r = 1$$

$$\therefore \quad Q_r = \left(\frac{1}{9}\right)^{5/2} \times 1 = \frac{1}{243}$$

Since $$Q_r = \frac{Q_m}{Q_p}; \text{ and } Q_p = 1000 \text{ m}^3/\text{s},$$

$$\therefore \quad Q_m = \frac{1000}{243} = 4.115 \text{ m}^3/\text{s}$$

Again according to Froude model law the scale ratio for power is given by

$$P_r = \rho_r L_r^{7/2} g_r^{3/2}$$

Since $P_r = (P_m/P_p)$; $\rho_r = 1$; $g_r = 1$;

and $P_m = 294.2$ watts

By substitution, we get

$$\frac{294.2}{P_p} = 1 \times \left(\frac{1}{9}\right)^{7/2} \times (1)^{3/2}$$

$$\therefore \quad P_p = 294.2 \times (9)^{7/2} \text{ watts}$$

$$= 643\ 415 \text{ watts}$$

$$= 643.415 \text{ kilo watts}$$

If $P_m = 0.4$ metric horse power, then

$$\frac{0.4}{P_P} = 1 \times \left(\frac{1}{9}\right)^{7/2} \times (1)^{3/2}$$

$$\therefore \quad P_p = 0.4 \times (9)^{7/2}$$

$$= 874.8 \text{ metric horse power}$$

Example 17.15. *A shallow river is 1500 m wide and the maximum depth of flow in it is 5 m. It carries a discharge of 3000 m^3/s, the velocity of flow being 1.5 m/s. The model of river is constructed to the horizontal scale of 1:800 and the vertical scale of 1:40. If Manning's n for the bed material in the river is 0.025, find the*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

value of n for the bed material of the model. Check whether the flow in the model is turbulent. The hydraulic mean depth may be assumed to be equal to mean depth of flow.

Solution

In the present case we have $B_p = 1500$ m, maximum depth of flow = 5 m, $Q_p = 3000\ m^3/s$, $V_p = 1.5$ m/s, $n_p = 0.025$, $L_r = (1/800)$ and $D_r = (1/40)$.

$$Q_p = A_p V_p = 3000$$

$$\therefore \quad A_p = \frac{3000}{1.5} = 2000\ m^2$$

$\therefore$ Mean depth of flow

$$D_p = \frac{A_p}{B_p} = \frac{2000}{1500} = 1.33\ m$$

Thus $R_p = D_p = 1.33$ m

Further $B_m = \frac{B_p}{800} = \frac{1500}{800} = 1.875$ m

and $D_m = \frac{D_p}{40} = \frac{1.33}{40} = 0.033\ m = R_m$

According to Manning's formula

$$V_r = \frac{1}{n_r} R_r^{2/3} S_r^{1/2}$$

or $$n_r = \frac{R_r^{2/3} S_r^{1/2}}{V_r}$$

Since $V_r = (D_r)^{1/2}$ in accordance with Froude model law, $R_r = D_r$ and $S_r = (D_r/L_r)$

$$n_r = \frac{D_r^{2/3}}{L_r^{1/2}} = \frac{(800)^{1/2}}{(40)^{2/3}} = 2.418$$

$$\therefore \quad n_m = n_p \times n_r = (0.025 \times 2.418) = 0.0605$$

It may however be noted that on account of vertical exaggeration the model will be more rough as compared to the prototype.

Further $V_m = V_r \times V_p = \left(\frac{1}{40}\right)^{1/2} \times 1.5 = 0.237$ m/s

$\therefore$ $V_m R_m = (0.237 \times 0.033) = 0.0078$ which is greater than 0.007, and hence the flow in the model will be turbulent.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. Dimensional analysis is a mathematical technique which makes use of the study of dimensions as an aid to the solution of several engineering problems.
2. In the study of fluid mechanics the uses of dimensional analysis are :
   (i) Testing the dimensional homogeneity of any equation of fluid motion.
   (ii) Deriving equations expressed in terms of non-dimensional parameters to show the relative significance of each parameter .
   (iii) Planning model tests and presenting experimental results in a systematic manner in terms of non-dimensional paraeters, thus making it possible to analyse the complex fluid flow phenomenon.
3. The various physical quantities can be described by a set of quantities which are in a sense independent of each other. These quantities are known as *fundamental or primary quantities.* The primary quantities are mass, length, time and temperature designated by the letters M, L, T and θ respectively. All other quantities such as area, volume, velocity, acceleration, force, energy, power, etc. are termed as *derived quantities* or *secondary quantities*, because they can be expressed in terms of primary quantities. The expression for a derived quantity in terms of primary quantities is called the *dimension* of the physical quantity.
4. An equation is said to be dimensionally homogeneous if the dimensions of the terms on its left hand side are the same as the dimensions of the terms on its right hand side.
5. The two methods of dimensional analysis used are
   (i) Rayleigh Method
   (ii) Buckingham π-method.
   In general if there are n variables involved in a phenomenon then these n variables can be grouped together into $(n–m)$ dimensionless groups (or π-terms) when m is the number of the fundamental dimensions required to express the n variables.
6. A small scale replica of the actual structure or machine is known as the *model* while the actual structure or machine is called *prototype*.
   The model test results can be utilized to obtain in advance the useful information about the performance of the prototype only if there exists a complete similarity between the model and the prototype.
7. For complete similarity to exist between the model and its prototype the three types of sinilarities to be established are
   (i) Geometric similarity
   (ii) Kinematic similarity
   (iii) Dynamic similarity
   Geometric similarity exists between the model and the prototype if the ratios of corresponding length dimensions in the model and the prototype are equal. Such a ratio is defined as scale ratio.
   Kinematic similarity exists between the model and the prototype if (1) the paths of the homologous moving particles are geometrically similar, and (2) if the ratios of the velocities as well as accelerations of the homologous particles are equal. Homologous point means corresponding point in the model and prototype.
   Dynamic similarity exists between the model and the prototype which are geometrically and kinematically similar if the ratio of all the forces acting at homologous points in the model and the prototype are equal. It may be mentioned that when the two systems are geometrically, kinematically and dynamically similar, then they are said to be completely similar of complete similitude exists between the two systems. However, as stated earlier dynamic similarity implies geometric and kinematec similarities and hence if the two systems are dynamically similar, they may be said to be completely similar.
8. In the problems concerning fluid flow the forces acting may be any one or a combination of the several of the following forces
   (i) Inertia forces
   (ii) Friction of viscous forces
   (iii) Gravity forces
   (iv) Pressure forces
   (v) Elastic forces
   (vi) Surface tension forces

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

9. The various force ratios or dimensionless numbers are given as
   (i) Inertia-Viscous force ratio-Reynolds number
   $$\text{Re or} N_R = \frac{\rho VL}{\mu}$$
   (ii) Inertia-Gravity force ratio-Froude number
   $$Fr \text{ or} N_F = \frac{V}{\sqrt{gL}}$$
   (iii) Inerita-Pressure force ratio-Euler number
   $$E_u \text{ or } N_E = \frac{V}{\sqrt{p/\rho}}$$
   (iv) Inerita-Elasticity force ratio-March number
   $$Ma \text{ or } N_M = \frac{V}{\sqrt{K/\rho}} = \frac{V}{C}$$
   (v) Inerita-Surface tension force ratio-Weber number We or $N_W = \frac{V}{\sqrt{\sigma/\rho L}}$
10. Depending on the significance of each of the forces of the different phenomenon the various model laws have been developed which are given as
    (i) Reynolds Model Law : $\frac{\rho_r V_r L_r}{\mu_r} = 1$
    (ii) Froude Model Law : $\frac{V_r}{\sqrt{g_r L_r}} = 1$; or $\frac{V_r}{\sqrt{L_r}} = 1$, *if* $g_r = 1$
    (iii) Euler Model Law : $\frac{V_r}{\sqrt{p_r/\rho_r}} = 1$
    (iv) Mach Model Law : $\frac{V_r}{\sqrt{K_r/\rho_r}} = 1$
    (v) Weber Model Law – $\frac{V_r}{\sqrt{\sigma_r/\rho_r L_r}} = 1$

    where
    V_r = velocity scale ratio ;
    L_r = Length scale ratio ;
    ρ_r = mass density scale ratio ;
    μ_r = viscosity scale ratio ;
    g_r = gravitational acceleration scale ratio ;
    p_r = pressure scale ratio ;
    K_r = bulk modulus of elasticity scale ratio ; and
    σ_r = surface tencion scale ratio .
11. Hydraulic models are classified as
    (i) Undistorted models
    (ii) Distorted models
    An undistorted model is that which is geometrically similar to its prototype. Distorted models are those in which one or more terms of the model are not identical with their counterparts in the prototype.
12. The scale-effect is the discrepancy caused if complete similitude does not exist between the model and its prototype.
13. The models of completely submerged objects such as aeroplanes are governed by Reynolds model law.
14. The models of partially submerged objects such as ships are governed by both Reynolds model law as well as Froude model law.
15. River models are generally of distorted type which may have either vertical exaggeration or may have slope exaggeration.

PROBLEMS

17.1 List all the variables that may influence the motion of a moving body fully submerged in a fluid, and by dimensional analysis derive an expression for resistance of its motion.

[*Ans.* $(R/\rho L^2 V^2) = \phi (\rho LV/\mu)$]

17.2 Obtain expressions for the resistance to motion of a motor boat, assuming it to be (a) entirely due to viscosity, (b) entirely due to waves, (c) due to viscosity and waves, and (d) due to viscosity and surface tension.

[*Ans.* (a) $(R/\rho L^2 V^2) = \phi (\rho LV/\mu)$ (b) $(R/\rho L^2 V^2) = \phi (V^2/gL)$ (c) $(R/\rho L^2 V^2) = \phi \{(\rho LV/\mu, (V^2/gL)\}$ (d) $(R/\rho L^2 V^2) = \phi \{(\rho LV/\mu), (\sigma/\rho V^2 L)\}$]

17.3 State and explain Buckingham's π-theorem. Show by π-theorem that a general equation for discharge Q over a weir of any shape is given by

$$Q = (H^{5/2} g^{1/2}) \phi \left[\left(\frac{\upsilon}{H^{3/2} g^{1/2}}\right), \left(\frac{\sigma}{H^2 \rho g}\right)\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where

H = head over the weir, υ = kinematic viscosity of the liquid, ρ = mass density of the liquid, and σ = surface tension of the liquid. Hence show that discharge over a rectangular weir of crest length L is given by

$$Q = CdLH^{3/2}.$$

17.4 By dimensional analysis show that the torque T on a shaft of diameter d, revolving at speed N in a fluid of viscosity μ and mass density ρ is given by the expression

$$T = (\rho d^5 N^2)\phi\left(\frac{\upsilon}{d^2 N}\right)$$

Use Rayleigh's method. Hence show that power P is given by

$$P = (\rho d^5 N^3)\,\phi\left(\frac{\upsilon}{d^2 N}\right)$$

17.5 The resistance R, to the motion of a supersonic aircraft of length L, moving with a velocity V in air of density ρ, depends on the viscosity μ and bulk modulus of elasticity K of air. Obtain using Buckingham's π-theorem, the following expression for the resistance R.

$$R = (\rho L^2 V^2)\,\phi\left[\left(\frac{\mu}{\rho LV}\right),\left(\frac{K}{\rho V^2}\right)\right]$$

17.6 Torque T exerted on the shaft of a turbine is found to be a function of diameter D, width B and speed N of the turbine runner, density ρ and viscosity μ of the water flowing in the turbine and difference of pressure p at inlet and outlet of turbine. If $p = wH$, where w is specific weight of water and H is the available head on the turbine, show that torque T may be expressed by the equation

$$T = (\rho N^2 D^5)\,\phi\left[\frac{B}{D},\frac{\mu}{\rho D^2 N},\frac{gH}{D^2 N^2}\right]$$

17.7 If the capillary rise h depends on specific weight w, surface tension σ of the fluid and the radius of the tube r, show that

$$\frac{h}{r} = \phi\left(\frac{\sigma}{wr^2}\right).$$

17.8 The efficiency η of a fan depends on density ρ and viscosity μ of fluid, angular velocity ω, diameter D and discharge Q. Obtain a functional relationship for η in terms of dimensionsless parameters. [*Ans.* $\eta = \phi\{(Q/\omega D^3), (\mu/\rho\omega D^2)\}$]

17.9 The performace of a lubricating oil ring depends on the inside diameter of the ring d, shaft speed N, oil discharge Q, density ρ, viscosity μ, surface tension σ and specific weight w of the fluid. Find a functional relationship in terms of dimensionless parameters.

[*Ans.* $\phi\{(Q/Nd^3), (\mu/\rho Nd^2), (\sigma/\rho N^2 d^3), (w/\rho N^2 d)\} = c$]

17.10 Obtain an expression for the critical depth y_C in a triangular channel which depends on discharge Q, gravitational acceleration g and angle of channel θ. [*Ans.* $y_c^5 = (Q^2/g)\,\phi(\theta)$]

17.11 The discharge Q through a rotating machine such as pump, turbine or compressor depends on the shaft work (gH), power supplied P, speed of rotation N, characteristic length D, mass density ρ and viscosity μ of the fluid. Using Buckingham's π-theorem, obtain a set of dimensionless parameters.

[*Ans.* $(Q/ND^3) = \phi\{(P/\rho D^5 N^3), (gH/N^2 D^2), (\mu/\rho ND^2)\}$]

17.12 For laminar flow in a pipe the drop in pressure Δp is a function of pipe length l, its diameter d, mean velocity of flow V, and viscosity of fluid m. Using Rayleigh's method obtain an expression for Δp. [*Ans.* $\Delta p = (\mu V/d)\,\phi\,(l/d)$]

17.13 The drop in pressure Δp due to an obstruction in a pipe depends on the pipe diameter D, average velocity in the pipe V, mass density ρ and viscosity μ of the flowing fluid; and characteristic length of obstruction d. Determine a set of dimensionless parameters.

[*Ans.* $(\Delta p/\rho V^2) = \phi\{(d/D), (\rho VD/\mu)\}$]

17.14 For small model and (small) prototype of surface ships and overflow structures, the actions of gravity, viscosity and surface tension may be of equal significance. For dynamic similarity between model and prototype, what relation must exist between viscosity, surface tension and model scale?

[*Ans.* $(\mu_r L_r^{1/2}/\sigma_r) = 1$]

17.15 Assuming that the same fluid at same temperature is to be used in the model as in the prototype, what is the force ratio, F_r for the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

various similarity criteria (a) Reynolds, (b) Froude, (c) Mach, (d) Weber ?

[*Ans.* (a) 1; (b) L_r^3; (c) L_r^2; (d) L_r]

17.16 A ship 150 m long moves in fresh water at 15°C at 36 km/hour. A 1:100 model of this ship is to be tested in a towing basin containing a liquid of sp. gr. 0.90. What viscosity must this liquid have for both Reynolds and Froude model laws to be satisfied? At what speed must the model be towed? If 117.7 watts is required to tow the model at this speed, what power is required by the ship? Dynamic viscosity of water at 15°C is 1.13×10^{-3} N.s/m^2.

[*Ans.* 1.017×10^{-6} N.s/m^2; 1 m/s; 1.308×10^6 kW]

17.17 A spillway 7.2 m high and 150 m long discharges 2150 m^3/s under a head of 4 m. If a 1:16 model of the spillway is to be constructed, find the model dimensions, head over the model and the model discharge.

[*Ans.* (Height = 0.45 m; Length = 9.375 m; Head = 0.25 m; Discharge =2.1m^3/s]

17.18 A river model constructed with a horizontal scale of 1:5000 and a vertical scale of 1:256 has a discharge scale ratio of $1:2 \times 10^7$. Obtain the scale ratios for velocity and rugosity coefficient. If the value of Manning's n for the prototype is 0.025, find the corresponding value of n for the model.

[*Ans.* $V_r = 0.064$; $n_r = 1.713$; $n_m = 0.043$]

17.19 In order to estimate the frictional head loss in a pipe 1 m in diameter, through which castor oil of sp. gr. 0.96 and dynamic viscosity 9.9 poise, is to be transported at the rate of 5000 litres per second, a test was conducted on a pipe of diameter 50 mm using water at 15°C as the model fluid. Calculate the discharge required for the model pipe. If the head loss in 40 m length of the model pipe is measured as 13.6 mm of water, determine the corresponding head loss in the prototype. Also obtain the value of Darcy's friction factor for the prototype. Given absolute viscosily of water at 15°C = 0.0131 poise.

[*Ans.* 0.318 l/s; 26.35 mm of oil per metre length of pipe; 0.013]

17.20 A spillway model is to be made to a scale of $\frac{1}{25}$ across a flume which is 0.5 m wide. The prototype is 15 m high and the maximum head expected is 2 m.

(a) What height of model and what head on model should be used?

(b) If flow over the model at 60 mm head is 0.02 m^3/s, what flow per metre length of prototype may be expected?

(c) If the model shows a measured hydraulic jump of 30 mm, how high is jump in prototype?

(d) If the energy dissipated in the model at the hydraulic jump is 110.34 watts, what would be the energy dissipated in the prototype?

[*Ans.* (a) 0.6 m, 80 mm; (*b*) 5 m^3/s per m; (c) 0.75 m; (d) 0.862×10^4 kW]

17.21 In an open channel having a rectangular section water flowing at 3 m/s with a depth of 0.6 m rises suddenly at a point in the flow to form a jump so that the depth increases to 1.5 m. If a 1:10 scale model of the jump is to be constructed in a laboratory channel, find the flow rate per unit width of the model channel. What are the Froude numbers upstream and downstream of the jump in model and prototype.

[*Ans.* 0.057 m^3/s/m; Fr_1 =1.24, Fr_2 = 0.31 (same for the model and the prototype)]

17.22 A distorted model of a rectangular channel of best cross-section is made with horizontal and vertical scale ratios L_r and D_r respectively. Show that velocity ratio $V_r = [(D_r^{7/6})/(n_r L_r^{1/2})]$

17.23 A harbour model has a horizontal scale of 1/150 and a vertical scale of 1/60. Storm waves of amplitude 2 m and velocity 10 m/s strike against the breakwater of the harbour. Determine for the model the amplitude and velocity of the waves, neglecting the viscous effects. Further if the time between the successive high tides in the prototype is 12 hours, what should be the tidal period in the model ?

[*Ans.* 0.26 m ; 1.29 m/s ; 37.18 min.]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Fluid Flow Around Submerged Objects–Drag and Lift

Chapter 18

18.1 INTRODUCTION

Several problems involving the flow of fluid around submerged objects are encountered in the various engineering fields. Such problems may have either a fluid flowing around a stationary submerged object, or an object moving through a large mass of stationary fluid, or both the object and the fluid being in motion. Some of the examples which may be quoted are the motion of very small objects such as fine sand particles in air or water, very large objects such as airplanes, submarines, automobiles, ships etc., moving through air or water, and the structures such as buildings, bridges etc., which are submerged in air or water. In the analysis and design of such objects the knowledge of the forces exerted on them by the fluid is of significant importance.

If a body of arbitrary shape, held immersed in a large stationary mass of fluid, is moved with constant velocity through the fluid, the body experiences a force which tends to oppose its motion. The body in turn exerts a force on the fluid, since every action is accompanied by equal and opposite reaction. Now if a uniform velocity equal in magnitude to that of the moving body but opposite in direction is applied to the above system, then the body will be held stationary and the fluid will be moving in the opposite direction. The above indicated superimposition does not have any dynamic effect on the system, because the force exerted on the body or the fluid, results from the relative motion between the immersed body and its surrounding fluid. As such it makes no difference in the magnitude of the force whether the body moves through the stationary fluid or the fluid flows about the immersed body which is at rest.

The force exerted by the fluid on the moving body may in general be inclined to the direction of motion, and hence it has a component in the direction of motion as well as one perpendicular to the direction of motion. The component of this force in the direction of motion is called the *drag* F_D, and the component perpendicular to the direction of motion is called the *lift* F_L, (Fig. 18.1). However, for a symmetrical body, such as sphere or a cylinder, facing the flow symmetrically, there is no lift and thus the total force exerted by the fluid is equal to the drag on the body.

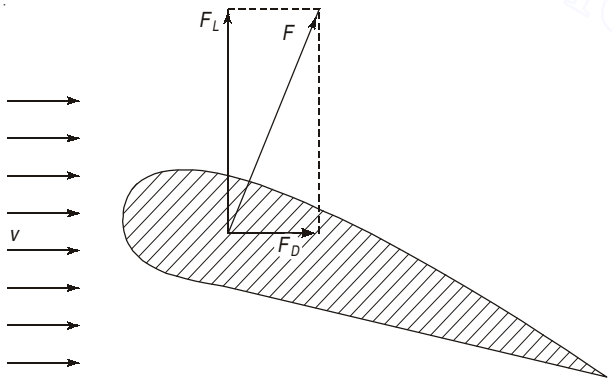


Figure 18.1 *Forces on an immersed body*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

It is known from the principles of hydrodynamics that for a symmetrical body moving through an ideal fluid (i.e., having no viscosity) at a uniform velocity, the pressure distribution around the body is symmetrical and hence the resultant force acting on the body is zero. The real fluids such as air, water etc., possess viscosity, and if a body is moved through these fluids at a uniform velocity, it is observed that the body does experience a resistance to its motion. Hence it may be concluded that the viscosity of the fluids is mainly responsible for causing the drag on the bodies.

Consider a body held stationary in a stream of real fluid moving at a uniform velocity V. The force acting at any point on the small element dA of the surface of the body can be considered to have two components (τdA) and (pdA) acting along the directions tangential and normal to the surface respectively (Fig. 18.2). The tangential components are 'shear forces' and the normal components are 'pressure forces'. The drag on the body is therefore given by the summation of the components of these forces acting over the entire surface of the body in the direction of the fluid motion. The sum of the components of the shear forces in the direction of flow of fluid is called the *friction drag* F_{Df}, which may be expressed as

$$\text{Friction drag} \qquad F_{Df} = \int_A \tau \, dA \cos\theta \qquad \text{...(18.1)}$$

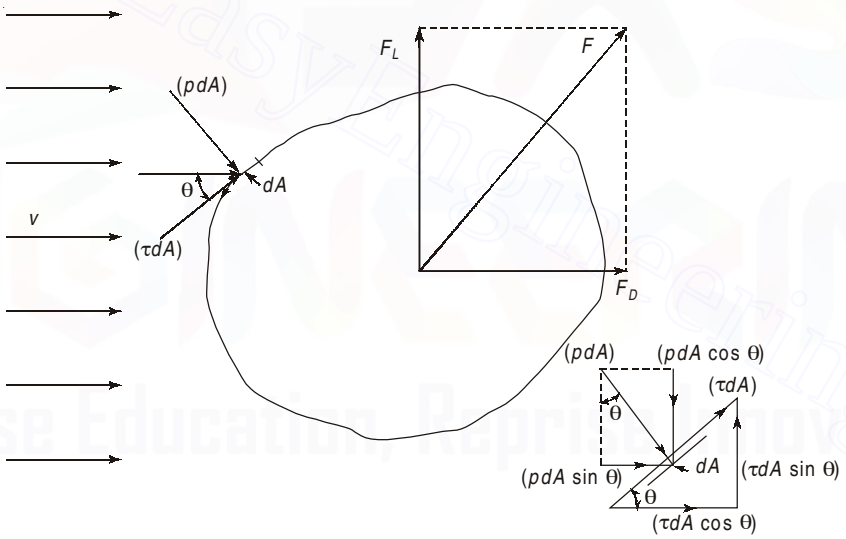


Figure 18.2 *Components of pressure and frictional forces on an element of surface of an immersed body*

Similarly the sum of the components of the pressure forces in the direction of the fluid motion is called the *pressure drag* F_{Dp}, which may be expressed as

$$\text{Pressure drag} \qquad F_{Dp} = \int_A p \, dA \sin\theta \qquad \text{...(18.2)}$$

The *total drag* F_D acting on the body is therefore equal to the sum of the friction drag and the pressure drag. Thus

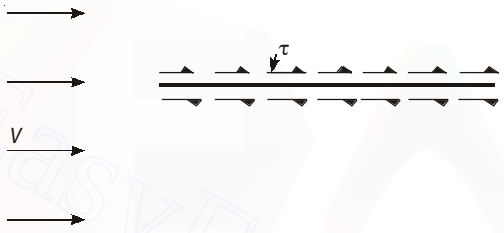
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

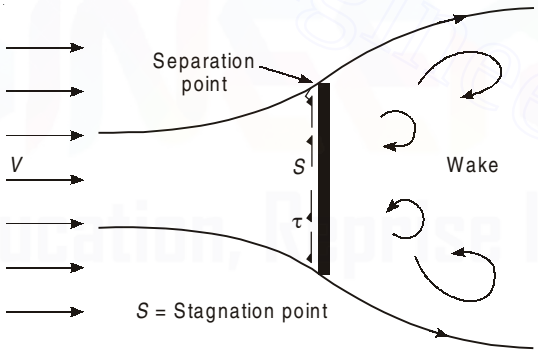
$$F_D = F_{Df} + F_{Dp}$$

$$= \int_A \tau \, dA \cos\theta + \int_A p \, dA \, \sin\theta \qquad ...(18.3)$$

The relative magnitude of the two components of the total drag viz., friction drag and pressure drag, depends on the shape and the position of the immersed body. Thus if a thin flat plate is held immersed in a fluid, parallel to the direction of flow as shown in Fig. 18.3 (a), the pressure drag $\left(\int_A p \, dA \, \sin\theta\right)$ is practically equal to zero. As such in this case the total drag is equal to the friction drag. On the other hand, if the same plate is held perpendicular to the flow as shown in Fig. 18.3 (b), the friction drag



(a) Thin plate parallel to the flow



(b) Thin plate perpendicular to the flow

Figure 18.3 *Flat plate held (a) parallel, and (b) perpendicular to flow*

$\left(\int_A \tau \, dA \, \cos\theta\right)$ is practically equal to zero and the total drag is due to the pressure difference between the upstream and downstream sides of the plate. In between these two extreme cases, there are several body shapes for which the contribution of each of the two components to the total drag varies considerably depending on the shape and position of the immersed body, and the flow and the fluid characteristics.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The lift on the body is given by the summation of the component of the shear and the pressure forces acting over the entire surface of the body in the direction perpendicular to the direction of the fluid motion. Thus, following expression is obtained for the total lift F_L acting on the body shown in Fig. 18.2

$$F_L = \int_A \tau \, dA \sin\theta + \int_A p \, dA \cos\theta \qquad ...(18.4)$$

For a body moving through a fluid of mass density ρ, at a uniform velocity V, the mathematical expressions for the calculation of the drag and the lift may also be written as follows:

$$F_D = C_D A \frac{\rho V^2}{2} \qquad ...(18.5)$$

$$F_L = C_L A \frac{\rho V^2}{2} \qquad ...(18.6)$$

In the above expressions C_D and C_L are known as the drag and the lift coefficients respectively, both of which are dimensionless. The area A is a characteristic area, which is usually taken as either the largest projected area of the immersed body; or the projected area of the immersed body on a plane perpendicular to the direction of flow of fluid. The term $(\rho V^2/2)$ is the dynamic pressure of the flowing fluid. In the case of several objects the area A may be represented in terms of a characteristic length L, in which case Eqs 18.5 and 18.6 may be expressed as

$$F_D = C_D L^2 \frac{\rho V^2}{2} \qquad ...(18.7)$$

$$F_L = C_L L^2 \frac{\rho V^2}{2} \qquad ...(18.8)$$

18.2 TYPES OF DRAG

The existence of viscosity for the real fluids is mainly responsible for causing drag on the bodies. Whenever a real fluid flows past a stationary object held submerged, a layer of fluid, which comes in contact with the surface of the object, adheres to it. Since this layer of fluid cannot slip away from the surface of the object, it undergoes retardation. This condition is however known as *no slip* condition. This retarded layer of fluid further causes retardation for the adjacent layers of the fluid, thereby developing a small region in the immediate vicinity of the solid surface in which the velocity of flowing fluid varies from zero at the surface of the object to the maximum velocity of the flowing fluid. This region is known as 'boundary layer'. In the boundary layer zone since there is a large variation of velocity in a relatively small distance, there exists a fairly large velocity gradient. According to Newton's law of viscosity shearing or tangential stress τ is equal to the product of the viscosity μ and the velocity gradient $(\partial v/\partial y)$. As such in the boundary layer zone even for the fluids of low viscosity, on account of large velocity gradient, considerable shearing stresses are caused. These shearing stresses exert a tangential force on the object, which is known as *surface or friction drag*. The development of boundary layer is, therefore, one of the ways in which the viscosity of the fluid exerts drag on the objects.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The second effect of viscosity is that it causes the fluid particles to undergo deformation. In the case of fluids possessing larger viscosity the deformation of fluid particles takes place in a very wide zone extending from the body in the lateral direction. However, for the fluids having low viscosity the deformation is confined to the zone of boundary layer only. In order to cause the deformation of various fluid particles certain forces are necessarily developed, which offer an additional resistance to the motion. The component, in the direction of fluid motion, of the force so developed is known as *deformation drag*. The deformation drag mainly exists in the case of very small object moving at very small velocities through fluids of large viscosity. It, therefore, means that deformation drag exists when viscous forces are much more predominant than the inertial forces, or in other words when the Reynolds number is very small–less than about 0.10.

The widespread fluid deformation in the lateral direction results in developing a variation of pressure due to which a pressure drag is developed which constitutes the deformation drag. However, the surface or friction drag at the boundary is also included in the deformation drag. As such the deformation drag consists of the surface drag at the boundary and the pressure drag, developed due to variation of pressure caused by the deformation of the fluid particles. The relative magnitude of the surface drag and the pressure drag in constituting the deformation drag, for a given body, is essentially a function of Reynolds number. As the Reynolds number increases the extent of the deformation zone decreases and it is more or less limited to the boundary layer. With the reduction in the size of the deformation zone the contribution of the pressure drag (due to pressure variation in the deformation zone) to the deformation drag becomes almost negligible. The deformation drag then consists of the friction drag only.

The third effect of viscosity is exhibited in the separation of flow. If the surface of the immersed object, along which the boundary layer forms, is such that it curves away from the flow, there exists a tendency for the flowing fluid to leave the boundary. This phenomenon is known as the 'separation of flow', which usually occurs at higher values of Reynolds numbers. With the separation of flow, the flow pattern is considerably modified and hence the pressure distribution is changed. On the downstream side of the body on account of separation a region of low pressure is developed which is known as 'wake'. Since on the upstream side of the body the pressure being considerably high, there exists a pressure difference between the upstream and downstream sides. The pressure difference so created results in producing a drag on the body, which is known as *form drag* or *pressure drag*.

The total drag exerted on any immersed body is, therefore, equal to the sum of the deformation drag and the form drag. As already mentioned, at higher Reynolds number the deformation drag consists of friction drag only, and hence the total drag consists of the friction drag and the form drag. The relative magnitude of the friction drag and the form drag in the total drag essentially depends on the shape and position of the immersed body, and the flow and the fluid characteristics.

The magnitude of the form drag depends upon the size of the wake, which in turn depends upon the shape and position of the immersed body. Consider a circular disc, a sphere and a streamlined body as shown in Fig. 18.4, all of which have the same cross-sectional area, and are immersed in the same turbulent flow. In all these cases there exists a stagnation point *S* (shown by a black dot) on the upstream side of the object at which there is a maximum local pressure.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For the object having sharp edges the point of separation is almost fixed. Thus, in the case of a disc or a plate held normal to the flow the separation of the flow will take place at the edges. A very wide

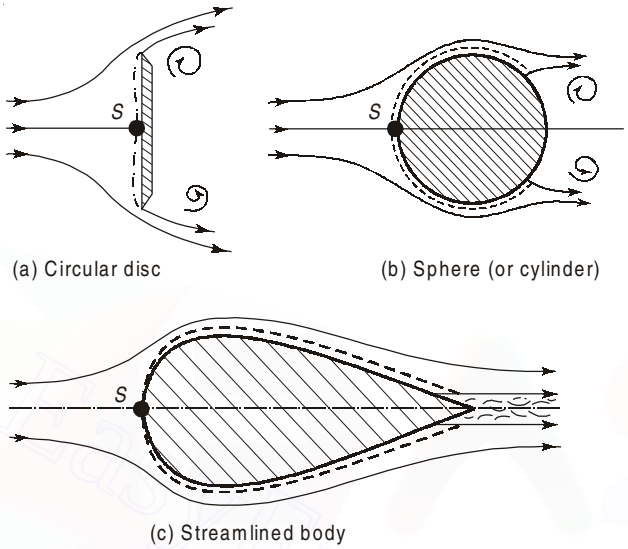


Figure 18.4 *Flow past different objects*

wake is developed on the downstream side of the disc to be exposed to a zone having pressure considerably below that on the upstream side. The result is that a large drag is exerted on the plate which is wholly a pressure or form drag, since none of the shear forces on the disc have components in the original direction of motion.

In the case of well rounded bodies such as a cylinder or a sphere there is no fixed point of separation. The location of the point of separation depends on the Reynolds number. The wake in such cases is smaller than that of the disc. As such the drag of the sphere is considerably smaller than that of the disc. Experiments have shown that the total drag of the sphere is about one-third that of the disc of the same cross-sectional area. However, for the sphere or a cylinder the friction drag is not zero since all the shear stresses acting on the surface will have components parallel to the original direction of motion but the shear stresses are so small that the resulting frictional drag is negligible as compared to the pressure drag. Thus, for such objects also the total drag is practically equal to the pressure drag only.

For a well streamlined body the separation occurs only at the downstream end. As such the wake in this case is extremely small. Hence the pressure drag of such objects is very small fraction of that of the disc. However, the frictional drag of streamlined bodies is considerably large than that of the sphere, since there being more surface area in contact with the flow. Furthermore, for well streamlined objects friction drag is usually larger than pressure drag, but both are so small that their total drag is only about one-fortieth (1/40) of that of the disc.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

18.3 DIMENSIONAL ANALYSIS OF DRAG AND LIFT

Usually it is not possible to predict the total drag on a body merely by analytical methods. As such in almost all the cases the general practice is to determine the total drag on a body experimentally. The planning of the experiments and the analysis of the results obtained therefrom may conveniently be carried out in terms of the dimensional analysis of the problem as described below.

Consider an object of characteristic length L, moving with a velocity V, through a fluid of mass density ρ, viscosity μ and modulus of elasticity K. Then the drag F_D and lift F_L exerted on the body may be expressed in terms of the following functional relationships:

$$F_D = f_1(L, \rho, \mu, V, K, g, \eta) \quad ...(18.9)$$

$$F_L = f_2(L, \rho, \mu, V, K, g, \eta) \quad ...(18.10)$$

where η is a dimensionless shape factor, which represents the effect of the shape of the object on the drag and the lift. By adopting any of the methods of dimensional analysis Eqs 18.9 and 18.10 may be transformed to the following form:

$$\frac{F_D}{\frac{1}{2}\rho L^2 V^2} = f_3\left(\eta, \frac{\rho VL}{\mu}, \frac{V}{\sqrt{K/\rho}}, \frac{V}{\sqrt{gL}}\right) \quad ...(18.11)$$

$$\frac{F_L}{\frac{1}{2}\rho L^2 V^2} = f_4\left(\eta, \frac{\rho VL}{\mu}, \frac{V}{\sqrt{K/\rho}}, \frac{V}{\sqrt{gL}}\right) \quad ...(18.12)$$

Since the left hand side of the Eqs 18.11 and 18.12 are C_D and C_L respectively, it follows that

$$C_D = f_3\left(\eta, \frac{\rho VL}{\mu}, \frac{V}{\sqrt{K/\rho}}, \frac{V}{\sqrt{gL}}\right) \quad ...(18.13)$$

$$C_L = f_4\left(\eta, \frac{\rho VL}{\mu}, \frac{V}{\sqrt{K/\rho}}, \frac{V}{\sqrt{gL}}\right) \quad ...(18.14)$$

Equations 18.13 and 18.14 indicate that the drag and lift coefficients depend on the parameters: shape factor η. Reynolds number $\text{Re}\left(=\frac{\rho VL}{\mu}\right)$, Mach number $Ma\left(=\frac{V}{\sqrt{K/\rho}}\right)$ and Froude number $Fr\left(=\frac{V}{\sqrt{gL}}\right)$. The shape factor η describes the geometry of the objects, and the numbers Re, Ma, and Fr, represent the effect of viscosity and elasticity of the fluid and gravity respectively. In every case all these numbers are never simultaneously predominant. Thus Fr is predominant only if there exists a free surface so that the object remains partly immersed in the liquid and partly outside the liquid i.e., in the air. On the other hand Re and Ma are predominant if the object is completely immersed in a fluid. Out of these two, Re is predominant when the fluid may be considered incompressible and Ma is predominant when compressibility effects of fluid must be considered. However, for most of the problems gravity and compressibility are not significant, then the Eqs 18.13 and 18.14 reduce to the following form :

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$C_D = f_5(\eta, \text{Re}) \quad \ldots(18.15)$$

$$C_L = f_6(\eta, \text{Re}) \quad \ldots(18.16)$$

In the following paragraphs the variation of C_D for objects, such as sphere, cylinder, flat plate, airfoil has been discussed, when these objects are held in an infinite mass of flowing fluid which is incompressible, so that only the viscous and inertia forces are predominant. However, the effect of the free surface (i.e., gravitational acceleration) and that of the compressibility of the flowing fluid on the drag has been separately discussed. In addition the variation of C_L for some of the objects has also been discussed.

18.4 DRAG ON A SPHERE

For an ideal fluid flowing past a sphere (i.e., for irrotational flow) it is possible to obtain analytically the pressure distribution around the sphere. In this case due to the viscosity of the flowing fluid being absent the flow pattern as well as the pressure distribution is symmetrical on the front and the rear of the sphere. As shown in Fig. 18.5 (a), the highest intensity of pressure occurs at the front and the rear stagnation points; and the lowest around the circumference at right angles to the flow. Now if this pressure is integrated over the entire surface of the sphere and its component in the direction of motion is obtained then it will be equal to zero on account of symmetrical pressure distribution. It therefore means that there is no form drag. Furthermore, since there is no viscosity, there is neither deformation drag nor friction drag. It is thus observed that in the case of an ideal fluid flowing past a sphere (or any other object) there is no drag.

However, when a real fluid flows past a sphere the resulting pressure distribution is entirely different from that obtained analytically for the case of an ideal fluid. Such modification in the pressure distribution is evidently due to the viscosity possessed by the real fluids. Let D be the diameter of the sphere, V be the velocity of flow of fluid of mass density ρ and viscosity μ. If the velocity of flow is very small or the fluid is quite viscous such that the Reynolds number $\left(\frac{\rho VD}{\mu}\right)$ is very small, being as low as 0.2 or even less, then the viscous forces are much more predominant than the inertial forces. For the flow of real fluid of an infinite extent, past a sphere at such low Reynolds numbers, G.G. Stokes developed a mathematical solution in which the inertia forces were assumed negligible as compared with the viscous forces. According to this analysis the total drag is

$$F_D = 3\pi\mu VD \quad \ldots(18.17)$$

out of which two-thirds is contributed by skin friction and one-third by the pressure difference (developed on account of deformation), that is

$$\text{Skin friction drag} = \frac{2}{3}F_D = 2\pi\mu VD$$

Pressure drag due to deformation

$$= \frac{1}{3}F_D = \pi\mu VD$$

From Eq. 18.5, we have

$$C_D = \frac{\frac{F_D}{A}}{\frac{1}{2}\rho V^2} = \frac{(3\pi\mu VD)\Big/\frac{\pi D^2}{4}}{\frac{1}{2}\rho V^2} = \frac{24\mu}{\rho VD}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$C_D = \frac{24}{Re} \qquad ...(18.18)$$

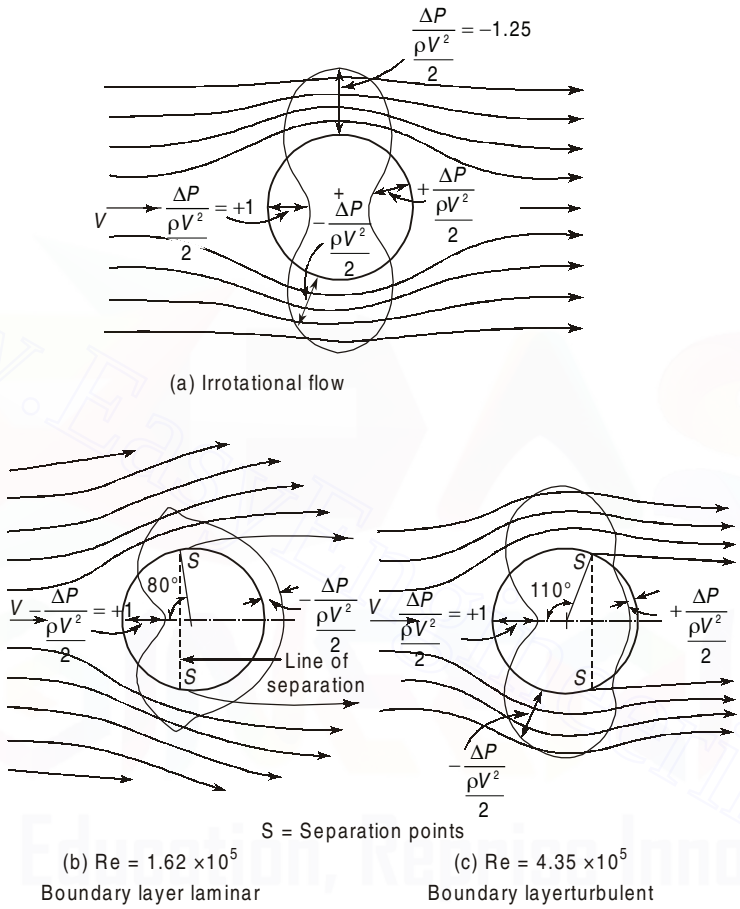


Figure 18.5 *Pressure distribution around a sphere*

Equation 18.17 or 18.18 is generally designated as 'Stokes' law'. Equation 18.18 has been found to agree closely with experimental results for Re ≤ 0.2, provided the fluid is of infinite extent. However, due to the proximity of the boundaries the resistance to the motion is increased and in such a case the drag coefficient is given by the following modified expression

$$C_D = \frac{24}{\text{Re}}\left(1 + 2.1\frac{D}{D_1}\right) \qquad ...(18.19)$$

where D_1 is the smallest lateral dimension of the container and D is the diameter of the sphere.

In 1972 C.W. Oseen a Swedish physicist improved Stokes' analysis by partly taking into account the inertia terms which Stokes had omitted. Oseen's solution which is found to be valid for Re < 1, is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$C_D = \frac{24}{\text{Re}}\left(1+\frac{3}{16}\text{Re}\right) \qquad ...(18.20)$$

However, an empirical relation acceptable upto *Re* equal to about 100 is

$$C_D = \frac{24}{\text{Re}}\left(1+\frac{3}{16}\text{Re}\right)^{1/2} \qquad ...(18.21)$$

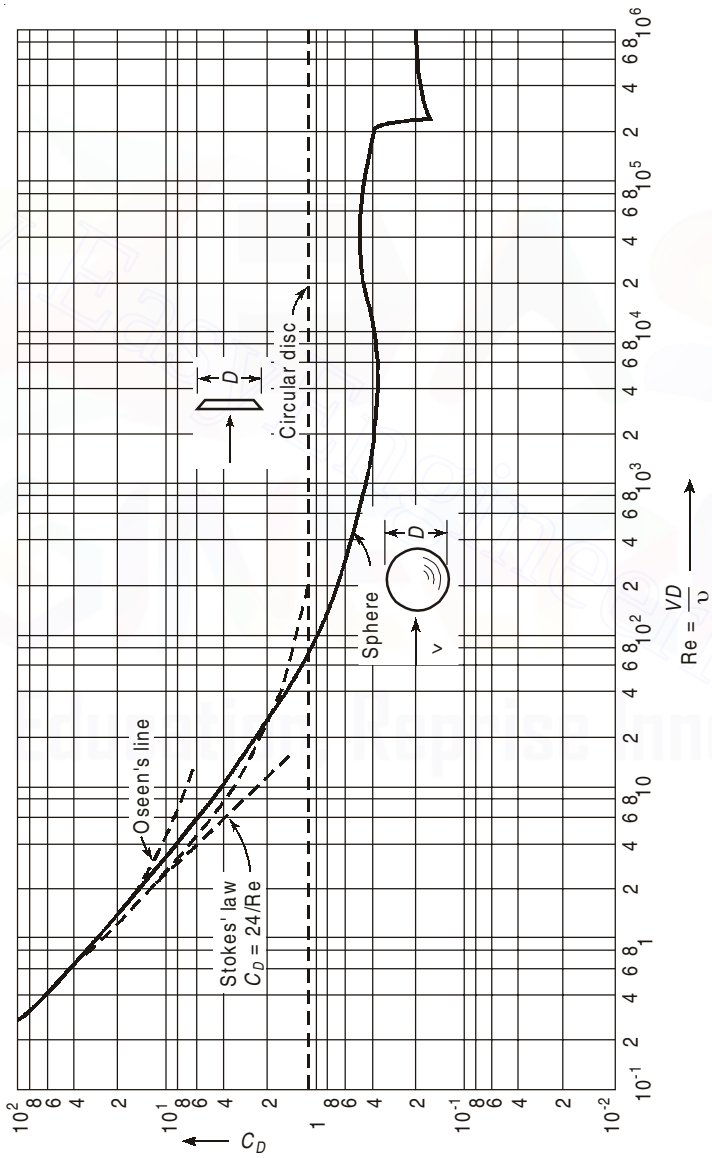


Figure 18.6 *Variation of* C_D *with* Re *for a sphere and a circular disc*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As can be seen from Fig. 18.6, Oseen's solution, however does not follow experimental trend beyond Re equal to 2.

With the increase in the Reynolds number the region in which the influence of viscosity is predominant is considerably reduced and is restricted only to a very small zone of boundary layer formed close to the sphere. The separation of the boundary layer however begins from the downstream stagnation point and the points of separation move further forward towards upstream direction as Re increases, until when Re $\underline{\Omega}$ 1000, a more or less stable position for the points of separation is achieved, which is at about 80° from the upstream stagnation point, as shown in Fig. 18.5 (b). On account of the shifting of the separation points considerably towards the upstream side, a very wide wake is formed on the rear of the sphere, which results in a large contribution of the pressure or (form) drag (about 95%) in the total drag as compared with a relatively small magnitude of skin friction drag (about 5%). Because of this, as may be seen in Fig. 18.6 in the range of Reynolds number from 10^3 to 10^5, C_D becomes more or less independent of Reynolds number. However, in this range of Re, C_D, increases slightly from 0.4 to 0.5 only.

A typical pressure distribution around the sphere corresponding to *Re* equal to 1.62×10^5 is shown in Fig. 18.5 (b). Since upto Re $< 3 \times 10^5$ the boundary layer may be considered to be laminar, in this case the boundary layer is laminar. It will be observed in this case that the pressure distribution around the sphere on the upstream side upto the points of separation is almost the same as obtained by ideal fluid theory. However, beyond the points of separation, the pressure distribution is altogether different from that obtained by the ideal fluid theory.

With a further increase in the Reynolds number, at Re $\underline{\Omega}$ 3×10^5, the boundary layer becomes turbulent which can travel further downstream without separation. As such in this case the points of separation shift considerably to the downstream side, which are now located at about 110° from the front stagnation point as shown in Fig. 18.5 (c). Due to this shift in the points of separation the size of the wake on the rear of the sphere is reduced and the value of C_D drops sharply from 0.5 to 0.2 as shown in Fig. 18.6.

It is thus seen that when the boundary layer changes from laminar to turbulent the drag coefficient is greatly reduced. The transition of the boundary layer from laminar to turbulent takes place at critical Reynolds number which however decreases with the increase in the turbulence of the oncoming fluid and the increase in the surface roughness. Moreover, by increasing the surface roughness the point of transition of the boundary layer from laminar to turbulent is shifted towards the upstream side. This early transition of the boundary layer from laminar to turbulent on the boundary surface would result in shifting the points of separation to the downstream side which in turn would result in the reduction of the size of the wake and hence the reduction of the drag and the drag coefficient. Such reduction in the drag coefficient may be achieved for example in the case of golf and cricket balls. The surface of the golf ball is dented in order to increase its surface roughness which, as explained above, results in a lower drag coefficient for the ball, so that the golf players are able to send the ball to larger distances. Similarly in the case of a cricket ball the seam of the ball (i.e., the joint where the two halves of the ball are stitched together) is very rough as compared to the rest of the surface of the ball. Thus while throwing the ball the seam may be so oriented that the points of separation are shifted considerably to the downstream side thereby reducing the size of the wake and hence reducing the drag coefficient. The bowler will then be able to send the ball to the batsman at a greater speed.

Figure 18.5 (c) shows a typical pressure distribution around the sphere with turbulent boundary layer and corresponding to Re equal to 4.35×10^5. It will be observed that in this case also the pressure distribution around the sphere on the upstream side and more or less upto the points of separation is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

almost the same as obtained by ideal fluid theory. Beyond the points of separation the pressure distribution is considerably changed. However, on account of the boundary layer being turbulent in this case the size of the wake is reduced and the pressure in the wake is slightly positive.

It may thus be concluded that the pressure distribution differs little from that of irrotational flow in the zone of accelerating flow, but in the zone of deceleration where separation necessarily occurs the pressure distribution may deviate considerably from that of irrotational flow.

Figure 18.6 shows the complete variation of C_D for a sphere with Reynolds number ranging from 0.2 to 10^6. In addition the variation of C_D with Reynolds number for a circular disc is also shown in Fig. 18.6. It may be noted that for the circular disc (which is a body of maximum 'bluffness') there is no reduction in C_D, because, except at low Reynolds numbers, C_D is independent of viscous effects, and the position of separation points is fixed at the sharp edges of the disc.

18.5 DRAG ON A CYLINDER

Consider an infinitely long cylinder of radius R, lying with its axis perpendicular to the direction of flow in a uniform stream of fluid of infinite extent having velocity of flow V. Again for this case also if it is assumed that the fluid flowing past the cylinder is ideal i.e., non-viscous, then the flow pattern will be symmetrical as shown in Fig. 18.7 (a), which may be represented by velocity potential ϕ and stream function ψ given by

$$\phi = V\left(r + \frac{R^2}{r}\right)\cos\theta$$

and

$$\psi = V\left(r - \frac{R^2}{r}\right)\sin\theta$$

in which r is the radial distance of any point in the flow field from the centre of the cylinder and θ is the angular distance of the point from the front or the rear stagnation point. The derivation of the above noted expressions for the velocity potential and the stream function may however be seen in any standard book on classical hydrodynamics. The velocity components, V_r and V_θ at any point in the flow field may be obtained from either the velocity potential or the stream function as

$$V_r = -V\left(1 - \frac{R^2}{r^2}\right)\cos\theta$$

$$V_\theta = V\left(1 + \frac{R^2}{r^2}\right)\sin\theta$$

The values of the velocity components at any point on the surface of the cylinder may be obtained by introducing $r = R$ in the above noted expression as

$$V_r = 0; \text{ and } V_\theta = 2V\sin\theta$$

It thus follows that the resultant velocity v at any point on the surface of the cylinder is along the tangent to the cylinder and it is given by

$$v = 2V\sin\theta \qquad \ldots(18.22)$$

Applying Bernoulli's equation between any point in the unaffected flow and any point on the surface of the cylinder, the pressure p at any point on the cylinder is obtained as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

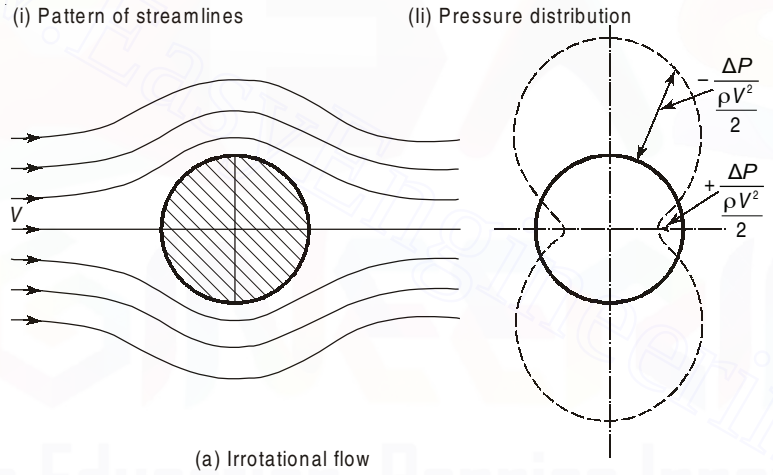
$$p = p_0 + \frac{1}{2}\rho V^2 - \frac{1}{2}\rho v^2$$

in which p_0 is the pressure in the uniform flow at some distance ahead of the cylinder. By substituting the value of v from Eq. 18.22, we have

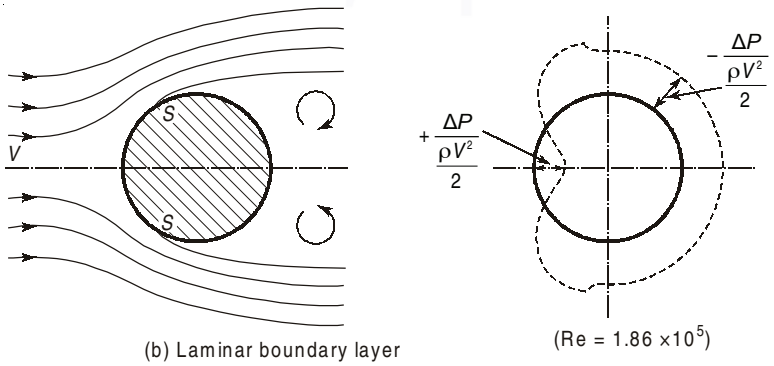
$$p = p_0 + \frac{1}{2}\rho V^2 - 2\rho V^2 \sin^2\theta$$

This expression is independent of the sign of sin θ and hence the pressure distribution around the cylinder is symmetrical about the mid-section as shown in Fig. 18.7 (a). Consequently the drag on the cylinder is zero.

However, due to the viscosity possessed by the real fluid flowing past a cylinder the actual pressure distribution around the cylinder is considerably modified. Some of the typical pressure distribution diagrams around a cylinder for different values of Reynolds numbers (Re $=VD/\upsilon$; where D is the diameter of the cylinder) are shown in Fig. 18.7 (b) and 18.7 (c). It is indicated from these diagrams that,



(a) Irrotational flow



(b) Laminar boundary layer (Re = 1.86 ×10^5)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

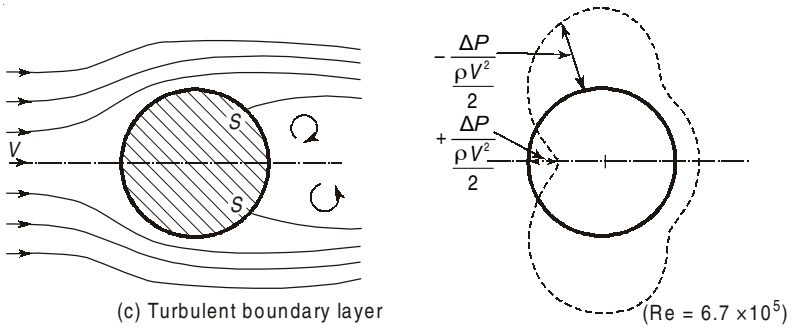


Figure 18.7 *Pressure distribution around a cylinder*

as in the case of a sphere, as long as the boundary layer is laminar the points of separation are located on the upstream half portion of the cylinder, but when the boundary layer becomes turbulent, the points of separation shift farther downstream towards the rear of the cylinder. Furthermore, the pressure distribution diagrams around a cylinder as shown in Fig. 18.7 are similar to those around a sphere as shown in Fig. 18.5. But as indicated later the flow pattern behind a cylinder is altogether different from that behind a sphere.

For the case of flow past a cylinder at very low values of Reynolds number (say Re < 0.5), Lamb has obtained an analytical solution for the drag experienced by the cylinder, on the basis of the consideration that at such low values of Re inertia forces are negligible as compared with the viscous forces. The drag is then almost entirely proportional to the velocity V. Therefore, as in the case of a sphere, the drag coefficient C_D of a cylinder is also approximately inversely proportional to Re. However, the analytical solution so obtained agrees closely with experiments only for *Re* less than 0.2 and the experimental points start deviating form the theoretical curve for Re greater than about 0.2. Evidently at such low values of R*e* skin friction drag accounts for a large part of the total drag.

With the increase in the Reynolds number, the inertia forces become more important and therefore these cannot be neglected in the analysis. Then the analytical solution of the problem is not possible and one has to resort to experimental approach only.

As the Reynolds number increases the flow pattern with respect to an axis perpendicular to the direction of flow becomes unsymmetrical. It is on account of the fact that in the wake developed just behind the cylinder, a more or less orderly series of vortices (which alternate in position about the centre line) are developed. At Re ranging from about 2 to 30, very weak vortices are formed on the downstream of the cylinder, as shown in Fig. 18.8 (a). It is, therefore, the initial stage for the development of the wake. In the wake region there exists a flow in the opposite direction along the axis of the wake, but in the outer portion the flow is in the general direction of motion. At Re ranging from about 40 to 70 the wake as well as the pair of vortices become quite distinct as shown in Fig. 18.8 (b). With a further increase in the value of Re the vortices become more and more elongated in the direction of flow, and at Re equal to about 90 these vortices become symmetrical, they leave the cylinder and slowly move in the downstream direction.

Karman Vortex Trail. Experiments have shown that when the Reynolds number exceeds about 30, the two vortices formed at the points of separation elongate to such an extent that they become unstable and are washed down. This process of the formation of the vortices and their shedding away from the cylinder continues. As such the wake then consists of a series of vortex pairs moving in the downstream

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

direction with very small velocity. Since Von Karman was first to study and analyse the stability of these regular vortex trails behind a cylinder, these trails of vortices are commonly known as "*Karman vortex trails*" (or *Karman vortex street*). The two possible vortex configurations in the Karman vortex

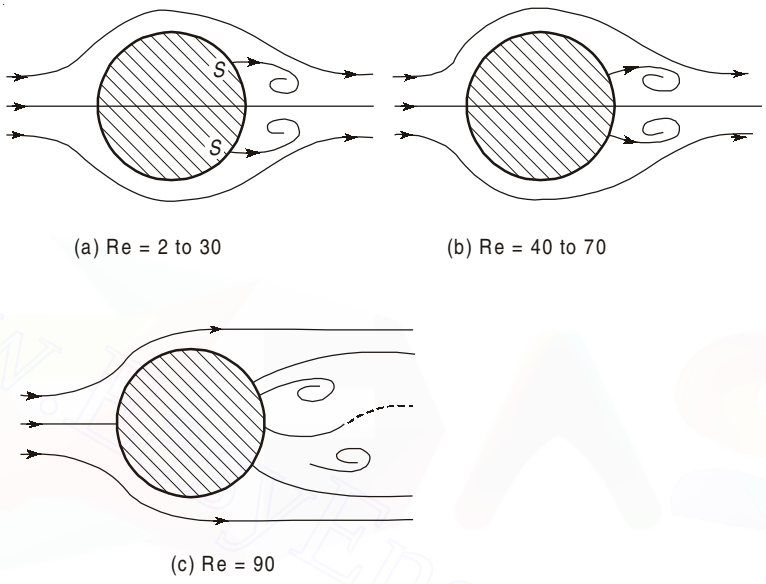


Figure 18.8 *Flow past an infinitely long cylinder*

trails as suggested by Karman are as shown in Fig. 18.9. However, a theoretical analysis of the stability of vortices of these two configurations revealed that the symmetrical configuration of vortex pair shown in Fig. 18.9 (a) is not at all stable; and the staggered configuration shown in Fig. 18.9 (b) is also

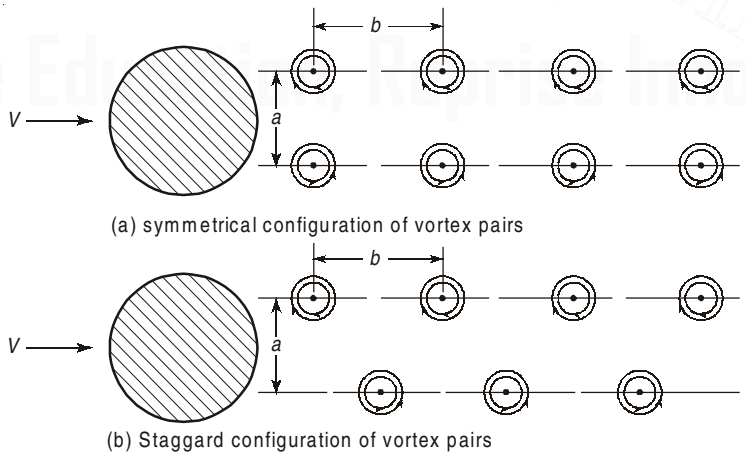


Figure 18.9 *Possible vortex configuration in Karman vortex trail*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

unstable except when the relative spacing $\left(\frac{a}{b}\right)$ of the vortices have the value $\frac{1}{\pi}\sinh^{-1} 1 = 0.281$, i.e., $\frac{a}{b} = \frac{1}{\pi}\sinh^{-1} 1 = 0.281$. Further these vortices move with respect to the fluid at a velocity $V_v = \frac{3.355\Gamma}{b}$, in which Γ is circulation which is a measure of the vortex intensity. The validity of these expressions has been confirmed by several other investigators. Therefore it may be concluded that as the fluid flows past a cylinder, an alternate shedding of the vortices occurs, because this is the only stable type of pattern which may be developed for the Karman vortex trails.

Since the absolute velocity of the vortices is equal to $(V - V_v)$, the parameter $\left(\frac{V - V_v}{b}\right)$ then represents the frequency f with which the vortices are shed alternately from the cylinder on the downstream side. The frequency f with which vortices are shed from a cylinder is given by the empirical formula

$$f = 0.198\frac{V}{D}\left[1 - \frac{19.7}{\text{Re}}\right]$$

or
$$\frac{fD}{V} = 0.198\left[1 - \frac{19.7}{\text{Re}}\right] \quad ...(18.23)$$

The parameter (fD/V) is dimensionless and it is called the *Strouhal number* (after V. Strouhal, a Czech physicist), which as indicated by Eq. 18.23 is a function of Reynolds number. Although the alternate shedding of the vortices begins at Re equal to about 45, but these can be seen only at Re equal to about 120 and it continues upto Re equal to about 10^5. In this range of Reynolds number, equation 18.23 shows that for cylinders Strouhal number varies from about 0.17 to 0.20.

In fact such vortices are always shed when any two dimensional bluff body is held in a stream and the flow separates. Thus same is the case with long flat plates held normal to the direction of flow of stream of fluid for which however the Strouhal number is equal to about 0.14 for Re greater than 10^3.

The alternate shedding of these vortices produces periodic transverse forces on the cylinder which tend to cause transverse oscillations. If the natural frequency of vibration of the cylinder is close to the frequency of the vortex shedding, severe deflection and damage can result due to resonance. These considerations are vital to the design of elastic structures such an suspension bridges, tall chimneys, etc., which are exposed to the wind. Similarly the 'singing' of telephone or transmission line wire in high winds is caused when the frequency of vortex shedding is close to the natural frequency of the wires.

As stated earlier when the separation of the boundary layer occurs and the wake develops, then the pressure (or form) drag makes a larger contribution in the total drag. Thus in the case of cylinders also as the Reynolds number increases the contribution of the pressure drag in the total drag increases from about one third of the total at small Reynolds numbers to about half of the total drag as the Reynolds number increases and vortices begin to form. The contribution of the pressure drag in the total drag further increases to about three quarters of the total at Re equal to about 200, when the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Karman vortex street is well established. Fig. 18.10 shows the variation of C_D with Re for a cylinder, from which it is seen that at high values of Re the variation of C_D with Re for a cylinder follows a pattern similar to that for a sphere. The drag coefficient for a cylinder reaches a minimum value of about 0.95 at Re = 2000 and then there is slight rise to 1.2 for Re = 3×10^4. The reason for this rise in the

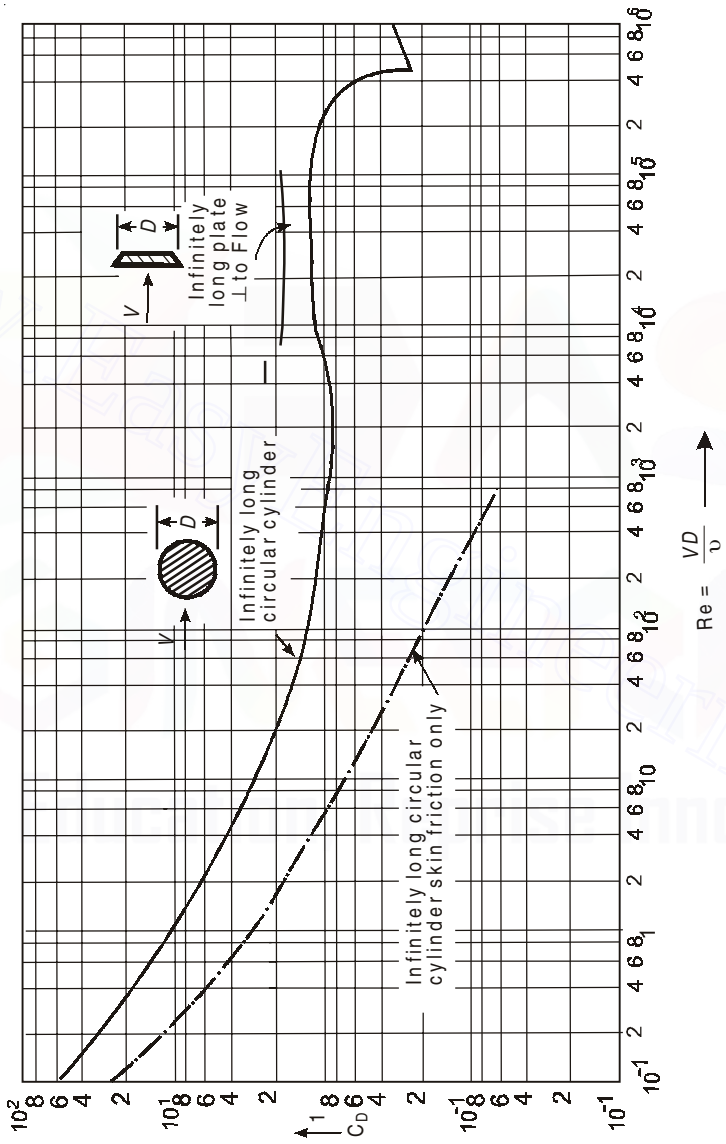


Figure 18.10 *Variation of C_D with Re for a cylinder and flat plate*

value of C_D is the increasing turbulence in the wake and also the widening of the wake due to the separation points gradually advancing upstream.

Downloaded From : www.EasyEngineering.net

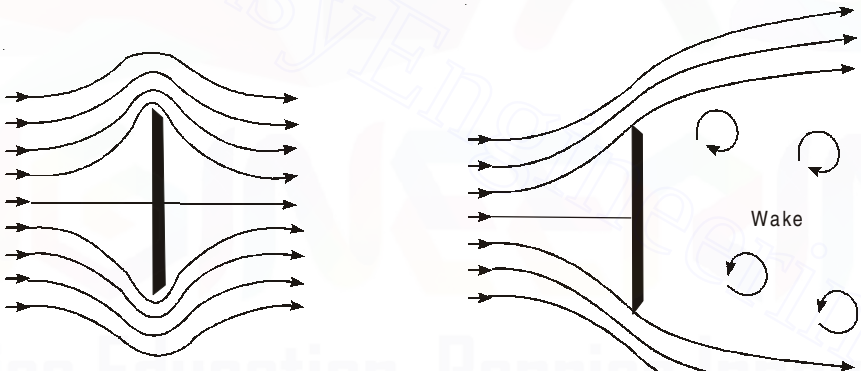
Downloaded From : www.EasyEngineering.net

At Re = 2×10^5 the boundary layer which was up to now laminar, becomes turbulent before separation and therefore there is a drop in the value of C_D from 1.20 to about 0.3. However, with a further increase in Re the value of C_D increases gradually from 0.3 to about 0.7 over the approximate range of $5 \times 10^5 < \text{Re} < 3 \times 10^6$. Thereafter since viscous effects are relatively small, it is probable that C_D is practically independent of Re.

18.6 DRAG ON A FLAT PLATE

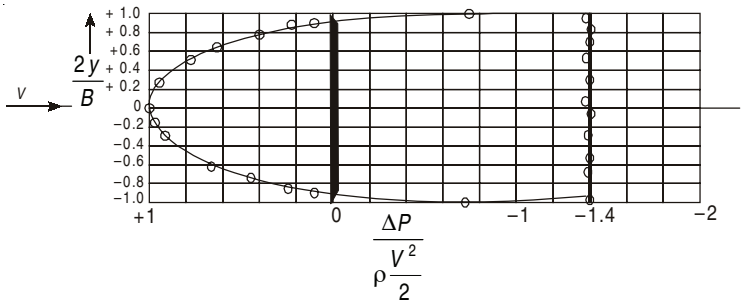
(a) Plate Held Parallel to the Direction of Flow of Fluid. When a thin flat plate is held parallel to the direction of flow of fluid (i.e., the plate is held at zero incidence) then as stated earlier the total drag exerted by the fluid on the plate is equal to the friction drag which is due to the formation of the boundary layer on the surface of the plate. The magnitude of the friction drag exerted on the plate however depends on whether the boundary layer formed over the surface of the plate is laminar or turbulent or partly laminar and partly turbulent. For computing the magnitude of the drag as well as drag coefficients for all these three cases, expressions have been developed in Chapter 12, which may be used.

(b) Plate Held Perpendicular to the Direction of Flow of Fluid. Consider an infinitely long plate held perpendicular to the direction of flow of fluid in a stream of infinite extent. For an ideal fluid flowing past the plate the flow pattern will be symmetrical on the upstream and the downstream sides of the plate as shown in Figure 18.11 (a). Further, because of symmetrical flow pattern the pressure



(a) Irrotational flow

(b) Real fluid flow



(c) Pressure Distribution on Upstream and downstream Faces of an Infinite Flat Plate.

Figure 18.11 *Flow past a flat plate held perpendicular to the flow*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

distribution will also be symmetrical and hence there will be no drag. However, for a real fluid flowing past a flat plate the flow pattern as well as the pressure distribution are markedly different from those for an ideal fluid. As shown in Fig. 18.11 (b) the flow separates at the top and the bottom edges of the plate, thereby forming a fairly wide wake on the downstream side. Figure 18.11 (c) shows the pressure distribution on the upstream and the downstream faces of the plate as obtained by actual measurement. As expected the pressure on the upstream face of the plate is positive for the entire width, with a maximum at the middle section along which the stagnation points are located. On the downstream face of the plate in the wake zone a constant pressure drop takes place as indicated by a more or less constant negative pressure over the entire width of the plate.

As stated in Sec. 18.1, for a flat plate held perpendicular to the flow the skin friction drag is negligible as compared with the pressure drag, and so the effects of inertia forces become predominant even at much lower values of Re than for a cylinder. Moreover, unlike a sphere or a cylinder, in the case of a flat plate, except at low Reynolds numbers (say Re < 100), separation of the boundary layer always occurs at the same place–the sharp edges of the plate–whatever be the value of the Reynolds number. As such in the case of a flat plate the drag coefficient C_D is a function of Re only at low and moderate values of *Re*. However, as the value of Re exceeds 10^3, C_D assumes a constant value of about 2.0, as shown in Fig. 18.10. A reduction in the value of C_D however occurs if the ratio of the length *L* of the plate to its width *B* is not very large. The reduction in the value of C_D in this case is on account of the fact that the flow in not truly two-dimensional in character. The flow of fluid round the ends tends to reduce the pressure at the upstream face of the plate and increase at the rear face. The value of C_D therefore decreases as the length of the plate is reduced and these end effects become more significant. This is demonstrated by the values of C_D , obtained for a thin flat plate held perpendicular to the flow, for different values of the ratio of length to breadth (*L/B*) of the plate which are tabulated below:

L / B	1	2	4	10	18	∞
CD	1.10	1.15	1.19	1.29	1.40	2.01

The coefficient of drag, although remains independent of Reynolds number for *Re* > 1000, it varies markedly with the (*L/B*) ratio of the plate.

The variation of C_D with Re, for an infinitely long flat plate follows the trend similar to that for a circular disc, both held perpendicular to the flow. However, because of the flow in the case of circular disc being three-dimensional in character the limiting value of C_D is only about 1.1, which is much smaller than the limiting value of C_D equal to about 2, for the flat plate having flow which is two-dimensional in character.

18.7 DRAG ON AN AIRFOIL

An airfoil is a streamlined body as shown in Fig. 18.12. In such a case, because of the streamlining of the body, the separation occurs only at the extreme rear of the body, resulting in a small wake and consequently small pressure drag. Therefore in the case of an airfoil even at relatively high values of Reynolds number, it is the skin friction drag which makes the major contribution to the total drag, and the contribution of the pressure drag is comparatively very small. Figure 18.12 shows the pressure distribution around an airfoil, based upon the actual measurement and the theoretical irrotational flow analysis. It can be seen that the two pressure distributions agree very well except at the rear end, where the separation may occur.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The coefficient of drag C_D in the case of an airfoil will necessarily depend on Re and its shape. Further in the case of an airfoil, a sudden drop in the value of C_D, as in the case of a sphere or a cylinder when the boundary layer changes from laminar to turbulent, cannot be expected. This is because in this case the change of boundary layer from laminar to turbulent does not alter the wake size significantly to vary the contribution of the pressure drag.

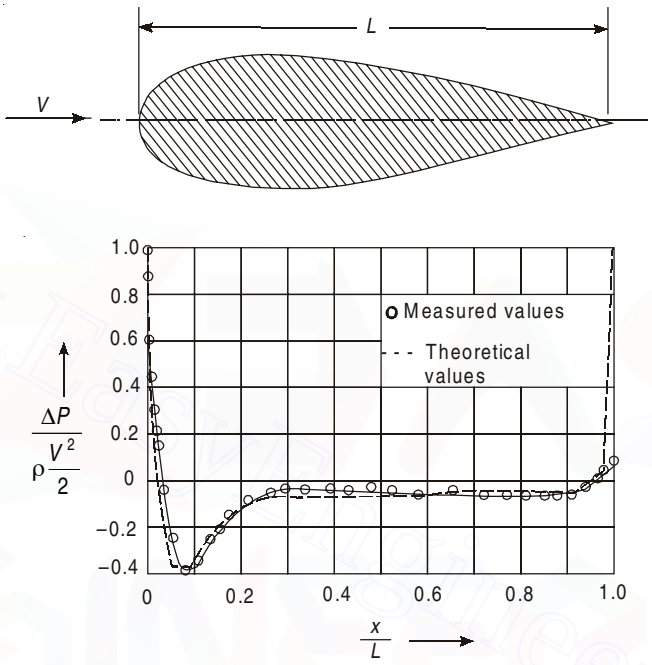


Figure 18.12 *Pressure distribution around an airfoil*

18.8 EFFECT OF FREE SURFACE ON DRAG

If the effects due to the compressibility of the fluid flowing past any object can only be neglected and the viscous as well as gravity forces are predominant, then the general Eq. 18.13 for the coefficient of drag will take the form

$$C_D = \phi(\eta, \text{Re}, Fr) \qquad ...(18.24)$$

The gravity forces become predominant when the object is lying at the interface (or common surface) between the two fluids of different densities. In such cases for the geometrically similar bodies the drag coefficient will depend on both Re and *Fr*, as indicated by Eq. 18.24. Such a condition may arise, for instance, in the case of ships moving on the water surface. As indicated in Chapter 17, the total drag in the case of a ship is the sum of skin friction drag, pressure (or form) drag, and the drag due to surface waves. The skin friction drag is essentially due to the viscosity of the fluid, and it can be computed separately by using the method indicated in Sec. 17.16. The difference between the total drag and the skin friction drag is commonly known as *residual drag*. The residual drag is thus made up of pressure (or form) drag and the drag due to surface waves. Since the ships are generally streamlined, the pressure (or form) drag is relatively small and within the usual range of Reynolds number it can be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

assumed to remain constant. Moreover, it is usual practice to consider this small pressure drag to be included in the drag due to surface waves. Since the formation of surface waves is associated with gravitational action, the residual drag is essentially a function of Froude number. The residual drag may, however, be determined by model studies as indicated in Sec. 17.16.

18.9 EFFECT OF COMPRESSIBILITY ON DRAG

If the gravity forces are not predominant due to the non-existence of the free surface or the interface, but the elastic forces (or compressibility effects) are significant then Eq. 18.13 for drag coefficient will take the form

$$C_D = \phi(\eta, \text{Re}, Ma) \qquad ...(18.25)$$

The elastic forces become significant when the velocity of flow approaches the velocity of sound in that fluid, and the drag coefficient becomes a function of Mach number *Ma* also as indicated above. If the Mach number is less than one, the flow is called subsonic, while if the Mach number is more than one, the flow is called supersonic and for the Mach number equal to one the flow is known as sonic. As long as the flow is subsonic with Mach number less than about 0.4, the effects of compressibility are found to be negligible. But with the increase in the value of Mach number the compressibility of fluid has significant influence on the flow characteristics as well as the drag.

In general the drag on a submerged object is caused by both shear forces and pressure differences on the object. But in compressible flow the shear and pressure distributions round the immersed object are modified appreciably from that in an incompressible flow. Such modifications are on account of the fact that in a compressible flow additional forces are transmitted through the fluid by the shock waves produced. The shock waves are essentially the elastic waves (or pressure waves) spherical in form, which travel at the velocity of sound, and these are produced in the vicinity of the object immersed in a compressible fluid flowing past the object (see Sec. 19.8). In the case of a subsonic flow since the velocity of flow is less than the velocity of sound, the effects of the disturbances developed in the flow in the form of shock waves, due to the presence of the object, are already spread through the flow field. As such the fluid adjusts itself accordingly before it encounters the object, and the compressibility effects are thus rendered insignificant.

On the other hand in the case of a supersonic flow the conditions are altogether different. This is so because the velocity of flow being greater than the velocity of sound, the fluid approaches the object all of a sudden without any previous adjustment, and it is only when the fluid encounters the object, that the fluid suddenly changes its direction, producing the shock waves. Such shock waves with conical wave front are observed to extend backwards from the leading edge of the objects in the case of supersonic flows past immersed object as shown in Fig. 18.13. An abrupt change of pressure always occurs across such a shock wave which produces the drag. In addition the drag also results from the energy dissipated in the shock waves as well as the skin friction and the separation effects discussed earlier.

Equation 18.25 shows that in the case of high velocity flows the drag coefficient depends on the shape of the object, the Reynolds number and the Mach number. However, when the velocity is high enough for compressibility effects to be important, the corresponding Reynolds number is usually high enough for viscous effects to be significant. In other words, at high values of Mach number, the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

drag is practically independent of Reynolds number whereas at low values of Mach number, Reynolds number is the significant parameter. It may however be stated that there is no velocity at which the

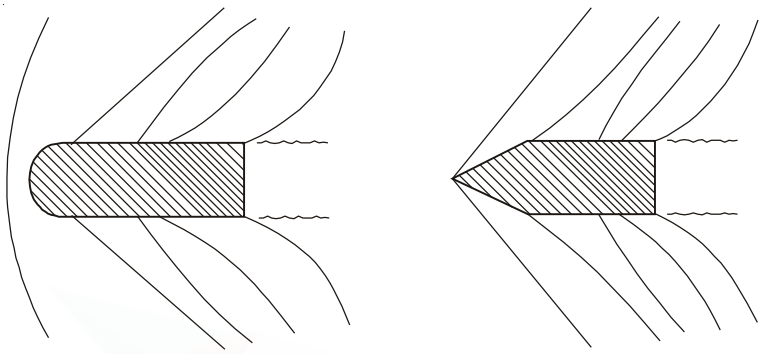


Figure 18.13 *Shock wave patterns around objects placed in supersonic flow field*

effects of compressibility may be considered to begin or those of viscosity cease, and in some situations Reynolds number and Mach number may even be of comparable significance.

Experimental data from drag tests in high velocity flows reveal that the drag coefficient of a given body rises sharply as the Mach number *Ma* of the oncoming flow approaches 1.0. As the Mach number

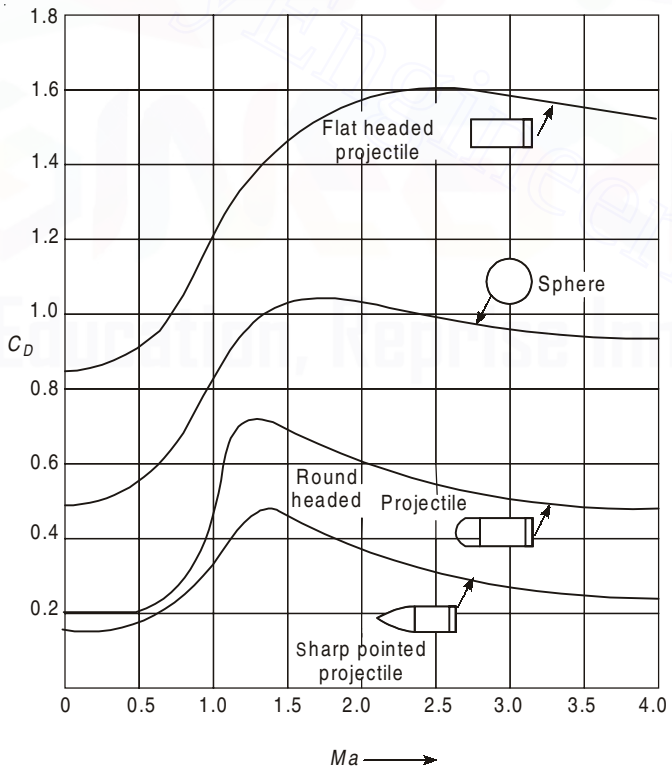


Figure 18.14 *Variation of C_D with Mach number for different bodies*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

increases further, the drag coefficient falls gradually and tends to approach a constant value asymptotically. The variation of drag coefficient with Mach number for the bodies (or projectiles) of different shapes is shown in Fig. 18.14. It may be noted from these curves that the numerical values of C_D drop steadily when the nose of the body becomes successively more pointed, the rear of the body remaining unchanged in all cases. This is so because in supersonic flow a sharp pointed nose creates a narrow shock wave front which tends to minimize the drag. Moreover, in such flows the contribution of the wake resistance to the total drag is relatively insignificant in comparison with the shock front, because the absolute pressure in the wake cannot fall below zero, and therefore the form drag has a limited value which more or less remains constant irrespective of the shape of the tail end of the object.

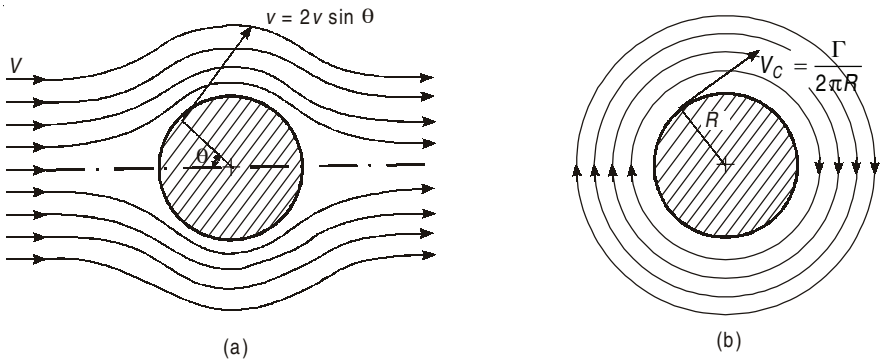
From the above discussion it is observed that for minimum drag in supersonic flow the body should have a sharp forward edge, or conical nose, and the shape of the rear end is of secondary importance. This requirement is, however, the reverse of that for subsonic flows, for which as stated earlier the drag is the least for a streamlined body well tapered at the rear and rounded at the front. Thus a body well streamlined for subsonic flows may be poorly shaped for supersonic flows and vice versa.

18.10 DEVELOPMENT OF LIFT ON IMMERSED BODIES

As stated in Sec. 18.1 the force exerted by the fluid on the body may in general be inclined to the direction of motion, and hence it can be resolved into two components—one along the direction of motion and the other perpendicular to the direction of motion. The component perpendicular to the direction of motion is called the *lift*. When the body is symmetrical with respect to its axis and so located that its axis is parallel to the direction of motion, then the resultant force exerted by the fluid on the body is in the direction of motion, and in such a case the lift is zero. However, if the axis of symmetry of the body makes an angle with the direction of motion, the resultant force acting on the body will have a lift component. In order to study the development of lift some of the specific cases are discussed below.

(a) Lift on a Circular Cylinder. Consider an ideal fluid flowing past a cylinder of radius R, with a uniform velocity V. The flow pattern in this case will be symmetrical about both axes of the cylinder as shown in Fig. 18.15 (a), which may be represented by velocity potential ϕ and stream function ψ as indicated in Sec. 18.5. Further as indicated in Sec. 18.5 the resultant velocity v at any point on the surface of the cylinder is along the tangent to the cylinder and it is given by Eq. 18.22 as

$$v = 2V \sin \theta$$



(a) (b)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

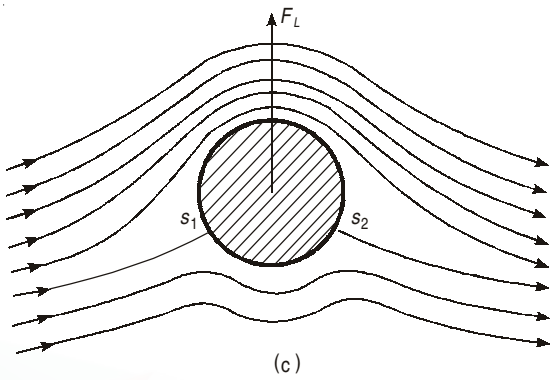


Figure 18.15 *Development of lift on a cylinder due to circulation*

In this case since the flow pattern is symmetrical about the horizontal axis, the pressure distributions on the upper and lower halves of the cylinder are identical as shown in Fig. 18.7 (a), and hence there is no lift acting on the cylinder.

Next consider a constant circulation Γ imparted to the same cylinder, due to which the flow pattern around the cylinder will consist of a series of concentric circular streamlines as shown in Fig. 18.15 (b). From the mathematical definition of circulation given in Chapter 6, the peripheral velocity on the surface of the cylinder is given by

$$v_c = \frac{\Gamma}{2\pi R} \qquad ...(18.26)$$

If these two flow patterns are superimposed one over the other, then a composite flow pattern as shown in Fig 18.15 (c) will be obtained. It is clearly seen that the resulting composite flow pattern is unsymmetrical about the horizontal axis. The velocity v of the composite flow on the surface of the cylinder is the vector sum of the two components, and hence Eqs 18.25 and 18.26 can be combined to give

$$v = 2V \sin\theta + \frac{\Gamma}{2\pi R} \qquad ...(18.27)$$

Since the circulation Γ has been taken as clockwise, the superposition causes the velocity around the upper half portion of the cylinder to be higher than that around the lower half portion. This is so because around the upper half portion of the cylinder the velocity of flow and the velocity due to circulation being in the same direction are added together, while around the lower half portion of the cylinder both these velocities being in opposite directions are subtracted. Application of Bernoulli's equation then shows that the pressure around the lower portion of the cylinder is greater than the pressure around the upper portion. As such there will be a force acting on the cylinder in the direction perpendicular to the direction of motion. In other words, a lift is exerted on a cylinder lying in a uniform flow when a circulation is superimposed on the uniform flow field. Further since the composite flow pattern as well as the resulting pressure distribution is symmetrical about the vertical axis, there is no drag on the cylinder in this case.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The position of the stagnation points S_1 and S_2 on the surface of the cylinder, as shown in Fig. 18.15 (c), may be determined by considering $v = 0$ in Eq. 18.27 and solving for sin θ which gives

$$\sin\theta = -\frac{\Gamma}{4\pi RV}$$

The negative sign in the above expression for sin θ indicates that angle θ is equal to – θ or (180 + θ), since in both these cases sin θ is negative. Further as long as sin θ > –1, then there will be two stagnation points on the lower surface of the cylinder as shown in Fig. 18.15 (c). When sin θ = –1, both the stagnation points will coincide and will lie at the bottom-most portion of the cylinder (i.e., at the lower extremity of the vertical diameter). This will therefore be the limiting condition for the location of the stagnation points on the surface of the cylinder. By substituting for Γ from Eq. 18.26 the above expression for this case becomes

$$\sin\theta = -1 = -\frac{v_c}{2V}; \text{ or } \frac{v_c}{V} = 2$$

For sin θ < –1 it is evident that the stagnation points will not lie on the surface of the cylinder but they will move out into the fluid.

The magnitude of the lift exerted on the cylinder due to the composite flow pattern may be determined by integrating over the entire surface of the cylinder, the components of the pressure forces on elementary surface areas normal to the direction of uniform flow. Applying Bernoulli's equation between any point in the unaffected flow and any point on the surface of the cylinder, the pressure at any point on the cylinder is obtained as

$$p = p_0 + \frac{1}{2}\rho V^2 - \frac{1}{2}\rho v^2$$

in which p_0 is the pressure in the uniform flow at some distance ahead of the cylinder. By substituting the value of v from Eq. 18.27, we have

$$p = p_0 + \frac{1}{2}\rho\left[V^2 - \left(2V\sin\theta + \frac{\Gamma}{2\pi R}\right)^2\right]$$

The lift dF_L acting on an elementary surface area of the cylinder ($LRd\theta$) as shown in Fig. 18.16 is given by

$$dF_L = -(LRd\theta)\, p \sin\theta$$

in which L is the length of the cylinder. The negative sign has been introduced because the pressure force is always directed towards the surface, and hence for sin θ being positive its component is negative being in the vertical downward direction. The total lift F_L exerted on the cylinder is obtained by integration as

$$F_L = -\int_0^{2\pi}(LRd\theta)\, p \sin\theta$$

$$= -LR\int_0^{2\pi}\left[p_0 + \frac{1}{2}\rho\left\{V^2 - \left(2V\sin\theta + \frac{\Gamma}{2\pi R}\right)^2\right\}\right]\sin\theta\, d\theta$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which reduces to a simple relationship

$$F_L = \rho V L \Gamma \qquad ..(18.28)$$

Equation 18.28 is commonly known as *Kutta-Joukowski equation* in honour of a German scientist M.W. Kutta and a Russian scientist N.E. Joukowski who independently derived this equation at almost the same time. Although Kutta-Joukowski equation was developed for a cylinder of circular cross-section, it has been found to hold good for a cylinder of any cross-section having a circulation around it.

In fact this phenomenon of the lift produced by circulation around a circular cross-section placed in a uniform stream of fluid, was first investigated experimentally by a German physicist H.G. Magnus in 1852. As such it is commonly known as *Magnus effect*.

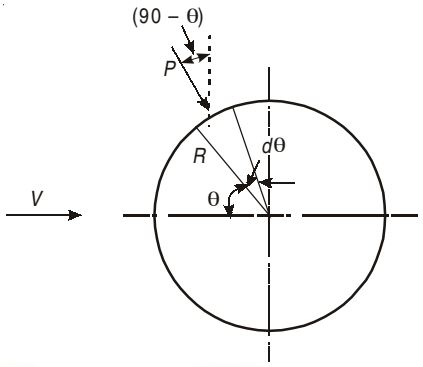


Figure 18.16 *Development of lift on a cylinder due to circulation*

The lift coefficient C_L as defined by Eq. 18.6 may be expressed as

$$C_L = \frac{F_L / A}{\frac{1}{2}\rho V^2} = \frac{(\rho L V \Gamma)/(2RL)}{\frac{1}{2}\rho V^2} = \frac{\Gamma}{RV} \qquad ...(18.29)$$

where A is the projected area which is equal to $2RL$. Introducing Eq. 18.26 in Eq. 18.29 lift coefficient may also be expressed as

$$C_L = \frac{2\pi v_c}{V} \qquad ...(18.30)$$

From Eqs 18.29 and 18.30, it may be seen that the lift coefficient C_L varies directly with the circulation Γ and the velocity ratio (v_c/V) for irrotational flow. The velocity ratio (v_c/V) also affects the location of the stagnation points at the lower portion of the cylinder as shown in Fig. 18.15 (c) and as indicated earlier when $(v_c/V) = 2$ the two stagnation points meet at the bottom of the cylinder which is then the limiting condition for this case. From Eq. 18.30 for $(v_c/V) = 2$ the lift coefficient $C_L = 4\pi = 12.56$, which is thus theoretical maximum value of the lift coefficient C_L.

The above analysis regarding the development of lift for a rotating circular cylinder is for the flow of an ideal fluid having no viscosity, i.e., for the irrotational flow. However, the real fluids have viscosity which results in an altogether different flow pattern and as indicated below the actual value of the lift coefficient C_L obtained experimentally is very much different from the theoretical value given by Eq. 18.30.

Figure 18.17 shows the variation of drag and lift coefficients with the velocity ratio (v_c/V) for a rotating circular cylinder. The dotted line on the left is the plot of Eq. 18.30 representing the theoretical value of lift coefficient for a circular cylinder in an irrotational flow. The solid lines represent the actual values of the lift and drag coefficient obtained experimentally for a circular cylinder of length to diameter ratio $(L/D) > 10$, for which the end effects are negligible and the flow may be considered as two dimensional. From this plot it can be seen that to produce a given lift coefficient the actual velocity required is greater than twice that required theoretically. Furthermore, as the peripheral velocity v_c increases to about four times the velocity of flow of fluid V, the lift coefficient C_L approaches a maximum

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

value of about 9.0 as compared with the theoretical maximum value of about 12.6. In addition to this, Fig. 18.17 shows that the drag coefficient C_D also varies with velocity ratio (v_c/V) from about 1.0 for

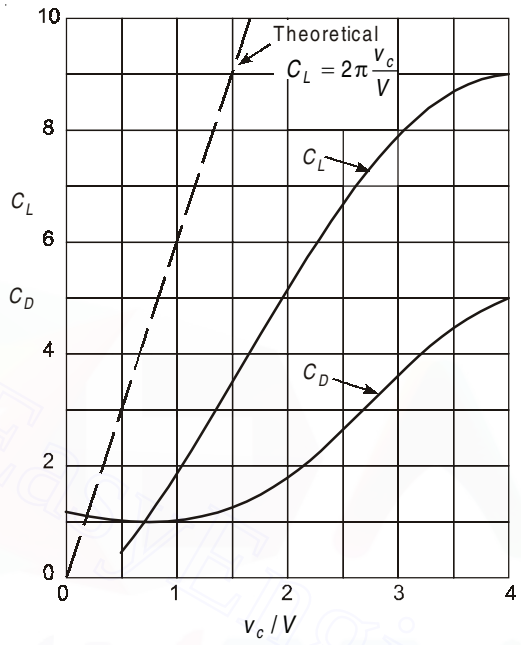


Figure 18.17 *Lift and drag coefficient for a rotating circular cylinder*

small velocity ratios to about 5 at a velocity ratio equal to 4. For (v_c/V) ≈1.0 the value of the drag coefficient C_D is minimum and afterwards it increases steeply with an increase in the value of (v_c/V).

If the cylinder is relatively short i.e., it has length to diameter ratio (L/D) < 10 then there is considerable effect of flow around its ends which appreciably reduces the lift coefficient. For example, when the ratio (L/D) = 5, the lift coefficient C_L is about half of that for a longer cylinder having ratio (L/D) >10. The flow around the ends of a short cylinder also causes an induced drag in addition to the normal drag. The induced drag for a short cylinder is caused in the same manner as in the case of an airfoil of finite length which has been discussed in Sec. 18.11.

Alike rotating cylinder a rotating sphere lying in a stream of fluid is also acted upon by lift. The phenomenon of development of lift for rotating sphere may be used in the games such as cricket, table tennis, golf and tennis in which by imparting a spin or rotation to the ball in the backward or forward direction the lift may be exerted on it in the upward or downward direction and hence the trajectory of the ball may be modified by the player to his best advantage.

(b) Lift on an Airfoil. An airfoil is a streamlined body which may be either symmetrical or unsymmetrical as shown in Fig. 18.18. It is generally characterised by the chord length c and the angle of attack α. The chord length is the length of the line joining the leading (i.e., front) edge and the trailing (i.e., rear) edge of an airfoil and this line is termed as its chord line. For a symmetrical airfoil the chord line coincides with its axis of symmetry. The angle of attack is the angle between the chord line and the direction of flow of fluid. The overall length of an airfoil (in the direction perpendicular to the cross-

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

section) is termed as its span. The ratio of span L and chord length c of an airfoil (i.e., L/c) is known as *aspect ratio.*

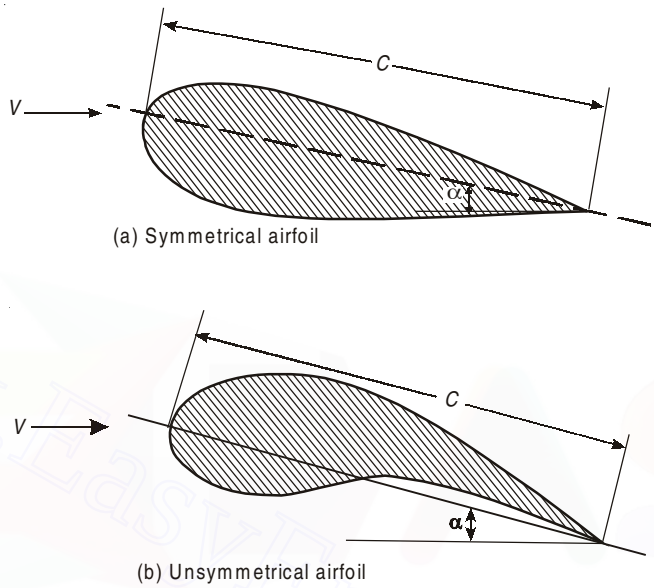


Figure 18.18 *Types of airfoil sections*

Consider a uniform flow of ideal fluid flowing past an airfoil of long span. The resulting flow pattern of irrotational flow is shown in Fig. 18.19 (a). Next consider a constant circulation imparted to the same airfoil and the resulting flow pattern is shown in Fig. 18.19 (b). Further if these two flows are combined then the resulting flow pattern is as shown in Fig. 18.19 (c). Joukowski showed that the pattern of flow round a circular cylinder could be used to deduce the flow pattern around a body of any other shape. This is however done by a mathematical process called 'conformal transformation.' As such it may be noted that Kutta-Joukowski equation, though developed for a cylinder of circular cross-section, is found to hold good for a cylinder having a cross-section of any shape provided there is circulation around it.

From the theoretical analysis it has been found that by properly adjusting the circulation it is possible to obtain the flow pattern such that the streamline at the trailing end of the airfoil is tangential to it as shown in Fig. 18.19 (c). The circulation Γ required to do this has been found analytically as

$$\Gamma = \pi c V \sin\alpha \qquad ...(18.31)$$

where c is chord length, V is uniform velocity of flow and α is the angle of attack. Introducing Eq. 18.31 in Kutta-Joukowski Eq. (Eq. 18.28) the lift F_L on the airfoil of span L becomes

$$F_L = \pi c L \rho V^2 \sin\alpha \qquad ...(18.32)$$

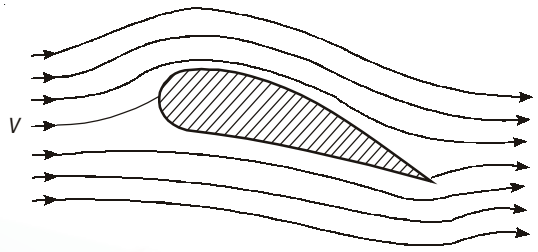
The lift coefficient C_L as defined by Eq. 18.6 may be expressed as

$$C_L = \frac{F_L / A}{\frac{1}{2}\rho V^2} = \frac{(\pi c L \rho V^2 \sin\alpha)/cL}{\frac{1}{2}\rho V^2} = 2\pi \sin\alpha \qquad ...(18.33)$$

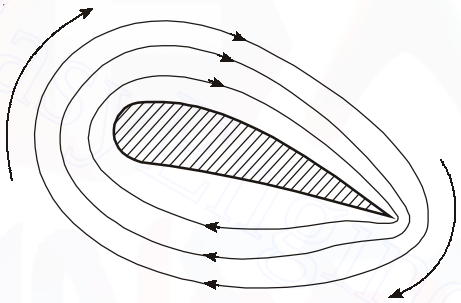
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

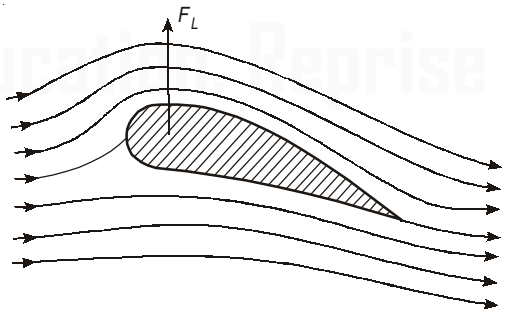
where A is the area of the projection of the airfoil on a plane perpendicular to its cross-section, which in the case of an airfoil of span L and chord length c is equal to (cL). It is seen from Eq. 18.33 that the angle of attack α has a considerable effect upon the lift on an airfoil. Therefore α is an important variable in determining the characteristic of flow past an airfoil.



(a) Irrotational flow translation



(b) Circulation



(c) Combined translation and circulation

Figure 18.19 *Development of lift on an air foil due to circulation*

From Kutta–Joukowski equation, it is thus observed that the lift produced on a cylinder of any cross-sectional shape depends on the combination of fluid velocity and a circulation developed around the cylinder. For rotational bodies such as circular cylinder or sphere the development of circulation

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

can be easily visualized. When a rotating cylinder is placed in a stream of real fluid, circulation is induced through the viscosity of the fluid by the rotational motion of the periphery of the cylinder i.e., it maintains its circulation through the viscous action. However, airfoils being non-rotational bodies, the development of circulation around these is by entirely different means. The process of producing circulation around an airfoil may be described briefly as follows.

Figure 18.20 shows the various stages in the development of circulation around an airfoil. When a uniform stream of real fluid flows past an airfoil, then initially the flow pattern is same as that for an ideal fluid flowing past an airfoil i.e., an irrotational flow pattern develops as shown in Fig. 18.20 (a). However, as explained below, this pattern of irrotational flow cannot persist too long. In this flow pattern the fluid flowing along the lower surface of the airfoil is required to turn around the sharp trailing edge to reach the rear stagnation point which is located on the upper surface of the airfoil slightly

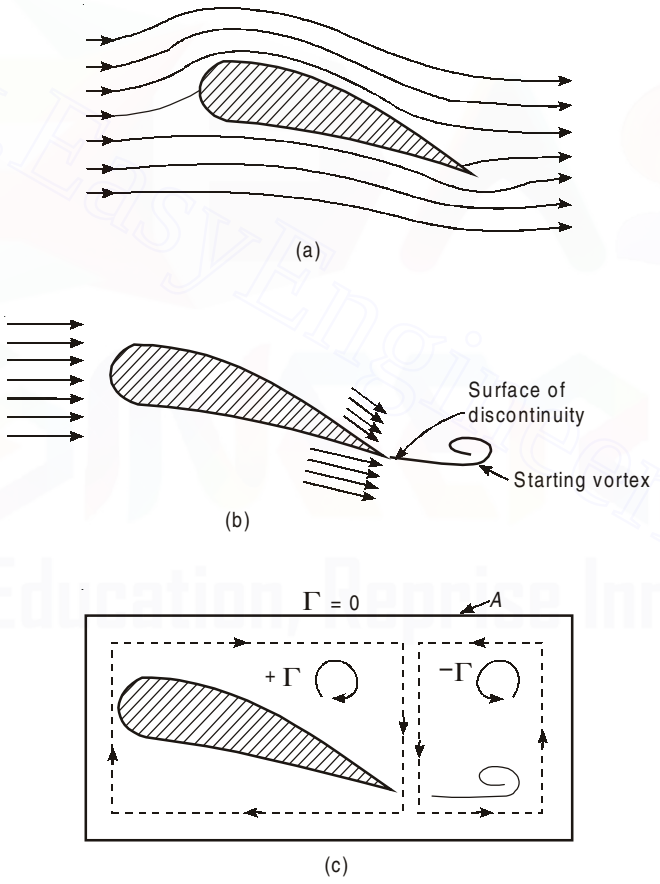


Figure 18.20 *Development of circulation around an airfoil*

upstream of its trailing edge. But shortly after the commencement of the flow due to the development of the boundary layer the retarded fluid of the lower surface boundary layer has no enough energy to take such a sharp turn at the trailing edge. Consequently the boundary layer separates from the lower surface at the trailing edge and on the upper surface a flow is induced from the stagnation point

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

towards the trailing edge. This flow is in the opposite direction to that of the ideal fluid which results in the formation of an eddy called *starting vortex* on the upper surface in the region of the trailing edge. The starting vortex has a definite strength of circulation associated with it, but it is quite unstable. As such the starting vortex is rapidly washed away from the trailing edge of the airfoil but, in leaving the airfoil, it generates an equal and opposite circulation round the airfoil. In this way, the net circulation round the curve A shown by firm line in Fig. 18.20 (c), which surrounds the airfoil as well as the starting vortex, remains zero. This must be so in order to satisfy Thomson's theorem, which states that in a frictionless fluid the circulation, around a closed curve which moves with the fluid so as always to touch the same particles, does not change with time. In other words, it follows from the Thomson's theorem that if around any curve circulation is zero before motion commences, it continues to be zero after the motion has started, even though finite circulations may be present within the curve. It may however be stated that although the Thomson's theorem has been developed for the ideal or frictionless fluid, it can be applied to the flow of real fluids as well. This is possible because the viscosity of a real fluid is significant only in a relatively small region of boundary where the velocity gradients are appreciable and in the region farther away from the boundary the fluid may for all purposes be treated as ideal or frictionless. Since the curve A is considered to be located far away from the airfoil, it lies in a region of more or less ideal flow, and hence the Thomson's theorem can be applied for this case too. Now in order to counterbalance the counter clockwise circulations of the starting vortex, a clockwise circulation of the same strength must be set up around the airfoil, so that the sum of the circulation around the curve A is zero in accordance with Thomson's theorem. The clockwise circulation around the airfoil is usually known as *boundary circulation.*

The circulation produced around the airfoil as a reaction to the starting vortex, brings the rear stagnation point nearer to the trailing edge. With more and more starting vortices produced, the circulation around the airfoil increases till the rear stagnation point reaches the stable position at the trailing edge. Corresponding to this position of the rear stagnation point, stable flow pattern is developed in which the upper and lower boundary layer merge smoothly at the trailing edge. However, in the flow pattern so developed the layers of fluid that pass over the upper and the lower surfaces of the airfoil possess markedly different velocities. When these layers possessing different velocities meet at the trailing edge of the airfoil, it results in the formation of a so called surface of discontinuity across which there exists a sharp velocity gradient. The surface of discontinuity so formed being quite unstable, gives rise to starting vortices which are continuously formed and washed away from the trailing edge of the airfoil and in turn develop a constant circulation around the airfoil which results in setting up of a constant lift on the airfoil. The existence of the starting vortices has been confirmed by the experimental observations. It may however be pointed out that though the starting vortices are instrumental in inducing the circulation around an airfoil, once they leave the airfoil they have no further effect on the flow round the airfoil and the vortices are ultimately dissipated by viscous action as they move downstream with the general flow of fluid. Furthermore, as indicated below, due to the viscosity, for a real fluid the actual lift exerted on an airfoil is somewhat less than that obtained by Kutta—Joukowski equation.

The drag and lift of an airfoil may also be expressed by Eqs 18.5 and 18.6 in which the area A is represented by the product of the length or span L and the chord length c of the airfoil. Thus for an airfoil Eqs 18.5 and 18.6 become

$$F_D = C_D Lc \frac{\rho V^2}{2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$F_L = C_L Lc \frac{\rho V^2}{2}$$

The coefficients C_D and C_L for an airfoil of any shape may be determined experimentally by using the above equations.

It is evident that the angle of attack α has considerable effect on the lift exerted on an airfoil and therefore angle α is an important variable in determining the lift and drag characteristics of an airfoil. However, for the sake of convenience in place of angle α, another angle α' is generally considered which is the inclination of the tangent to the lower boundary of the airfoil profile as shown in the inset sketch of Fig. 18.21, from which it may be noted that $\alpha = \alpha'$ plus a constant angle depending on the shape of the airfoil. The variation of the drag and lift coefficients with angle α' for a Joukowski airfoil

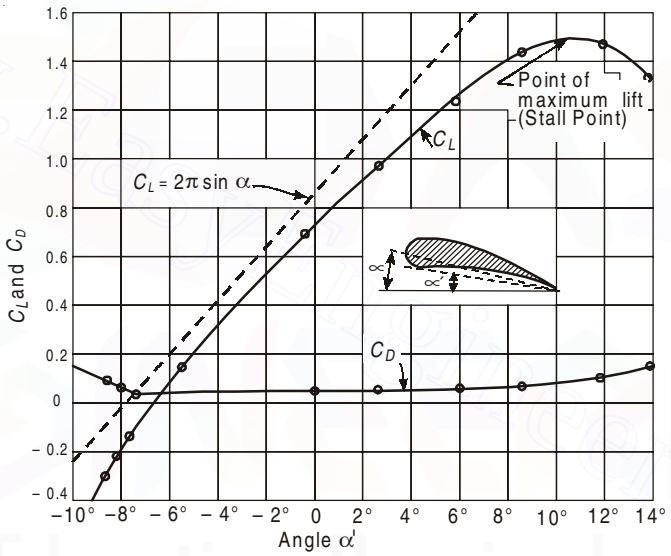


Figure 18.21 *Lift and drag coefficients of a typical Joukowski airfoil of infinite span*

is shown in Fig. 18.12. The dotted line on the left is the plot of Eq. 18.33 which represents the theoretical value of the lift coefficient for an airfoil in an irrotational flow. The solid lines represent the actual values of the lift and drag coefficients obtained experimentally. It may be seen from this plot that the experimental curve for lift coefficient C_L is very close to the theoretical curve for small values of angle α' (or small values of angle of attack α) thereby indicating that the actual flow pattern developed for the flow of real fluid past an airfoil is very much similar to that for an irrotational flow. However, when the angle α' reaches about 8°, then a marked deviation develops between the actual and the theoretical curves. With further increase in the value of angle α' to about 10° the lift coefficient attains a maximum value and for $\alpha' > 10°$ there is decrease in the value of lift coefficient. The point corresponding to maximum lift coefficient (or lift) is known as *stall point* and the angle of attack corresponding to this point is known as *stalling angle.* The reduction in the lift coefficient (or lift) with increase in the angle of attack beyond the stalling angle is due to the separation of flow at some point on the upper side of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the airfoil, and the condition in which the flow separates from practically the whole of the upper surface of the airfoil is known as *stall*.

It may be seen from Fig. 18.21 that the drag coefficient C_D of an airfoil varies little with the angle α' (or the angle of attack α), and it is only at the higher values of the angle α', since the flow separates, the drag coefficient C_D is slightly increased.

The lifting efficiency for any body is defined by the ratio (C_L/C_D) (or lift drag ratio). It may be seen from Fig. 18.21 that for an airfoil the ratio (C_L/C_D) is much higher as compared to that for a rotating cylinder (see Fig. 18.17) and hence airfoil is a superior lifting device. However, beyond the stall point there is a sudden drop in the lift-drag ratio of an airfoil.

(c) Effect of Fluid Compressibility on the Lift on an Airfoil. Experiments have shown that in subsonic compressible flow the effect of fluid compressibility is to cause an increase in the coefficient of lift C_L at a given angle of attack α. This increase in the lift coefficient can be computed by the so called *Prandtl-Glauert* rule which states that in subsonic compressible flow past thin symmetrical airfoil of infinite span and small angle of attack, the lift coefficient is increased by a factor $(1-Ma^2)^{-1/2}$, so that

$$(C_L)_{comp} = \frac{(C_L)_{incomp}}{\sqrt{1-Ma^2}} \qquad \ldots(18.34)$$

where Ma is the Mach number of the flow in the free stream. From equation 18.33, the lift coefficient for incompressible flow is expressed as

$$(C_L)_{incomp} = 2\pi \sin \alpha \approx 2\pi\alpha$$

for small angle of attack. Thus Eq. 18.34 becomes

$$(C_L)_{comp} = \frac{2\pi \sin \alpha}{\sqrt{1-Ma^2}} \approx \frac{2\pi\alpha}{\sqrt{1-Ma^2}} \qquad \ldots(18.35)$$

When the free stream velocity is increased from a subsonic to supersonic range, due to the formation of shock waves, there is a decrease in the value of C_L with the increase in the value of Ma. For supersonic flow past thin symmetrical airfoils of infinites span and small angle of attack the lift coefficient is approximately given by

$$C_L = \frac{4\alpha}{\sqrt{Ma^2 - 1}} \qquad \ldots(18.36)$$

18.11 INDUCED DRAG ON AN AIRFOIL OF FINITE LENGTH

In the previous section an airfoil of infinite span was considered, so that only a two-dimensional flow could occur around the airfoil with no end effects. However, if an airfoil of a finite span or length L is placed in a fluid stream, then the flow of fluid also takes place along the two ends, on account of which both the drag and the lift will be affected. When a lift is exerted on an airfoil, the pressure on the underside of the airfoil is greater than that on the upper side. This causes the fluid to escape at the two ends of the airfoil in a direction from below the airfoil towards the top. Therefore, there is a general flow of fluid in the direction outward from the centre to the two ends along the underside of the airfoil, it turns upward around the ends and flows inward from the two ends towards the centre along the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

upper side of the airfoil as shown in Fig. 18.22. This flow of fluid around the two ends of the airfoil produces vortices called *tip vortices* which are trailing from the two ends of the airfoil as shown in Fig. 18.22. The trailing tip vortices cause a downward velocity V_i, the effect of which may be considered

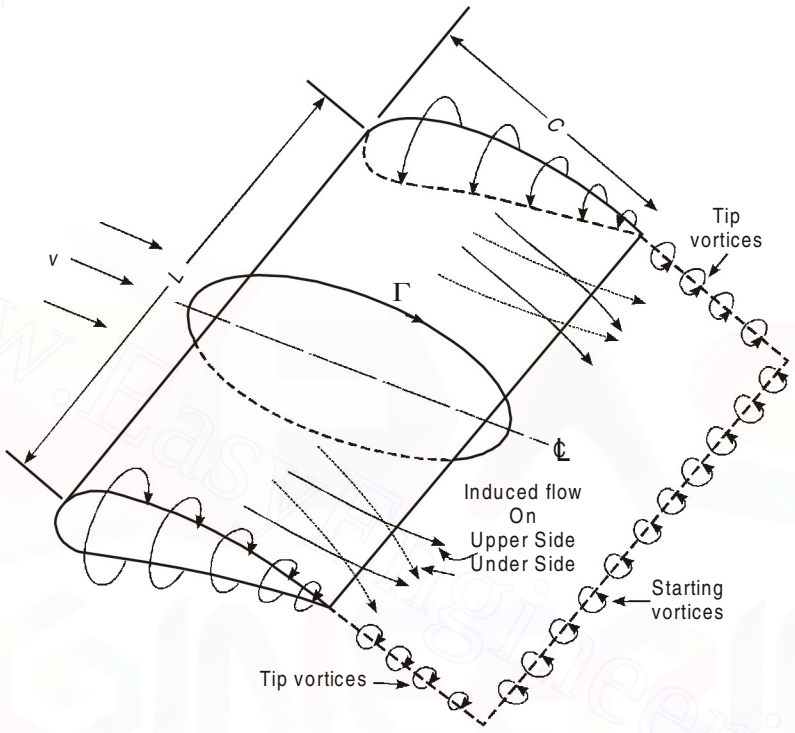


Figure 18.22 *Vortex system for an airfoil of finite span*

by adding it vectorially to the free stream velocity V, so that the effective velocity becomes V_0 as shown in Fig. 18.23. Further as may be seen in Fig. 18.23, the effect of the downward velocity is to reduce the angle of attack from α to α_0, where $\alpha_0 = (\alpha - \alpha_i)$ is the new angle of attack with respect to the direction of V_0 and $\alpha_i = \tan^{-1}\left(\frac{V_i}{V}\right)$. The finite airfoil may now be treated as one having infinite span placed in a stream of velocity V_0 and set at an angle of attack α_0. By definition, the lift F_{L_0} exerted on an infinite airfoil is perpendicular to the direction of V_0, and hence a force F_{L_0} is exerted on the finite airfoil as shown in Fig. 18.23. The force F_{L_0} can be resolved into two components, one perpendicular to the direction of the free stream velocity V and another parallel to the direction of V. The component of F_{L_0} in the direction perpendicular to V is F_L, the lift on the finite airfoil and the component parallel to V is F_{D_i}, called the *induced drag*. The induced drag F_{D_i} may also be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F_{D_i} = C_{Di}\, cL\, \frac{\rho V^2}{2} \qquad \ldots(18.37)$$

where C_{D_i} is the *coefficient of induced drag*.

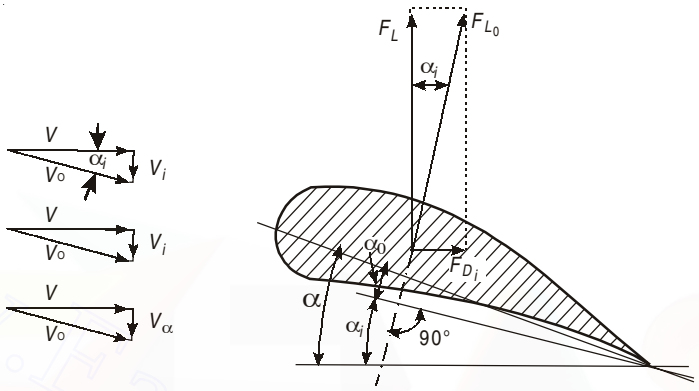


Figure 18.23 *Induced drag and lift on an airfoil of finite span*

From the foregoing analysis, it can be seen that in the case of a finite airfoil, induced drag is exerted which is in addition to the normal drag exerted on an airfoil of infinite span as indicated in Sec. 18.7. As such the total drag F_{DT} on a finite airfoil is given by

$$F_{DT} = F_{D0} + F_{Di} \qquad \ldots(18.38)$$

where F_{D0} is the drag on an airfoil of infinite span due to velocity V. By dividing each term in Eq. 18.38 by $\left(\frac{\rho c L V^2}{2}\right)$ the drag coefficient can be related as

$$C_{DT} = C_{D0} + C_{Di} \qquad \ldots(18.39)$$

Since the angle α_i is usually small, from Fig. 18.23, we have

$$F_L \approx F_{L0}\,;\, F_{Di} \approx F_L\, \alpha_i\,;\, V \approx V_0$$
$$V_i \approx V\alpha_i \approx V_0\, \alpha_i\,;\, C_{Di} \approx C_L\, \alpha_i.$$

By assuming an elliptical distribution of lift on an airfoil of finite span, Prandtl has obtained the following approximate expression for the coefficient of induced drag

$$C_{Di} = \frac{C_L^2}{(\pi L/c)} \qquad \ldots(18.40)$$

in which (L/c) is the aspect ratio of the airfoil. Further from the above noted expressions

$$\alpha_i \approx \frac{F_{Di}}{F_L} \approx \frac{C_L}{(\pi L/c)} \qquad \ldots(18.41)$$

Equation 18.40 shows that as the span of airfoil L becomes relatively large or the chord length c becomes relatively small the coefficient of induced drag tends to become zero.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

18.12 POLAR DIAGRAM FOR LIFT AND DRAG OF AN AIRFOIL

The various drag and lift coefficients viz., C_{Di}, C_D, C_{D0}, and C_L of an airfoil having a particular aspect ratio may be determined by testing the airfoil at different angles of attack in a wind tunnel. The values of these coefficients obtained by such tests may be presented graphically by plotting these coefficients against the angle of attack α or angle α' as shown in Fig. 18.21. These plots thus indicate the variation

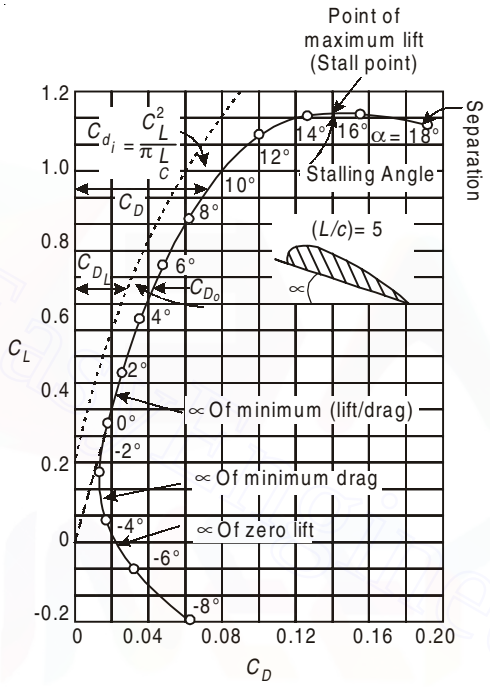


Figure 18.24 *Polar diagram for an airfoil of aspect ratio 5*

of the coefficients with the angle of attack α or angle α'. However, the variation of these coefficients may also be shown by a single curve known as the *polar diagram,* shown in Fig. 18.24, which was developed by Prandtl. In this diagram the lift coefficient C_L is plotted against the drag coefficient C_D. The corresponding angles of attack are represented by points on the curve. The slope of a line drawn from the origin to a point on the curve will thus give the ratio of lift to drag, and the maximum value of this ratio will be equal to the slope of a line tangential to the curve and drawn from the origin. The points corresponding to zero lift and the minimum drag are also shown on the curve. Further the stall point as well as the stalling angle are also indicated on the curve. On the same diagram Eq. 18.40 is also plotted as shown by dotted parabola at the left which represents the induced drag for the airfoil of the same aspect ratio (L/c) as adopted for testing. The horizontal intercept between the two curves will represent the drag coefficient C_{D0} for flow around an airfoil of infinite length. As indicated in section 18.11, since the induced angle of attack α_i is usually small, $F_L = F_{L0} \cos \alpha_i \approx F_{L0}$ the lift coefficient is same for airfoils of finite and infinite span.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ILLUSTRATIVE EXAMPLES

Example 18.1. *A circular disc 3 m in diameter is held normal to a 26.4 m/s wind of density 1.2 kg/m^3 . What force is required to hold it at rest? Assume coefficient of drag of disc as 1.1.*

Solution

The force F required to hold the plate at rest is equal to the drag exerted on it and from Eq. 18.5

$$F = F_D = C_D A \rho \frac{V^2}{2}$$

$$C_D = 1.1$$

$$A = \frac{\pi D^2}{4} = \frac{\pi \times (3)^2}{4} = 7.069 \text{ m}^2$$

$$\rho = 1.2 \text{ kg/m}^3$$

$$V = 26.4 \text{ m/s}$$

Thus by substitution

$$F = \frac{1.1 \times 7.069 \times 1.2 \times (26.4)^2}{2}$$

$$= 3251.69 \text{ N}$$

Example 18.2. *Calculate the total drag, shear drag and the pressure drag exerted on 1m length of an infinite circular cylinder which has a diameter equal to 30 mm, air of density 1.236 kg/m^3 flowing past the cylinder with velocity 3.6 m per minute. Take total drag coefficient equal to 1.4 and shear drag coefficient equal to 0.185.*

Solution

Total drag $$F_D = C_D A \rho \frac{V^2}{2}$$

$$C_D = 1.4 \; ; A = (1 \times 0.03) = 0.03 \text{ m}^2$$

$$\rho = 1.236 \text{ kg/m}^3$$

$$V = \frac{3.6}{60} = 0.06 \text{ m/s}$$

$\therefore$ Total drag $$F_D = \frac{1.4 \times 0.03 \times 1.236 \times (0.06)^2}{2}$$

$$= 9.344 \times 10^{-5} \text{ N}$$

Shear drag $$F_{Df} = C_{Df} A \rho \frac{V^2}{2}$$

$$= \frac{0.185 \times 0.03 \times 1.236 \times (0.06)^2}{2}$$

$$= 1.235 \times 10^{-5} \text{ N}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Pressure drag $F_{DP} = (F_D - F_{Df})$

$= (9.344 \times 10^{-5} - 1.235 \times 10^{-5})$

$= 8.109 \times 10^{-5}$ N

Example 18.3. *A semi-tubular cylinder of 75 mm radius with concave side upstream (drag coefficient = 2.3) is submerged in flowing water of velocity 0.6 m/s. If the cylinder is 7.2 m long, calculate the drag. Assume density of water as 1000 kg/m^3.*

Solution

Drag $F_D = C_D A \rho \frac{V^2}{2}$

$C_D = 2.3$

$A = (0.075 \times 7.2) = 0.54$ m^2

$\rho = 1000$ kg/m^3

$V = 0.6$ m/s

Thus by substitution, we get

$$F_D = \frac{2.3 \times 0.54 \times 1000 \times (0.6)^2}{2}$$

$= 223.56$ N

Example 18.4. *The drag coefficient C_D of a 5 kN and 0.5 m diameter bomb as a function of the Mach number Ma is tabulated below. Linear interpolation may be used between tabulated values.*

Ma	*0.50*	*0.75*	*1.00*	*1.25*	*1.50*
C_D	*0.25*	*0.28*	*0.35*	*0.55*	*0.70*

If the bomb is dropped from a high flying plane, estimate the terminal velocity with which it will strike the earth. Near the ground surface assume atmospheric pressure as 101 kN/m^2 and temperature as 290.15 K. Assume the gas constant for air as 287 N.m/kg.K and adiabatic index k = 1.4.

Solution

From equation of state, we have

$$\rho = \frac{p}{RT}$$

By substituting the gives values, we get

$$\rho = \frac{101 \times 10^3}{287 \times 290.15} = 1.213 \text{ kg/m}^3$$

Velocity of sound is given by

$C = \sqrt{kRT}$

$= \sqrt{1.4 \times 287 \times 290.15} = 341.44$ m/s

Since Mach number

$$Ma = \frac{V}{C}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V = Ma \times C$$

or $$V = 341.44\ Ma \quad \text{...(i)}$$

Also $$C_D = \frac{5 \times 10^3}{(\pi/4)(0.5)^2 \times 1.213 \times (V^2/2)}$$

or $$C_D = \frac{41986.46}{V^2} \quad \text{...(ii)}$$

From Eqs (i) and (ii), we have

$$C_D = \frac{41986.46}{(341.44)^2 \times (Ma)^2}$$

or $$C_D = \frac{0.3601}{(Ma)^2}$$

Solving by trial with the help of given table, we get

$$Ma = 1.0068$$

$\therefore$ $$V = (1.0068 \times 341.44) = 343.76 \text{ m/s}$$

Example 18.5. *A submarine which may be supposed to approximate a cylinder 3 m in diameter and 15 m long travels submerged at 1.54 m/s in sea water at 4° C. Find the drag exerted on it. Take υ for sea water at 4°C as 1.67× 10^{-6} m^2/s and ρ as 1025 kg/m^3. Also find the friction drag.*

Solution

$$Re = \frac{VD}{\upsilon}$$

$$= \frac{1.54 \times 3}{1.67 \times 10^{-6}} = 2.77 \times 10^6$$

Drag $$F_D = C_D A \rho \frac{V^2}{2}$$

From Fig. 18.10, for *Re* equal to 2.77×10^6, $C_D = 0.7$

$$A = (3 \times 15) = 45 \text{ m}^2$$

$$V = 1.54 \text{ m/s}$$

and $$\rho = 1025 \text{ kg/m}^3$$

Thus by substitution, we get

$$F_D = \frac{0.7 \times 45 \times 1025 \times (1.54)^2}{2} = 38\ 287 \text{ N}$$

At such a high value of Re the viscous effects are negligible and hence the friction drag will be quite negligible and it may be considered to be zero.

Example 18.6. *A cylinder 1.2 m in diameter is rotated about its axis in air having a velocity of 128 km per hour. A lift of 5886 N per metre length of the cylinder is developed on the body. Assuming ideal fluid theory, find the rotational speed and the location of the stagnation points. Given $\rho = 1.236$ kg/m^3.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

From Eq. 18.28 the lift on the cylinder is given as

$$F_L = \rho L V \Gamma$$

$$(F_L/L) = 5{,}886\ N;\ \rho = 1.236\ \text{kg/m}^3$$

and
$$V = \frac{128 \times 1000}{60 \times 60} = 35.56\ \text{m/s}$$

Thus by substitution, we have

$$5886 = 1.236 \times 35.56 \times \Gamma$$

or
$$\Gamma = \frac{5886}{1.236 \times 35.56} = 134\ \text{m}^2/\text{s}$$

But from Eq. 18.26, we have

$$V_c = \frac{\Gamma}{2\pi R}$$

and
$$V_c = R\omega = \left(\frac{2\pi N}{60}\right) R$$

where ω is angular velocity and N is the rotational speed of the cylinder in r.p.m. Thus we have

$$\frac{\Gamma}{2\pi R} = \left(\frac{2\pi N}{60}\right) R$$

or
$$N = \frac{60\Gamma}{4\pi^2 R^2}$$

$$= \frac{60 \times 134}{4 \times \pi^2 \times (0.6)^2} = 566\ \text{r.p.m.}$$

For the stagnation points from Eq. 18.27, we have

$$v = 2V \sin\theta + \frac{\Gamma}{2\pi R} = 0$$

or
$$2 \times 35.56 \sin\theta + \frac{134}{2 \times \pi \times 0.6} = 0$$

or
$$\sin\theta = -\frac{1}{2}$$

$\therefore$
$$\theta = -30^\circ \text{ and } +210^\circ$$

Example 18.7. *The wing of an aeroplane is designed to develop a lift of 5×10^4 N. If the span is 10.5 m and the mean chord is 1.8 m, calculate the total drag at a speed of 400 km per hour. Assume elliptical lift distribution. $\rho = 1.207\ kg/m^3$ and profile drag coefficient is 0.012.*

Solution

$$C_L = \frac{F_L}{A\rho \dfrac{V^2}{2}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F_L = 5 \times 10^4 \text{ N}$$
$$A = (10.5 \times 1.8) = 18.9 \text{ m}^2$$
$$\rho = 1.207 \text{ kg/m}^3$$

and $$V = \frac{400 \times 1000}{60 \times 60} = 111.1 \text{ m/s}$$

Thus by substitution, we have

$$C_L = \frac{5 \times 10^4 \times 2}{18.9 \times 1.207 \times (111.1)^2} = 0.355$$

From Eq. 18.40, induced drag coefficient

$$C_{Di} = \frac{C_L^2}{\pi L/c} = \frac{(0.355)^2 \times 1.8}{\pi \times 10.5} = 0.006\ 88$$

Profile drag coefficient

$$C_{D0} = 0.012$$

Total drag coefficient

$$C_{DT} = (0.012 + 0.006\ 88)$$
$$= 0.018\ 88$$

$\therefore$ Drag $$F_D = 0.018\ 88 \times 18.9 \times 1.207 \times \frac{(111.1)^2}{2}$$
$$= 2658.09 \text{ N}$$

Example 18.8. *Experiments were conducted in a wind tunnel with a wind speed of 50 km/h on a flat plate of size 2 m long and 1m wide. The specific weight of air is 11.282 N/m³. The plate is kept at such an angle that the coefficients of lift and drag are 0.75 and 0.15 respectively. Determine (i) lift, (ii) drag, (iii) resultant force, and (iv) power exerted by the air stream on the plate.*

Solution

$$V = \frac{50 \times 1000}{60 \times 60} = 13.89 \text{ m/s}$$

$$\rho = \frac{11.282}{9.81} = 1.150 \text{ kg/m}^3$$

(i) Lift $$F_L = C_L A \rho \frac{V^2}{2}$$

$$= 0.75 \times (2.0 \times 1.0) \times 1.150 \times \frac{(13.89)^2}{2}$$
$$= 166.40 \text{ N}$$

(ii) Drag $$F_D = C_D A \rho \frac{V^2}{2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 0.15 \times (2.0 \times 1.0) \times 1.150 \times \frac{(13.89)^2}{2}$$

$$= 33.28 \text{ N}$$

(iii) Resultant force

$$F = \sqrt{F_L^2 + F_D^2}$$

$$= \sqrt{(166.40)^2 + (33.28)^2}$$

$$= 169.7 \text{ N}$$

inclined at an angle θ with the velocity of air such that

$$\tan\theta = \frac{F_L}{F_D} = \frac{166.40}{33.28} = 5.0$$

$$\therefore \quad \theta = 78°42'$$

(iv) Power exerted by the air stream is

$$P = (33.28 \times 13.89)$$

$$462.26 \text{ watts}$$

Example 18.9. *A kite weighing 12.26 N has an effective area of 0.9 m^2. The tension in the kite string is 32.37 N when the string makes an angle of 45° with the horizontal. For a wind of 32 km/hour, what are the coefficients of lift and drag if the kite assumes an angle of 8° with the horizontal? Take specific weight of air as 11.801 kg/m^3.*

Solution

The free-body diagram for the kite is shown below:

By resolving the force along the direction of the velocity of the wind, we have

$$F_D = 32.37 \cos 45° = \frac{32.37}{\sqrt{2}} = 22.89 \text{ N}$$

Resolving the forces along the direction perpendicular to that of the wind velocity, we have

$$F_L = (32.37 \sin 45° + 12.26)$$

$$= 35.15 \text{ N}$$

$$C_D = \frac{F_D}{\rho A \frac{V^2}{2}}$$

$$\rho = \frac{11{\cdot}801}{9{\cdot}81}$$

$$= 1.203 \text{ kg/m}^3$$

$$A = 0.9 \text{ m}^2$$

$$V = \frac{32 \times 1000}{60 \times 60} = 8.89 \text{ m/s}$$

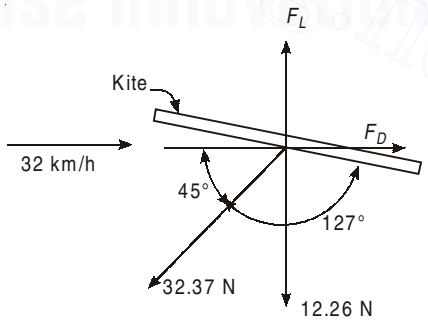


Figure Ex. 18.9

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $$C_D = \frac{22.89 \times 2}{1.203 \times 0.9 \times (8.89)^2} = 0.535$$

Similarly $$C_L = \frac{F_L}{\rho A \frac{V^2}{2}}$$

$$= \frac{35.15 \times 2}{1.203 \times 0.9 \times (8.89)^2} = 0.822$$

Note. The effect of the angle of attack (8° in this case) is implicit in the drag and lift exerted on the kite and hence in the solution of the problem no separate consideration of this angle is required to be made.

Example 18.10. *A cylinder of radius 0.5 m is held submerged in a stream of ideal fluid having an undisturbed velocity of flow 10 m/s. Find the value of $(p_1 - p_0)/\rho$ where p_1 is the pressure at a point in the flow field with coordinates r=1 m and $\theta = 30°$; p_0 is the pressure at a point far away on the upstream where the stream has undisturbed velocity of flow and ρ is the mass density of the flowing fluid.*

Solution

Applying Bernoulli's equation between the given points, we have

$$\frac{p_0}{\rho} + \frac{V_0^2}{2} = \frac{p_1}{\rho} + \frac{V_1^2}{2}$$

or $$\frac{p_1 - p_0}{\rho} = \frac{V_0^2 - V_1^2}{2}$$

$$V_1 = \sqrt{V_r^2 + V_\theta^2}$$

$$V_r = V_0 \cos\theta \left(1 - \frac{R^2}{r^2}\right)$$

$$= 10 (\cos 30°) \times \left(1 - \frac{1}{4}\right) = 6.495 \text{ m/s}$$

$$V_\theta = V_0 \sin\theta \left(1 + \frac{R^2}{r^2}\right)$$

$$= 10(\sin 30°) \times \left(1 + \frac{1}{4}\right) = 6.250 \text{ m/s}$$

Thus $$V_1 = \sqrt{(6.495)^2 + (6.250)^2} = 9.014 \text{ m/s}$$

$\therefore$ $$\frac{p_1 - p_0}{\rho} = \frac{(10)^2 - (9.014)^2}{2} = 9.374 \text{ m}^2/\text{s}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. When a body moves through a fluid it experiences a force which tends to oppose its motion. The component of this force in the direction of motion is called drag and the component in the direction perpendicular to the direction of motion is called lift.
2. For a body moving through a fluuid of mass density ρ at a uniform velocity V, the mathematical expressions for the drag F_D and the lift F_L may be written as

$$F_D = C_D A \frac{\rho V^2}{2}$$

and $$F_L = C_L A \frac{\rho V^2}{2}$$

where

C_D = coefficient of drag;
C_L = coefficient of lift; and
A = projected area of the moving body on a plane perpendicular to the direction of motion.

3. The total drag exerted on any body moving through a fluid is equal to the sum of the surface or friction drag and the form or pressure drag.
4. The relative magnitude of the friction drag and pressure drag depends on the shape and the position of the moving body. Thus if a thin plate is moving through a fuid with its surface parallel to the direction of motion, the pressure drag is practically equal to zero and hence in this case the total drag is equal to the friction drag. On the other hand if the same plate is moving with is surface perpendicular to the direction of motion, the friction drag is practically equal to zero and hence in this case the total drag is equal to the pressure drag.
5. The drag on a sphere for Reynolds number less than 0.2 is given by

$$F_D = 3\pi\mu VD$$

Out of this total drag

$$\text{Skin friction drag} = \frac{2}{3} F_D = 2\pi\mu VD \text{ ; and}$$

$$\text{Pressure drag} = \frac{1}{3} F_D = \pi\mu VD$$

6. For a sphere the coefficient of drag for Reynolds number less than 0.2 is given by

(i) When fluid is of infinite extent

$$C_D = \frac{24}{\text{Re}}$$

(ii) When fluid is of limited extent

$$C_D = \frac{24}{\text{Re}}\left[1 + 2.1\frac{D}{D_1}\right]$$

Where

D = diameter of the sphere; and
D_1 = smallest lateral dimension of the container.

9. According to oseen for a sphere the coefficient of drag is given by

(i) $$C_D = \frac{24}{\text{Re}}\left[1 + \frac{3}{16}\text{Re}\right] \quad \text{for Re} < 1$$

(ii) $$C_D = \frac{24}{\text{Re}}\left[1 + \frac{3}{16}\text{Re}\right]^{1/2} \quad \text{for Re} \le 100$$

The values of C_D for a sphere and α circular disc may be obtained from a graphucal plot showing the variation of C_D with Re.

10. The velocity of ideal fluid at any point on the surfaceof a cylinder is given by

$$v = 2V \sin\theta$$

where

v = tangential velocity on the surface of the cylinder;
V = velocity of flow of fluid, and
θ = angular distance of the point from the front or the rear stagnation point.

11. As the fluid flows past a cylinder, when Re exceeds about 30, trails of vortices are shed on the downstream side, which are known as *Karman vortex trails*. The frequency $f\left(=\frac{V - V_V}{b}\right)$ with which the vortices are shed alternately from the cylinder on the downstream side is given by

$$f = 0.198 \frac{V}{D}\left[1 - \frac{19.7}{\text{Re}}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\frac{fD}{V} = 0.198\left[1-\frac{19.7}{\text{Re}}\right]$$

The parameter $\frac{fD}{V}$ is dimension less and it is called Strouhal number.

12. The variation of C_D with Re for a cylinder and a flat plate is shown in a graphical plot.
13. An airfoil is a streamlined body for which the separation occurs only at the extreme rear of the body. As such in the case of an airfoil even at high values of Re, it is the skin friction drag which makes the major contribution to the total drag and the contributin of the pressure drag is very small.
14. When the velocity approaches the velocity of sound the coefficient of drag becomes a function of Mach number *Ma*. The variation of C_D with *Ma* for different objects is shown in a graphical plot.
15. When a cylinder moving through a fluid is imparted a constant circulation Γ, the velocity at any point on the surface of the cylinder is given by

$$v = 2V\sin\theta + \frac{\Gamma}{2\pi R}$$

The total lift F_L exerted on the cylinder is given by Kutta-Jaukowski equatin as

$$F_L = \rho LV\Gamma$$

where

L = length of the cylinder; and
R = radius of the cylinder.

The coefficient of lift for the cylinder in this case is given by

$$C_L = \frac{\Gamma}{RV}$$

or $$C_L = \frac{2\pi V_C}{V}, \text{ since } V_C = \frac{\Gamma}{2\pi R}$$

16. The location of the stagnation points for the cylinder in this case is given by

$$\sin\theta = \frac{\Gamma}{4\pi VR}$$

The two stagnation points will coincide when circulation is given by

$$\Gamma = 4\pi VR$$

17. The circulation developed around an airfoil is given by

$$\Gamma = \pi cV\sin\alpha$$

where

c = chord length of the airfoil; and
α = angle of attack

The lift F_L on an airfoil of span L is given by

$$F_L = \pi c\rho V^2\sin\alpha$$

The coefficient of lift for the airfoil is given by

$$C_L = 2\pi\sin\alpha$$

Due to fluid compressibility the coefficient of lift for the airfoil increases and it is given by

$$C_L = \frac{2\pi\sin\alpha}{\sqrt{1-M_a^2}}$$

18. The airfoil of finite length is subjected to induced drag F_{Di} in addition to the normal drag F_{Do} exerted on an airfoil of infinite span. As such the total drag F_{DT} on a finite airfoil is given by

$$F_{DT} = F_{DO} + F_{Di}$$

PROBLEMS

18.1 What is the difference between friction drag and pressure drag? In what position should a flat plate be held in a flow to subject the plate to these forces?

18.2 Distinguish between deformation drag, surface drag and form drag. In the case of a sphere, discuss their relative importance at various increasing values of Re.

18.3 Draw and explain the approximate flow pattern and the pressure distribution around a flat plate placed perpendicularly in a stream flow.

18.4 Why should circulation superimposed on flow past a body cause a lift?

18.5 Explain the origin of circulation around an airfoil and discuss the effects of the angle of attack (or incidence angle) on flow pattern around the airfoil.

18.6 Distinguish between form drag and surface drag. Sketch roughly the flow patterns and state the relative magnitudes of these drags for the following two dimensional objects: (i) flat plate set normal to the flow; (ii) a cylinder; (iii) a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

streamlined strut. The areas of these objects projected on a vertical plane are equal.

18.7 What is meant by Magnus effect? How does circulation originate around an airfoil?

18.8 No drag can be exerted upon a body by steady irrotational flow. Why is this true? Why is it not true if the flow is unsteady?

18.9 In investigating the possibility of using rotors in place of airplane wings it is assumed that each of the two rotating cylinders would have a diameter of 1 m and length 4.5 m. If the weight of the entire plane is 4.5×10^4 N, determine the speed of rotation of rotors which will support this load at a 240 km/hour cruising speed. Use Fig. 18.17 for determining the drag and lift coefficients. Also determine the power required to overcome the rotor drag. Take $\rho = 1.236$ kg/m^3.

[*Ans*. 1273 r.p.m; 1648 kW]

18.10 Calculate the diameter of a parachute to be used for dropping an object weighing 1000 N so that the maximum terminal velocity of dropping is 5 m/s. The drag coefficient for parachute which may be treated as hemispherical is 1.3 and the value of ρ for air is 1.216 kg/m^3 [*Ans*. 8.03 m]

18.11 A cylinder of 0.50 m diameter is rotated at 540 r.p.m. in air stream of velocity 12 m/s. If it develop a lift of 96 N per metre length of the cylinder, determine the ratio of actual to theoretical lift. Take $\rho = 1.236$ kg/m^3.

[*Ans.* 0.291]

18.12 The drag coefficient C_D of a 4 kN and 0.45 m diameter bomb as a function of the Mach number *Ma* is tabulated below. Linear interpolation may be used between tabulated values.

Ma	0.4	0.6	0.8	1.0	1.2
C_D	0.2	0.23	0.3	0.5	0.65

If the bomb is dropped from a high flying plane, estimate the terminal velocity with which it will strike the earth. Near the ground surface, assume atmospheric pressure as 101.04 kN/m^2 and temperature as 17°C. Assume the gas constant for air as 287.73 joules/kg.K and adiabatic index k = 1.4. [*Ans*.314.37 m/s]

18.13 A cloth banner 3 m wide and 0.8 m high, meant for advertisement, is mounted on poles. Calculate the net force acting on the banner when wind blows at 50 km/h. Assume $C_D = 1.1$, $\rho = 1.25$ kg/m^3 (0.127 msl/m^3). If a number of holes are punched on the cloth how would the result change and why?

[*Ans*. 318.34 N {32.34 kg (f)}]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Flow of Compressible Fluids

Chapter 19

19.1 INTRODUCTION

In general all the fluids are compressible, but in any fluid flow system if the density of the fluid does not change appreciably, then the fluid may be treated as incompressible. However, when the density changes are appreciable, the compressibility of the fluid needs to be taken into account in the analysis of such fluid flow phenomena. Significant density changes may usually be produced in a gas if the velocity (either of the gas itself or of a body moving through it) approaches or exceeds the speed of propagation of sound through the gas, if the gas is subjected to sudden acceleration; or if there are very large changes in elevation. The compressibility is also significant in certain problems involving the flow of water and one such example is the phenomenon of water hammer in pipes. In this chapter the discussion is however confined to only those problems of fluid flow which involve the variations of density of highly compressible fluids such as gases due to the changes of pressure and velocity.

The density of a gas is related to both the pressure and the temperature and all the changes of density are governed by certain laws of thermodynamics. As such in the next paragraph some of the basic relationships of thermodynamics are briefly reviewed.

19.2 BASIC RELATIONSHIP OF THERMODYNAMICS

(I) **Isothermal and Adiabatic Processes.** When the physical properties of a gas such as pressure, density and temperature are changed due to compression or expansion of the gas, it is said to undergo a *process*. A gas may be compressed or expanded either by *isothermal process* or by *adiabatic process*. An isothermal process is that in which the temperature is held constant and it is governed by Boyle's law according to which

$$\left.\begin{aligned} pv &= p_1v_1 = p_2v_2 &&= \text{constant} \\ \text{or} \quad (p/\rho) &= (p_1/\rho_1) = (p_2/\rho_2) &&= \text{constant} \\ (p/w) &= (p_1/w_1) = (p_2/w_2) &&= \text{constant} \end{aligned}\right] \quad \ldots (19.1)$$

On the other hand if during the course of a process, the gas neither absorbs heat from, nor gives heat to, its surroundings then the process is said to be adiabatic. An adiabatic process is governed by the following law:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left.\begin{aligned} pv^k &= p_1 v_1^k = p_2 v_2^k &&= \text{constant} \\ (p/\rho^k) &= (p_1/\rho_1^k) = (p_2/\rho_2^k) &&= \text{constant} \\ (p/w^k) &= (p_1/w_1^k) = (p_2/w_2^k) &&= \text{constant} \end{aligned}\right\} \quad \text{or} \quad \ldots (19.2)$$

In the above equation p is pressure, v is specific volume, ρ is mass density, w is weight density, and k is adiabatic index. The adiabatic index is given by $k = (C_p/C_v)$, the ratio of the specific heat at constant pressure C_p and the specific heat at constant volume C_v. For air and other diatomic gases in the usual ranges of temperature and pressure $k = 1.4$.

The process is said to be *reversible* if the gas and its surroundings could subsequently be completely restored to their initial conditions by adding to (or extracting from) the gas exactly the same amount of heat and work taken from (or added to) it during the process. A frictionless adiabatic process is a reversible process and it is often termed as *isentropic process* because as indicated later in this process the entropy does not change. However, a reversible process is ideal, i.e., it is never achieved in practice.

There are some actual processes which are not isentropic (so that pv^k is not constant) but in which the relation between p and v (or ρ) is given to a reasonable approximation by pv^n = constant (or p/ρ^n = constant) where n is a positive constant. Such processes are termed as polytropic processes.

(II) **Equation of State.** The density ρ of a particular gas is related to its absolute pressure p and absolute (or thermodynamic) temperature T by the *equation of state*, which for a perfect gas takes the form

$$p = \rho RT; \text{ or } pV = mRT$$

in which R is *gas constant* and V is the volume occupied by the mass m of the gas. The absolute (or thermodynamic) temperature is expressed in 'kelvin' (K) when the temperature is measured in °C and it is given by

$$T° \text{ (abs)} = T\text{K} = 273.15 + t°\text{C}$$

Further details of the equation of state including the dimensional formula of the gas constant R and its value for air are given in Chapter 1.

(III) **Internal Energy.** The energy possessed by the molecules of a fluid due to molecular activity is termed as *internal energy* or *microscopic energy*. It is to be distinguished from the external or macroscopic energies viz., kinetic and potential energies associated with the fluid mass. The internal energy is a property of the fluid which cannot be measured directly but manifests itself in terms of temperature, that is high or low temperature implies high or low internal energy respectively. If some quantity of heat is supplied to a certain mass of gas, a part of it may be stored in the gas as internal energy thus producing the rise in temperature of the gas and the remaining part of the heat supplied may be utilized in increasing the volume of the gas thus doing some external work.

(IV) **First Law of Thermodynamics.** The first law of thermodynamics states that the total heat H supplied to a gas must be equal to the increase in the internal energy $(I_2 - I_1)$ of the gas plus the external work W done by the gas in expanding. Thus

$$H = (I_2 - I_1) + W \quad \ldots (19.3)$$

However, if instead of heating, the gas is cooled then the heat will be rejected from the gas and hence H will be negative. The value of $(I_2 - I_1)$ may also be negative if the internal energy of the gas decreased. Further W is negative, if the gas is compressed and consequently contracts in volume, which means that work is done on the gas from some external source.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For isothermal process, since the temperature remains constant, $(I_2 - I_1) = 0$, and hence

$$H = W \qquad \text{... (19.3 a)}$$

On the other hand, for adiabatic process there being neither addition nor removal of heat, $H = 0$, and hence

$$W + (I_2 - I_1) = 0$$

or

$$W = (I_1 - I_2) \qquad \text{... (19.3 b)}$$

The work done by a gas in expanding (or on a gas in contracting) is given by

$$W = \int_{v_1}^{v_2} p\,dv \qquad \text{... (19.4)}$$

For isothermal process introducing the value of p from Eq. 19.1 in Eq. 19.4 and integrating it we obtain

or

$$\left.\begin{aligned} W &= p_1 v_1 \log_e (v_2 / v_1) = p_1 v_1 \log_e (p_1 / p_2) \\ W &= K \log_e (v_2 / v_1) = K \log_e (p_1 / p_2) \end{aligned}\right] \qquad \text{... (19.4 a)}$$

in which K is the constant of Eq. 19.1. Eq. 19.4 (a) gives the work done per unit mass or per unit weight of the gas depending on the specific volume being considered as the volume per unit mass or per unit weight of the gas respectively. Similarly for adiabatic process introducing the value of p from Eq. 19.2 in Eq. 19.4 and integrating it we obtain

$$W = \frac{1}{k-1}(p_1 v_1 - p_2 v_2) \qquad \text{... (19.4 b)}$$

Equation 19.4 (b) gives the work done per unit mass of the gas if the specific volume is considered as the volume per unit mass of the gas. However, if the specific volume is considered as the volume per unit weight of the gas then Eq. 19.4 (b) gives the work done per unit weight of the gas and it may then be expressed as

$$W = \frac{1}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) \qquad \text{... (19.4 c)}$$

(V) **Entropy.** *Entropy* of a gas may be defined as the measure of the maximum heat energy available for conversion into work. It is a property of the gas and it varies with its absolute temperature and its state. If ΔH is the heat transferred per unit weight of gas in a small interval of time and if T is the absolute temperature of the gas at that instant, then the change in the entropy $\Delta\phi$ is defined by the relation

$$\Delta\phi = \frac{\Delta H}{T}$$

or

$$(\phi_2 - \phi_1) = \int_{T_1}^{T_2} \frac{\Delta H}{T} \qquad \text{... (19.5)}$$

For isothermal process since the temperature remains constant

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(\phi_2 - \phi_1) = \frac{H}{T} \quad \text{... (19.5 a)}$$

But in adiabatic process since there is no transfer of heat, $\Delta H = 0$, hence

$$(\phi_2 - \phi_1) = 0;\ \phi = \text{constant} \quad \text{... (19.5 b)}$$

A process in which the entropy does not change is termed as *isentropic process*. However, the entropy does not change in an adiabatic process, only if it is frictionless. A frictionless adiabatic process is therefore isentropic, which is also reversible.

(VI) **Enthalpy.** The sum of the internal energy and pressure-specific volume product (or pressure divided by mass density; or pressure divided by weight density) (converted into heat units) is termed as *enthalpy*. It is a purely mathematical quantity, the consideration of which simplifies certain calculations, since in several problems of thermodynamics this sum occurs.

19.3 CONTINUITY EQUATION

The equation of continuity which is applicable for compressible as well as incompressible fluids has been developed in Chapter 6 which may be expressed as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

Further as discussed in Chapter 6 for one dimensional steady flow of compressible fluids the equation of continuity reduces to

$$\rho AV = \text{constant}$$

where ρ is the mass density of the fluid at any section of the flow passage having its cross-sectional area A and the average velocity of flow V. Obviously for the flow of compressible fluid all these viz., ρ, A and V may vary along the flow passage. The one dimensional equation of continuity for the flow of a compressible fluid may also be expressed in differential form as

$$d(\rho AV) = 0$$

19.4 MOMENTUM EQUATION

The momentum equation for compressible fluids is similar to the one derived in Chapter 8 for incompressible fluid. This is because in momentum equation the change in momentum flux is equated to the force required to cause this change. The momentum flux is the product of the mass flux (ρAV) and the velocity V. By the continuity equation the mass flux (ρAV) is constant from section to section. As such the momentum equation is completely independent of the compressibility effects and hence for compressible fluids too the momentum equation (for say x direction) may be expressed as

$$\Sigma F_x = (\rho AVVx)_2 - (\rho AVVx)_1.$$

19.5 ENERGY EQUATION

The total energy E_t at a point in a compressible fluid flow system comprises the following:

(1) Kinetic energy, E_k
(2) Potential energy, E_e
(3) Flow or pressure energy, E_p
(4) Molecular energy, E_m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

This may be expressed as

$$E_t = E_k + E_e + E_p + E_m \quad \text{... (19.6)}$$

According to the law of conservation of energy, the total energy at one point in a fluid flow system, plus the energy added E_a minus the energy subtracted E_s between two points, must be equal to the total energy at the second point in the system. That is

$$(E_k + E_e + E_p + E_m)_1 + E_a - E_s = (E_k + E_e + E_p + E_m)_2 \quad \text{... (19.7)}$$

Equation 19.7 shows that energy can be changed from one form to another on the two sides of the equation, but the total must be the same. In other words energy can neither be created nor destroyed.

For incompressible fluids, due to the resistance to the flow of fluid, a part of the energy is converted into heat energy which is considered indirectly as an 'energy loss'. However, for compressible fluids the heat energy produced or supplied may be stored as a molecular energy which is termed as internal energy and it must be included as a specific part of the energy equation. Since the molecular energy of a compressible fluid is developed due to activity of the molecules, it increases with the increase in the temperature of the fluid. Further as indicated earlier, in thermodynamics the sum of the pressure energy (p/w) and the internal energy I is known as enthalpy E_n, and hence $E_n = (p/w) + I$.

Equation 19.7 can thus be written in terms of the energy per unit weight of the flowing fluid as

$$\left(\frac{V^2}{2g} + z + E_n\right)_1 + (E_a - E_s) = \left(\frac{V^2}{2g} + z + E_n\right)_2 \quad \text{... (19.8)}$$

in which $(V^2/2g)$ is kinetic energy, z is potential energy and E_n is enthalpy, each of these being per unit weight of the fluid.

In the case of isothermal flow, since the temperature remains constant $I_1 = I_2$ and from Eq. 19.1 $(p_1/w_1) = (p_2/w_2)$, therefore $E_{n1} = E_{n2}$. Further E_a is usually the heat energy H, which for isothermal flow is equal to the work done and is given by Eq. 19.4 (a). Now if there is no loss of energy i.e., $E_s = 0$ then for a frictionless isothermal flow of fluid the energy equation becomes

$$K \log_e (p_1/p_2) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{... (19.9)}$$

In the case of adiabatic flow there being neither addition nor removal of heat energy $E_a = H = 0$. Also for this case from Eq. 19.3 (b) and 19.4 (c),

$$(I_1 - I_2) = W = \frac{1}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) \quad \text{... (19.10)}$$

Thus for this case

$$(E_{n1} - E_{n2}) = \frac{k}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) \quad \text{... (19.11)}$$

Again if there is no loss of energy i.e., $E_s = 0$, then for a frictionless adiabatic flow (or isentropic flow) of fluid the energy equation becomes

$$\frac{k}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{... (19.12)}$$

Equations 19.9 and 19.12 are same as those derived in Chapter 7.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further combining Eqs 19.2 and 19.12, we obtain

$$\frac{p_1}{w_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right] = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{... (19.13)}$$

For the flow of gas the potential head difference term $(z_2 - z_1)$ is usually small as compared to the pressure head terms, and hence it may be neglected in Eqs 19.9, 19.12 and 19.13.

19.6 PROPAGATION OF ELASTIC WAVES DUE TO COMPRESSION OF FLUID, VELOCITY OF SOUND

If the pressure at a point in a fluid is changed, the new pressure is rapidly transmitted throughout the rest of the fluid. It so happens because any change in pressure causes an elastic or pressure wave reflecting this change to travel through the fluid at a certain velocity. In a completely incompressible fluid the transmission of new pressure would take place instantaneously and hence the elastic or pressure wave would be propagated with infinite velocity. However, in a compressible fluid the elastic or pressure wave is propagated with finite velocity. But even in a compressible fluid the changes of pressure are transmitted quite rapidly and as indicated below the velocity of propagation of the elastic wave in a compressible fluid is equal to the velocity of the sound in that fluid medium.

Consider a long rigid tube of uniform cross-sectional area A fitted with a piston at one end as shown in Fig. 19.1. The tube is filled with a compressible fluid initially at rest. If the piston is moved suddenly to the right with a velocity V_p, a pressure wave would be propagated through the fluid at a velocity C. During a small time interval dt, the piston moves through a distance dx $(= V_p dt)$ while the pressure wave travels ahead of the piston through a distance $dL = (Cdt)$.

Before the sudden motion of the piston the fluid within the length dL has an initial density ρ, and its total mass is $(\rho dL\, A)$. After the piston is moved through a distance dx, the fluid density within the compressed region of length $(dL - dx)$ will be increased and becomes $(\rho + d\rho)$. Thus the total mass of fluid in the compressed region will be $[(\rho + d\rho)\,(dL - dx)A]$. Therefore, according to the principal of continuity

$$\rho\,(dL)\,A = (\rho + d\rho)\,(dL - dx)\,A$$

or

$$\rho\,(dL) = (\rho + d\rho)\,(dL - dx)$$

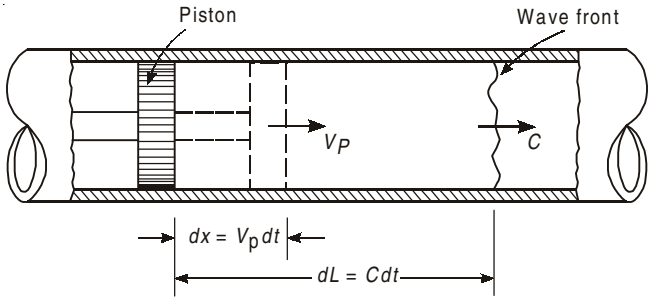


Figure 19.1 *A simplified model for one dimensional elastic or pressure wave propagation*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

But $dL = Cdt$ and $dx = V_p dt$, the above equation becomes

$$\rho C dt = (\rho + d\rho)(C - V_p)dt$$

or $$0 = Cd\rho - \rho V_p - d\rho V_p$$

Since the velocity V_p of the piston is much smaller than the elastic wave velocity C, the term $(d\rho\, V_p)$ may be neglected and hence

$$C(d\rho) = \rho V_p \text{; or } C = \frac{\rho V_p}{d\rho} \qquad ...(19.14)$$

Further in the region of compressed fluid, the fluid particles, have acquired velocity which is apparently equal to the velocity of the piston, accompanied by an increase in pressure dp due to the sudden motion of the piston. Thus applying the impulse-momentum equation for the fluid in the compressed region during dt, we get

$$dpAdt = \rho dLA(V_p - 0)$$

which may be simplified as

$$dp = \rho \frac{dL}{dt} \times V_p = \rho C V_p$$

since $$dL = C\,dt$$

Thus $$C = \frac{dp}{\rho V_p} \qquad ...(19.15)$$

By combining Eqs 19.14 and 19.15

$$C^2 = \frac{dp}{d\rho} \text{; or } C = \sqrt{\frac{dp}{d\rho}} \qquad ...(19.16)$$

Thus the velocity of propagation of the elastic or pressure wave in a compressible fluid is given by Eq. 19.16.

In Chapter 1 the bulk modulus of elasticity of fluid is defined as

$$K = -\frac{dp}{dv/v} \qquad ...(1.5)$$

Since volume $v \propto \frac{l}{\rho}$, or $v\rho$ = constant.

Differentiating both sides, we get

$$vd\rho + \rho dv = 0$$

$$\therefore \quad \frac{dv}{v} = -\frac{d\rho}{\rho} \qquad ...(19.17)$$

Combining Eqs 1.5 and 19.17, we have

$$K = \frac{dp}{d\rho/\rho} \qquad ...(19.18)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From Eqs 19.16 and 19.18 the expression for C becomes

$$C = \sqrt{\frac{K}{\rho}} \qquad ...(19.19)$$

Equation 19.19 shows that the velocity of the elastic or pressure wave depends on the bulk modulus of elasticity and the density of the fluid in which it is propagated. Furthermore the velocity of sound (which is also composed of elastic waves of this nature) in any fluid medium is also equal to $\sqrt{K/\rho}$, it may be mentioned that the elastic wave velocity C is same as the velocity of sound or sonic velocity in that fluid.

For isothermal process from Eq. 19.1

$$\frac{p}{\rho} = \text{constant}$$

or
$$\frac{dp}{\rho} - \frac{p d\rho}{\rho^2} = 0; \frac{dp}{d\rho} = \frac{p}{\rho} \qquad ...(19.20)$$

Thus from Eqs 19.18 and 19.20 for isothermal process, $K = p$ and hence from Eq. 19.19

$$C = \sqrt{(p/\rho)} = \sqrt{RT} \qquad ...(19.21)$$

since for perfect gas from the equation of state $p = \rho RT$.

Similarly for isentropic process from Eq. 19.2

$$\frac{p}{\rho^k} = \text{constant}$$

or
$$\frac{dp}{\rho^k} - \frac{kp d\rho}{\rho^{k+1}} = 0$$

or
$$\frac{dp}{d\rho} = \frac{kp}{\rho} \qquad ...(19.22)$$

Thus from Eqs 19.18 and 19.22, $K = kp$ and hence from Eq. 19.19

$$C = \sqrt{(kp/\rho)} = \sqrt{kRT} \qquad ...(19.23)$$

19.7 MACH NUMBER AND ITS SIGNIFICANCE

The *Mach Number Ma* (or N_M) at a point in the flow is defined as the ratio of velocity V of flow at the point to the velocity of sound or sonic velocity C in that fluid medium corresponding to the values of K and ρ existing at the point where V is measured. That is

$$Ma = \frac{V}{C} \qquad ...(19.24)$$

The Mach number is a dimensionless parameter which is the square root of the ratio of inertia force of the flow to the elastic force of fluid. Therefore for two geometrically similar compressible flow systems

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the dynamic similarity can be obtained when the ratio V/C or Ma is a constant for corresponding points in the two systems. Further it may be mentioned that Mach number is important in those problems in which the velocity of flow is comparable with the velocity of sound or sonic velocity. It may happen in the case of airplanes travelling at very high speed, bullets, projectiles etc. However, if for any flow system the Mach number is less than about 0.4 the effect of compressibility may be neglected for that flow system. Depending on the value of the Mach number, the flow can be classified as follows:

Subsonic Flow : $Ma < 1$; *Sonic Flow*: $Ma = 1$

Supersonic Flow : $1 < Ma < 3$; *Hypersonic Flow*: $Ma > 4$

When the Mach number in flow region is slightly less to slightly greater than unity, the flow is termed as *transonic flow*.

19.8 PROPAGATION OF ELASTIC WAVES DUE TO DISTURBANCE IN FLUID

Elastic or pressure waves are also generated when some disturbance is created in a compressible fluid. Such disturbance may be created when an object moves in a relatively stationary compressible fluid or when a compressible fluid flows past a stationary object, resulting in the generation of the elastic or pressure waves, which are transmitted radially in all directions with a velocity equal to that of sound in that fluid medium. In order to study the pattern of such waves, consider a small object, such as a tiny projectile, moving in a straight line with velocity V through a compressible fluid which is stationary. Let at time $t = 0$ the object be at A, then in time t it will move through a distance AB equal to Vt. During this time the wave which originated from the object when it was at A will grow into the surface of sphere of radius Ct as shown in Fig. 19.2, which also shows the growth of the other waves which will originate from the object at every $t/4$ interval of time as the object moves from A to B. Now depending upon the magnitude of velocity V relative to the velocity of sound C, different patterns of waves will be developed for the following three cases:

(i) ***V* less than *C*.** In this case since $V < C$ the elastic waves will travel in the forward direction ahead of the object, and hence as shown in Fig. 19.2 a, the object at point B is inside the sphere of radius Ct and also inside the other spheres formed by the waves started at intermediate times.

(ii) ***V* equal to *C*.** In this case both the object and the waves originating from it will reach point B at the same instant of time. As such as shown in Fig. 19.2 b, the object at point B is on the periphery of the sphere of radius Ct and also that of the other spheres formed by the waves started at intermediate times.

(iii) ***V* greater than *C*.** In this case the object travels faster than the waves. Thus the distance AB which the object has travelled is more than Ct, and hence as shown in Fig. 19.2 c the object at point B is outside the spheres formed due to the formation and growth of the elastic waves at time $t = 0$ and at the intermediate times. If from point B common tangents such as BT are drawn to all the spheres formed by the waves, then a cone will be generated, having its axis in the direction of motion and its vertex at B, the instantaneous position of the object. In this case the cone therefore advances with the object in the undisturbed fluid. This cone is known as *Mach cone*. The semi-vertex angle α of the cone is known as *Mach angle* which is given by

$$\sin \alpha = \frac{Ct}{Vt} = \frac{C}{V} = \frac{1}{Ma}$$

where Ma represents the Mach number (V/C) as indicated in the preceding section. The limiting value of angle α corresponds to $V = C$ or $Ma = 1$ which is shown in Fig. 19.2 b. The line such as BT is known

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

as *Mach line*. It is obvious that in this case outside the cone the fluid is completely unaffected by the waves, and hence the region outside the cone is often known as *zone of silence*. The region inside the cone is termed as *zone of action or region of influence*. Across the surface of the Mach cone there is an abrupt change of pressure and density.

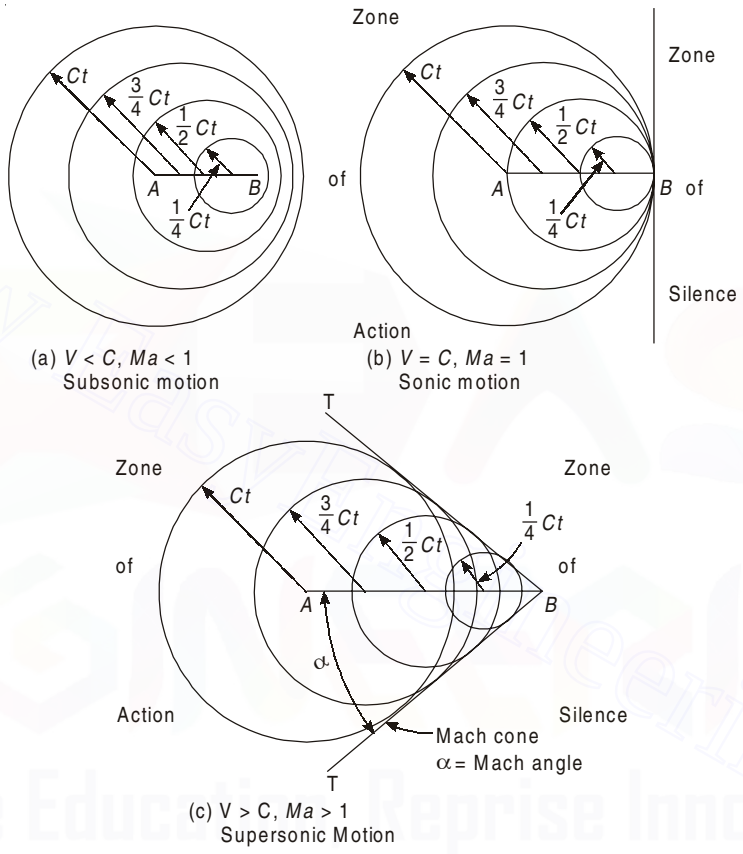


Figure 19.2 *Propagation of elastic or pressure waves by an object moving in a compressible fluid*

It may however be stated that although in the above discussion the object has been assumed to be very small such that it may be considered as a mere point and the resulting waves are also very small, yet the behaviour of the actual larger objects, such as aircraft, travelling at supersonic velocities is also very much similar to that discussed here, with the difference that in the case of larger objects the pressure waves developed are not very small and the same are propagated with a velocity greater than *C*. Further the Mach cone of an actual object travelling at supersonic velocity has a rounded apex.

19.9 STAGNATION PRESSURE IN COMPRESSIBLE FLOWS

Stagnation point is defined as a point in a fluid stream where the velocity of flow is reduced to zero and the kinetic energy is converted into pressure energy. The pressure at stagnation point is known as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

stagnation pressure. The foregoing principles can be applied to compute the stagnation pressure in a compressible flow. Thus consider the flow of a compressible fluid past an immersed object under frictionless adiabatic (or isentropic) conditions. Let V_1, p_1 and ρ_1 be the velocity, pressure and mass density of the approaching flow and V_2, p_2 and ρ_2 be the corresponding quantities at the point of stagnation. However, at the point of stagnation, the velocity head is converted into pressure head and therefore $V_2 = 0$. Thus from Eq. 19.13 if $(z_1 - z_2)$ is neglected, we have

$$\frac{p_1}{w_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right] = -\frac{V_1^2}{2g}$$

or
$$\frac{p_2}{p_1} = \left[1+\frac{k-1}{k}\left(\frac{V_1^2}{2g}\right)\frac{w_1}{p_1}\right]^{\frac{k}{k-1}}$$

or
$$\frac{p_2}{p_1} = \left[1+\frac{k-1}{2}\left(\frac{V_1^2}{kp_1/\rho_1}\right)\right]^{\frac{k}{k-1}} \quad ...(19.25)$$

since $w_1 = \rho_1 g$.

Expanding the above expression by binomial theorem, we have

$$\frac{p_2}{p_1} = 1 + \frac{k}{2}\left(\frac{V_1}{C_1}\right)^2 + \frac{k}{8}\left(\frac{V_1}{C_1}\right)^4 + \frac{k(2-k)}{48}\times\left(\frac{V_1}{C_1}\right)^6 + .. \quad ...(19.26)$$

where $C_1^2 = (kp_1/\rho_1)$ from Eq. 19.23

Rearrangement of Eq. 19.26 gives the pressure coefficient (also defined as ratio of pressure to dynamic pressure) or the dimensionless stagnation pressure as

$$\frac{p_2 - p_1}{\frac{\rho_1 V_1^2}{2}} = 1 + \frac{1}{4}\left(\frac{V_1}{C_1}\right)^2 + \frac{2-k}{24}\left(\frac{V_1}{C_1}\right)^4 + \ldots \quad ...(19.27)$$

It may be seen from Eq. 19.27 that when the approach velocity V_1 is small compared with the velocity of sound then it reduces to

$$\frac{p_2 - p_1}{\frac{\rho_1 V_1^2}{2}} = 1 \quad ...(19.28)$$

which is same as for an incompressible fluid. Further the ratio (V_1/C_1) is the Mach number Ma and hence Eq. 19.27 may also be expressed as

$$\frac{p_2 - p_1}{\frac{\rho_1 V_1^2}{2}} = 1 + \frac{1}{4}Ma^2 + \frac{2-k}{24}Ma^4 + \ldots \quad ...(19.29)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

A comparison of Eq. 19.27 and 19.28 show that as the approach velocity becomes larger, the stagnation pressure becomes greater for compressible fluids with isentropic flow than for incompressible fluids. Thus, for example, when the approach velocity V_1 is half the speed of sound or $Ma = 0.5$, the stagnation pressure is about 6 per cent greater and when the approach velocity approaches the sonic velocity or $Ma = 1$, the stagnation pressure is about 28 per cent greater.

19.10 FLOW OF COMPRESSIBLE FLUID WITH NEGLIGIBLE FRICTION THROUGH A PIPE OF VARYING CROSS–SECTION

When an incompressible fluid flows through a pipe of varying cross-section then the velocity simply varies inversely with the area, since in this case $Q = A_1V_1 = A_2V_2$. But when a compressible fluid flows through a pipe of varying cross-section the variation of velocity is affected by area as well as the density, since in this case according to continuity equation $\rho_1 A_1 V_1 = \rho_2 A_2 V_2$. The effect of the variation of the area of the flow passage on the velocity of flow of a compressible fluid may also be analysed by applying the foregoing principles as dislussed below

For an incompressible flow the continuity equation may be expressed as

$$AV = \text{constant}$$

which can be differentiated to give

$$VdA + AdV = 0; \text{ or } \frac{dA}{A} = -\frac{dV}{V} \qquad ...(19.30)$$

Similarly for compressible flow the continuity equation may be expressed as

$$\rho AV = \text{constant}$$

which can be differentiated to give

$$AVd\rho + \rho VdA + \rho AdV = 0$$

or
$$\frac{dA}{A} = -\frac{dV}{V} - \frac{d\rho}{\rho} \qquad ... (19.31)$$

For an isentropic flow from energy Eq. 19.12 neglecting the elevation difference term $(z_2 - z_1)$, it can be seen that

$$\frac{V^2}{2g} + \frac{k}{k-1}\left(\frac{p}{w}\right) = \text{constant}$$

or
$$\frac{V^2}{2} + \frac{k}{k-1}\left(\frac{p}{\rho}\right) = \text{constant} \qquad ...(19.32)$$

since the acceleration due to gravity g may be treated as constant.

Equation 19.32 can also be differentiated to give

$$VdV + \frac{k}{k-1}\left(\frac{dp}{\rho} - \frac{pd\rho}{\rho^2}\right) = 0 \qquad ...(19.33)$$

Further for an isentropic flow from Eqs 19.22 and 19.23

$$\frac{dp}{d\rho} = \frac{kp}{\rho}; \text{ or } \frac{dp}{\rho} = \frac{kpd\rho}{\rho^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $$C = \sqrt{\frac{kp}{\rho}}$$

Introducing the above expressions in Eq. 19.33, we have

$$-VdV = \frac{kp}{\rho}\frac{d\rho}{\rho} = C^2\frac{d\rho}{\rho}$$

or $$-\frac{V^2}{C^2}\frac{dV}{V} = \frac{d\rho}{\rho} \qquad ...(19.34)$$

Adding Eqs 19.31 and 19.34, we have

$$\frac{dA}{A} = \left[\frac{V^2}{C^2} - 1\right]\frac{dV}{V} = \left[Ma^2 - 1\right]\frac{dV}{V} \qquad ...(19.35)$$

This important equation is due to Hugoniot. Equations 19.30 and 19.35 give variation of (dA/A) for the flow of incompressible and compressible fluids respectively. The ratios (dA/A) and (dV/V) are respectively the fractional variations in the values of area and flow velocity in the flow passage.

Further in order to study the variation of pressure with the change in flow area, an expression similar to Eq. 19.35 may be obtained for the change of pressure dp as indicated below.

Since $$\frac{dp}{\rho} = \frac{kpd\rho}{\rho^2}$$

Introducing the above expression in Eq. 19.33 and simplifying it, the following expression may be obtained

$$\frac{dp}{\rho} = -VdV; \text{ or } \frac{dp}{\rho V^2} = -\frac{dV}{V}$$

Substituting for (dV/V) from Eq. 19.35, we have

$$dp = (\rho V^2)\frac{1}{1-Ma^2}\frac{dA}{A} \qquad ...(19.35\ a)$$

From Eqs 19.35 and 19.35 (a) it is possible to formulate the following conclusions of practical significance:

1. For *subsonic flows* (i.e., $Ma < 1$)

$$\frac{dV}{V} > 0;\ \frac{dA}{A} < 0;\ dp < 0 \text{ (convergent nozzle)}$$

$$\frac{dV}{V} < 0;\ \frac{dA}{A} > 0;\ dp > 0 \text{ (divergent diffuser)}$$

2. For *supersonic flows* (i.e., $Ma > 1$)

$$\frac{dV}{V} > 0;\ \frac{dA}{A} > 0;\ dp < 0 \text{ (divergent nozzle)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{dV}{V} < 0;\ \frac{dA}{A} < 0;\ dp > 0 \text{ (convergent diffuser)}$$

3. For *sonic flow* (i.e., $Ma = 1$)

$$\frac{dA}{A} = 0 \text{ (straight flow passage since } dA \text{ must be zero)}$$

and dp = (zero/zero) i.e., indeterminate, but when evaluated, the change of pressure $dp = 0$,

since dA = 0 and the flow is frictionless.

Thus the effect of area variation on subsonic flow of compressible fluids is same as that for the flow of incompressible fluids. But the effect of area variation on supersonic flow of compressible fluids is exactly opposite to that for subsonic flow of compressible fluids. In order to accelerate the flow of compressible fluids at supersonic velocity the flow passage must diverge. The divergence of the flow passage is necessary because in supersonic flow of compressible fluid the decrease in the density ρ of the fluid is greater than the increase in the flow velocity, and the area of flow passage must increase for the mass flow rate (ρVA) to remain constant.

When $V = C$ so that $Ma = 1$, dA must be zero and since the second derivative of A is positive, A must be minimum. It thus means that if the velocity of flow equals the sonic velocity anywhere it must therefore do so where the cross-section is of minimum area. However, dA could also be zero when $dV = 0$. As such at a position of minimum cross-section the velocity is either equal to the sonic velocity or it is a maximum velocity for subsonic flow or it is a minimum velocity for supersonic flow.

From the foregoing discussion it may be seen that in order to convert an initially subsonic flow of a compressible fluid in a nozzle into a supersonic flow, it is necessary to have a convergent passage followed by a divergent one. But this conversion will take place only if at the throat where the area is minimum the velocity of flow is equal to sonic velocity. Otherwise the divergent passage will have only the subsonic flow of compressible fluid and then it will simply function as a diffuser where the flow is decelerated.

19.11 FLOW OF COMPRESSIBLE FLUID IN CONVERGENT—DIVERGENT PASSAGES

The foregoing discussion may be applied to study the flow of compressible fluid through a convergent-divergent nozzle. Consider a convergent passage or nozzle as shown in Fig. 19.3 (a), through which a compressible fluid is flowing. Let the initial velocity V_1 of the flow in the approach section be small in comparison with the exit velocity V_2 at the throat and let the flow accelerate subsonically in the convergent passage. Thus for reversible adiabatic process with $V_1 = 0$, from energy Eq. 19.13, we obtain

$$\frac{V_2^2}{2g} = \frac{p_1}{w_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{(k-1)/k}\right] \qquad ...(19.36)$$

Since this is an adiabatic process for which from Eq. 19.2

$$\frac{p_1}{w_1^k} = \frac{p_2}{w_2^k}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence Eq. 19.36 can be written as

$$\frac{V_2^2}{2g} = \frac{p_2}{w_2}\frac{k}{k-1}\left[\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right]$$

or
$$\frac{V_2}{\sqrt{(kp_2/\rho_2)}} = \left[\frac{2}{k-1}\left\{\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right\}\right]^{1/2}$$

or
$$\frac{V_2}{C_2} = \left[\frac{2}{k-1}\left\{\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right\}\right]^{1/2} \quad ...(19.37)$$

in which $C_2 = \sqrt{(kp_2/\rho_2)}$ is the speed of sound in the jet. From Eq. 19.37 it can be seen that the velocity of the jet V_2 relative to the speed of sound C_2 is dependent only upon the pressure ratio (p_1/p_2).

From continuity equation the mass rate of flow G at the throat is given as

$$G = \rho_2 A_2 V_2$$

which by combining Eqs. 19.36 and 19.2 may be expressed as

$$\frac{G}{A_2} = \rho_2 V_2 = \sqrt{\frac{2k}{k-1}(p_1\rho_1)\left\{\left(\frac{p_2}{p_1}\right)^{2/k} - \left(\frac{p_2}{p_1}\right)^{(k+1)/k}\right\}} \quad ...(19.38)$$

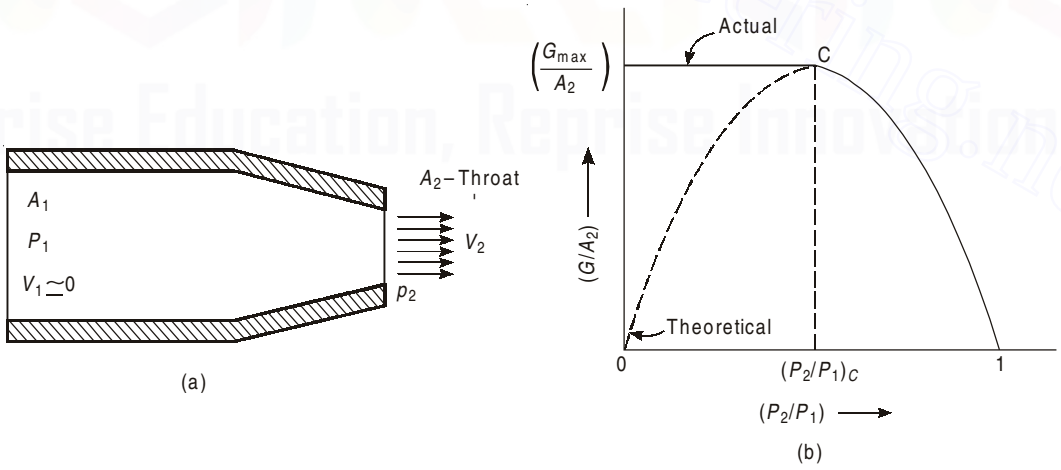


Figure 19.3 (*a*) *Flow of compressible fluid in a convergent nozzle;*
(*b*) *Variation of mass rate of flow of compressible fluid* (G/A_2) *with pressure ratio* (p_2/p_1)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 19.38 represents the mass rate of flow per unit area (G/A_2) at the throat, from which it may be seen that for any given nozzle with known conditions at the approach section 1 the mass rate of flow depends on the pressure ratio (p_2/p_1). A plot of (G/A_2) versus (p_2/p_1) is shown in Fig. 19.3 (b). It can be seen from the plot that (G/A_2) = 0 for (p_2/p_1) = 0 and (p_2/p_1) = 1; and as the throat pressure p_2 is reduced from $p_2 = p_1$ (i.e., $p_2/p_1 < 1$) the mass flow increases (as shown by the solid curve) until it reaches a maximum at a certain value of the pressure ratio. The pressure ratio at which the mass flow is a maximum is called the *critical pressure ratio*. The critical pressure ratio $(p_2/p_1)_c$ can be determined by differentiating both sides of Eq. 19.38 with respect to (p_2/p_1) and equating the resulting expression to zero. Thus, we obtain

$$\frac{2}{k}\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}-1} - \left(\frac{k+1}{k}\right)\left(\frac{p_2}{p_1}\right)^{\frac{k+1}{k}-1} = 0$$

or
$$\left(\frac{p_2}{p_1}\right)_c = \left(\frac{2}{k+1}\right)^{\frac{k}{k-1}} \qquad ...(19.39)$$

Equation 19.39 shows that the critical pressure ratio depends only upon the adiabatic exponent k. For air at normal conditions since k = 1.40, the critical pressure ratio has the value $(p_2/p_1)_c$ = 0.528. By substituting the value of (p_2/p_1) from Eq. 19.39 in Eq. 19.38 the maximum mass rate of flow is found to be

$$G_{max} = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}} \qquad ...(19.40)$$

It can be seen from Eq. 19.40 that the maximum mass flow rate of a compressible fluid through a convergent nozzle depends only upon the pressure p_1 and mass density ρ_1 at the approach section 1.

Again from Eq.19.38 it is seen that with further decrease in the throat pressure below the critical value [i.e., $p_2/p_1 < (p_2/p_1)_c$] the mass flow should decrease as shown by the dotted portion of the curved line in Fig. 19.2 (b). However, this decrease in the mass flow rate is only theoretically true, because actually for any given value of p_1, for pressure ratios lower than the critical pressure ratio, the mass flow rate does not decrease but it remains constant at the maximum value for all the values of (p_2/p_1) less than the critical pressure ratio. This may be explained as follows.

By introducing Eq. 19.39 in Eq. 19.37 the value of the Mach number at the throat corresponding to the critical pressure ratio is obtained as

$$[(Ma)_2]_c = \frac{V_2}{C_2} = \left[\frac{2}{k-1}\left(\frac{k+1}{2}-1\right)\right]^{1/2} = 1$$

That is, at the critical pressure ratio the velocity V_2 at the throat is equal to the speed of the sound. As explained in Sec. 19.10 for an accelerating flow of a compressible fluid in a convergent nozzle the velocity of flow within the nozzle is subsonic with a maximum velocity equal to the sonic velocity at the throat. Thus when once the velocity V_2 at the throat has reached the speed of sound at the critical pressure ratio, it remains at the same value for all the values of (p_2/p_1) less than critical pressure ratio, since the flow in the nozzle is being continuously accelerated with the reduction in the throat pressure below the critical values and hence the velocity cannot reduce. Therefore, for any given value of p_1, the mass flow rate for all values of (p_2/p_1) less than critical pressure ratio, remains constant at the maximum

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

value as indicated by the solid horizontal line in Fig. 19.3 (b). This fact has been verified experimentally.

Now if a properly designed divergent section is attached to the throat of the convergent section as shown in Fig. 19.4, then as discussed in Sec. 19.10, the flow of the compressible fluid can be accelerated to supersonic velocities in the divergent section when the throat pressure p_2 corresponds to the critical value given by Eq. 19.39. The application of these principles is found in the *de Laval nozzle*, developed by the Swedish engineer Carl Gustaf Patrik de Laval (1845–1913) for accelerating steam to supersonic velocity in his high speed steam turbine. Similar nozzles are used nowadays in jet propulsion units. The nozzle consists of converging and diverging sections (Fig. 19.4) so proportioned that at the throat

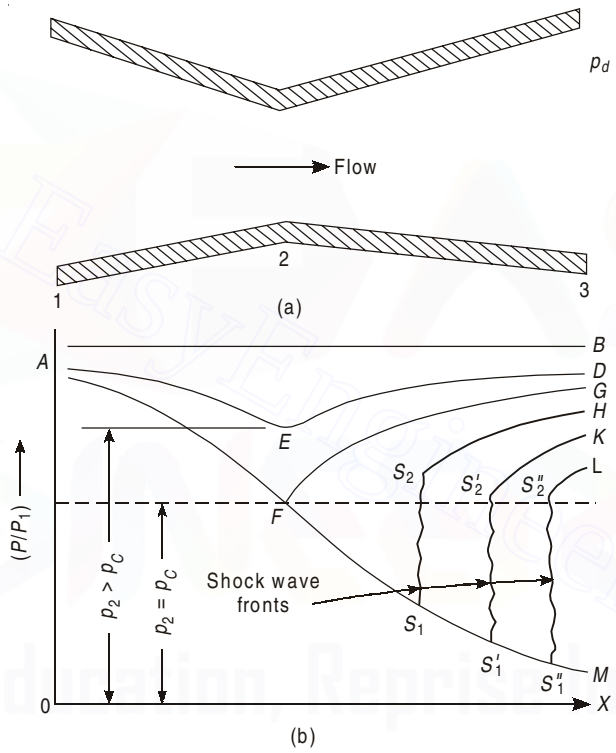


Figure 19.4 (*a*) *Convergent–divergent nozzle* (*b*) *Pressure distribution through a convergent– divergent nozzle with flow of compressible fluid*

section critical pressure and critical velocity (equal to sonic velocity) occurs. The various possible flow phenomena for a compressible fluid flowing in such a convergent-divergent passage may be analysed as follows.

Consider a compressible fluid flowing into a convergent–divergent nozzle from a large reservoir in which the approach velocity is negligible. The pressure in the reservoir may thus be assumed to be constant being equal to the pressure p_1 at the inlet section of the nozzle. It is, however, assumed that the pressure p_d at the downstream of the nozzle may be varied. Now if the pressure p_d is equal to the pressure p_1, then there can be no flow and the pressure throughout the nozzle, as well as downstream is same (i.e., $p/p_1 = 1.0$) as represented by the horizontal line AB in Fig. 19.4 (b). As the downstream

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

pressure p_d is gradually reduced, the pressure p_3 at section 3 of the nozzle is likewise-reduced, but since the velocity has nowhere yet reached the sonic velocity, the influence of this reduction of pressure can move upstream, thereby causing the flow to take place with a greater drop in pressure at the throat section 2 of the nozzle. The pressure distribution as represented by the curve *AED* of Fig. 19.4 (b) is then obtained. This is seen to be similar in shape to that for the flow of a liquid in venturimeter. The mass rate of flow which depends on the values of pressure ratio (p_3/p_1) is given by Eq. 19.38 with A_3 and p_3 replacing A_2 and p_2 respectively. Thus

$$\frac{G}{A_3} = \left(\frac{2k}{k-1} p_1 \rho_1\right)^{\frac{1}{2}} \left\{\left(\frac{p_3}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_3}{p_1}\right)^{\frac{k+1}{k}}\right\}^{\frac{1}{2}} \quad \ldots(19.41)$$

When the downstream pressure p_d is further reduced to such a value as shown at *G* the pressure at the throat section 2 becomes equal to the critical pressure and the velocity of flow at the throat section is just equal to the sonic velocity. The pressure distribution is then represented by the curve *AFG* of Fig. 19.4 (b), which indicates that the flow is being accelerated in the convergent section upto the throat and it decelerates in the divergent section of the nozzle. This is due to the fact that immediately after the throat section the flow reverts to subsonic conditions in the divergent section which then acts as a diffuser. Under such conditions the nozzle is said to be *choked*. The mass rate of flow is then given by Eq. 19.40.

If the downstream pressure p_d is further reduced, pressure or shock waves moving in the upstream direction are developed in the divergent section, which affect the flow conditions there and hence new flow conditions are developed in the divergent section. But the flow conditions in the convergent part of the nozzle remain unchanged, the throat pressure remains at the critical value and the upstream pressure distribution follows the single curve *AF*.

Since the pressure or shock waves moving upstream in the divergent section encounter the sonic flow at the throat and hence these waves, which are also moving at the speed of sound, cannot penetrate into the flow upstream of the throat, thereby not affecting the flow in the convergent portion of the nozzle. However, the formation of the shock wave front as well as its location in the divergent portion of the nozzle depends on the magnitude of the downstream pressure p_d. Thus if the downstream pressure p_d corresponds to a value as shown at *H* such that $(p_d)_H > p_c$ then the pressure distribution follows the curve AFS_1S_2H in which S_1S_2 represents a shock wave front formed in the divergent section. This formation of the shock wave front may be explained as follows.

The downstream pressure p_d being more than the critical pressure, subsonic flow conditions prevail in most of the downstream portion of the divergent section of the nozzle. However, the flow just downstream from the throat is accelerated to become slightly supersonic in the upstream portion of the divergent section, and the change of the approaching supersonic flow to the subsonic flow takes place through the formation of a shock wave front in the divergent section. Across the shock the velocity changes abruptly from supersonic to subsonic. There is also a finite pressure difference across the shock. The position of the shock wave front is fixed by the physical condition that the momentum difference across the shock is just balanced by the pressure difference between the supersonic flow on the upstream of the shock and the subsequent subsonic flow on the downstream of the shock. With further reduction in the downstream pressure p_d the shock wave front gets shifted and it will be established further downstream in the divergent section as indicated successively by the curves $AFS_1'S_2'K$ and $AFS_1''S_2''L$ of Fig. 19.4 (b). Now if the downstream pressure p_d is reduced to such a value

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

as that at M, then the pressure distribution would follow the curve AFM. The flow thus accelerates steadily throughout the entire passage, being subsonic in the convergent section, sonic at the throat and supersonic in the subsequent divergent section. In this case the gas expands isentropically to critical pressure p_c at the throat and continues its isentropic expansion along the curve FM. As such in this case there is no shock wave front developed in the divergent section. The mass rate of flow of fluid may therefore be computed either by Eq. 19.40 or by Eq. 19.41. By equating the mass rate of flow given by Eqs. 19.40 and 19.41, the ratio of the throat area A_2, and the exit area A_3. for a convergent-divergent nozzle having isentropic supersonic flow without shock in the divergent section, may be obtained as

$$\frac{A_2}{A_3} = \left(\frac{k+1}{2}\right)^{\frac{k+1}{2(k-1)}} \left(\frac{2}{k-1}\right)^{1/2} \left[\left(\frac{p_3}{p_1}\right)^{\frac{1}{k}} - \left(\frac{p_3}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2} \quad \ldots(19.42)$$

It may be seen from Eq. 19.42 that for a convergent-divergent nozzle required to develop an isentropic supersonic flow without shock in the divergent section, the ratio of the throat area A_2 and the exit area A_3 depends on the pressure ratio (p_3/p_1). For a convergent-divergent nozzle designed to produce a supersonic flow such that smooth expansion (along the curve AFM) takes place, the pressure ratio $p_1/(p_d)_M$ is known as the *design pressure ratio* of the nozzle.

A further reduction of the downstream pressure p_d below $(p_d)_M$ (the value corresponding to point M) will however have no effect on the flow in the nozzle. As such the flow conditions throughout the nozzle will remain exactly the same as those corresponding to the downstream pressure equal to $(p_d)_M$.

19.12 NORMAL SHOCK WAVES

It has been indicated in the foregoing section that shock waves may occur in the diverging section of a convergent-divergent nozzle when the approaching supersonic flow is converted to a subsonic flow. Such shock waves may also occur in pipes as well as in front of a blunt-nosed body, when an approaching supersonic flow is converted to subsonic flow. The flow on upstream of the shock is supersonic having a relatively lower pressure (being less than the critical pressure p_c), while on the downstream of the shock it is subsonic having a higher pressure (being more than the critical pressure p_c). It therefore means that the conversion of supersonic flow to a subsonic flow is always accompanied by a sudden pressure rise as the flow passes the shock. Although in practice a shock is not absolutely abrupt, but the distance over which it occurs being very small, for most purposes it may be supposed to be abrupt. Accordingly across a shock wave abrupt and discontinuous (i.e., sudden and finite) changes of flow properties such as pressure, density, velocity etc., may be supposed to occur. The process of the formation of the shock waves in a compressible fluid is analogous to the formation of 'hydraulic jump' in open channels. In case of hydraulic jump the flow changes from supercritical to subcritical and is accompanied by an increase in depth and decrease in the velocity of flow. In shock waves the critical velocity being the velocity of sound (or sonic velocity), the flow changes from supersonic to subsonic and it is accompanied by an increase in the pressure and density and a decrease in the velocity.

The shock waves can be either normal or oblique with respect to the direction of flow. However, only the normal shock waves have been discussed here. As indicated below the various elements of a normal shock wave can be determined by the application of continuity, momentum and energy equations.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Consider a stationary normal shock wave developed in a pipe as shown in Fig. 19.5. Let ρ_u, V_u and ρ_d, V_d represent the mass density and velocity upstream and downstream of the shock.

Thus according to continuity equation,

$$\rho_u V_u = \rho_d V_d = \frac{G}{A} \qquad ...(19.43)$$

where G/A is the mass rate of flow per unit area.

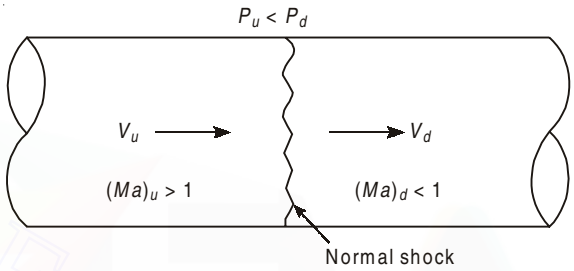


Figure 19.5 *Stationary normal shock wave in a pipe*

Further if p_u and p_d are the pressure intensities upstream and downstream of the shock and if the effects of boundary friction are negligible then applying the momentum equation across the shock, we have

$$(p_u - p_d) A = (\text{Mass flow}) (V_d - V_u)$$

i.e.
$$\left.\begin{aligned} p_u - p_d &= \rho_d V_d^2 - \rho_u V_u^2 \\ p_u + \rho_u V_u^2 &= p_d + \rho_d V_d^2 \end{aligned}\right\} \qquad ...(19.44)$$

or

If it is assumed that there is no net heat transfer to or from the pipe, then from Eq. 19.12 the energy equation for the flow across the shock may be obtained as

$$\frac{k}{k-1}\left(\frac{p_d}{\rho_d} - \frac{p_u}{\rho_u}\right) = \frac{V_u^2}{2} - \frac{V_d^2}{2} \qquad ...(19.45)$$

By combining continuity Eq. 19.43 and momentum Eq. 19.44, we get

$$p_u + \frac{(G/A)^2}{\rho_u} = p_d + \frac{(G/A)^2}{\rho_d} \qquad ...(19.46)$$

Equation 19.46 is the *equation of the Rayleigh line*.

Similarly by combining the continuity Eq. 19.43 and the energy Eq. 19.45, we get

$$\frac{(G/A)^2}{2\rho_u^2} + \frac{k}{k-1}\frac{p_u}{\rho_u} = \frac{(G/A)^2}{2\rho_d^2} + \frac{k}{k-1}\frac{p_d}{\rho_d} \qquad ...(19.47)$$

Equation 19.47 is the *equation of the Fanno line*.

Further elimination of (G/A) from Eqs. 19.46 and 19.47 results in the following relationship between the pressure and densities of flow on both sides of a normal shock

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_d}{p_u} = \frac{\left(\frac{k+1}{k-1}\right)\left(\frac{\rho_d}{\rho_u}\right)-1}{\left(\frac{k+1}{k-1}\right)-\frac{\rho_d}{\rho_u}} \qquad ...(19.48)$$

or
$$\frac{\rho_d}{\rho_u} = \frac{V_u}{V_d} = \frac{1+\left(\frac{k+1}{k-1}\right)\frac{p_d}{p_u}}{\left(\frac{k+1}{k-1}\right)+\frac{p_d}{p_u}} \qquad ...(19.49)$$

Equations 19.48 and 19.49 are known as *Rankine–Hugoniot Equations* in honour of W. Rankine and H. Hugoniot who first developed these equations.

From Eqs 19.23 and 19.24 since sonic velocity $C = \sqrt{(pk/\rho)}$ and Mach number $Ma = (V/C)$, Eqs. 19.48 and 19.49 can be written in alternative form as

$$\frac{p_d}{p_u} = \frac{2k(Ma)_u^2 - (k-1)}{(k+1)} \qquad ...(19.50)$$

and
$$\frac{\rho_d}{\rho_u} = \frac{V_u}{V_d} = \frac{(k+1)(Ma)_u^2}{(k-1)(Ma)_u^2+2} \qquad ...(19.51)$$

where $(Ma)_u$ is the Mach number just upstream of the shock wave.

Further by combining Eqs 19.50 and 19.51 the following expression for $(Ma)_d^2$, may also be obtained

$$(Ma)_d^2 = \frac{(k-1)(Ma)_u^2+2}{2k(Ma)_u^2-(k-1)} \qquad ...(19.52)$$

It may however be noted that as the flow crosses a shock the energy of flow is conserved, but there is an increase in entropy in the flow process. Therefore the flow crossing a shock is an irreversible process.

19.13 MEASUREMENT OF COMPRESSIBLE FLUID FLOW

(a) Measurement of Discharge. For measuring the mass rate of flow of compressible fluids in pipe lines, venturimeter, orifice meter and nozzle meter are commonly used. In each of these meters the flow conditions resemble those in a convergent-divergent nozzle. Thus as the fluid passes through the throat or minimum section of the flow meter the fluid pressure is partly converted to kinetic energy, thereby developing a pressure difference between the inlet and the throat sections of the flow meter, which can be measured. The distance between inlet and throat sections is usually small so that friction and heat transfer between these sections can be neglected. Therefore as indicated below the expression for the mass rate of flow G through any of these flow meters can be obtained by applying the continuity equation, isentropic or polytropic process equation for pressure variation and the general energy equation for compressible fluid flow.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From the continuity equation for one dimensional flow

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2$$

Assuming the flow process in the flow meter to be polytropic, the relationship between the pressure and density may be expressed as

$$\frac{p_1}{\rho_1^n} = \frac{p_2}{\rho_2^n}$$

For a compressible fluid flow the general energy equation may be expressed as

$$\frac{1}{2}V_1^2 + \frac{n}{n-1}\frac{p_1}{\rho_1} = \frac{1}{2}V_2^2 + \frac{n}{n-1}\frac{p_2}{\rho_2}$$

However, for insentropic flow $n = k$ in the above expressions. By combining the above noted three expressions following expression for the rate of flow G may be readily obtained

$$G = CA_2\rho_2\left[\frac{\frac{2n}{n-1}\frac{p_1}{\rho_1}\left\{1-\left(\frac{p_2}{p_1}\right)^{\frac{n-1}{n}}\right\}}{1-\left(\frac{A_2}{A_1}\right)^2\left(\frac{P_2}{p_1}\right)^{2/n}}\right]^{\frac{1}{2}} \quad ...(19.53)$$

where C is the meter coefficient that takes care of the overall friction losses in the flow meter. Tests have shown that the numerical values of C for these meters are approximately the same as those of coefficient of discharge C_d (or K) for incompressible fluids flowing through them.

(b) Measurement of Velocity (*i*) *Pitot Tube*. For determining the velocity of flow of a compressible fluid *pitot tube* (as described in Chapter 7) is a commonly used device. When pitot tube is inserted in a stream of compressible fluid, its tip becomes a stagnation point since the velocity of flow there is reduced to zero and stagnation pressure is developed. By measuring the stagnation pressure p_s and the static pressure p_1 in the approaching stream, the velocity of flow can be determined as indicated below. The mathematical relationship between the stagnation pressure and the velocity of compressible flow depends on whether the flow is subsonic or supersonic.

If the flow is 'subsonic' it can be assumed to be isentropic and from Eq. 19.25, the ratio of the stagnation pressure p_s to the pressure p_1 in the approaching stream may be obtained as

$$\frac{p_s}{p_1} = \left[1+\frac{k-1}{2}\left(\frac{V_1^2}{kp_1/\rho_1}\right)\right]^{\frac{k}{k-1}}$$

or

$$\frac{p_s}{p_1} = \left[1+\frac{k-1}{2}(Ma_1)^2\right]^{\frac{k}{k-1}} \quad ...(19.54)$$

For small values of $(Ma_1)^2$, the right hand side of Eq. 19.54 may be expanded into a convergent series by means of the binomial theorem. Thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_s}{p_1} = 1 + \frac{1}{2}k\,(Ma_1)^2\left[1+\frac{1}{4}(Ma_1)^2+\frac{2-k}{24}(Ma_1)^4+\ldots\right]$$

The terms within the bracket in the above expression are usually referred to as the *compressibility factor* because the pressure ratio (p_s/p_1) for a flow of an incompressible fluid is

$$\frac{p_s}{p_1} = 1 + \frac{1}{2}\frac{\rho_1 V_1^2}{p_1} = 1 + \frac{1}{2}\;k\;\frac{V_1^2}{kp_1/\rho_1} = 1 + \frac{1}{2}\;k\,(Ma_1)^2$$

When a pitot tube is introduced into a 'supersonic' stream, a shock will occur upstream of pitot tube. If p_2 is the pressure on the downstream of the shock then across the shock the pressure ratio (p_2/p_1) is given by Eq. 19.50 as

$$\frac{p_2}{p_1} = \frac{2k(Ma_1)^2-(k-1)}{(k+1)}$$

The Mach number Ma_2 of the subsonic flow downstream of the shock is given by Eq. 19.52 as

$$Ma_2 = \left[\frac{(k-1)(Ma_1)^2+2}{2k(Ma_1)^2-(k-1)}\right]^{\frac{1}{2}}$$

Further the pressure ratio (p_s/p_2) may be expressed by Eq. 19.54 as

$$\frac{p_s}{p_2} = \left[1+\frac{k-1}{2}(Ma_2)^2\right]^{\frac{k}{k-1}}$$

By combining the above expressions the following expression for (p_s/p_1) may be readily obtained

$$\frac{p_s}{p_1} = \frac{\frac{p_s}{p_2}}{\frac{p_1}{p_2}} = \left[\frac{\left\{\frac{k+1}{2}(Ma_1)^2\right\}^k}{\frac{2k(Ma_1)^2-(k-1)}{k+1}}\right]^{\frac{1}{k-1}} \qquad \ldots(19.55)$$

Equation 19.55 is the wellknown *Rayleigh–Pitot equation* for stagnation pressure in a supersonic stream with a shock wave front.

(ii) Hot-wire Anemometer. A *hot-wire anemometer* is another instrument which is commonly used for measuring the velocity of flow of a compressible fluid such as air or gas. The anemometer consists of a platinum, nickel or tungsten wire of about 5×10^{-3} to 8×10^{-3} mm diameter and 6 mm length. The wire is mounted on the ends of two pointed prongs and it is introduced in the flow field so as to be perpendicular to the direction of flow. A small electric current is passed through the wire, in order to heat it above the temperature of the surrounding air or gas. As the air or gas flows round the wire, the wire is cooled. The amount of heat lost from the heated wire to the air or gas flow is a function of the flow velocity past the wire. The amount of heat transfer from the wire to the surrounding air or gas results in altering the temperature of the wire which in turn results in the variation of the electrical resistance since the electrical resistance of the wire is a function of its temperature. Thus a relationship between the heat transfer or the resistance of the wire and the flow velocity can be obtained by calibration.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Figure 19.6 illustrates two circuits for a hot-wire anemometer either of which may be used for measuring the velocity of flow. R_1, R_2 and R_3 are electric resistances in the Wheatstone–bridge circuit and the hot-wire forms the remaining resistance in the bridge. *B* represents a variable electric resistance, and *A* represents a battery. In the arrangement shown in Fig. 19.6 (a) constant current is passed through the wire by keeping the voltage across the bridge (i.e., across points *CD*) constant after the circuit is adjusted so that the galvanometer indicates zero current when the heated wire is in air or gas at rest. As the air or gas flows the hotwire cools, its resistance changes and the galvanometer deflects. The galvanometer deflection is correlated with the velocity of flow of air or gas by calibration. It is then termed as *constant current hot-wire anemometer.* Because of its high sensitivity the constant-current hot-wire anemometer is especially suitable for flow in which velocity fluctuations are small.

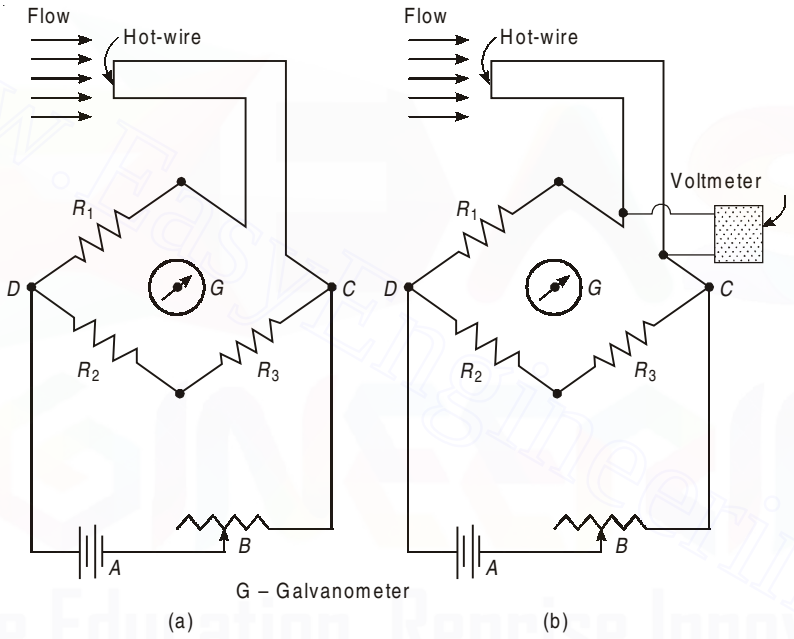


Figure 19.6 (*a*) *Constant-current hot-wire anemometer;* (*b*) *Constant-temperature hot-wire anemometer*

Figure 19.6 (b) illustrates another arrangement for hot wire anemometer which is termed as *constant temperature* (*or constant-resistance*) *hot-wire anemometer*. In this case the temperature of the hot-wire is held constant and hence its resistance remains constant. Initially when there is no flow and the hot-wire is in contact with air or gas at rest, a small current is passed through the hot-wire and the circuit is adjusted so that the galvanometer reads zero. As the air or gas flows past the hot-wire, its temperature and hence its resistance will vary, which will cause the galvanometer needle to deflect from zero reading. Now by adjusting the variable resistance *B* the current passing through the hot-wire is suitably adjusted so that its temperature and hence the resistance is maintained constant and the galvanometer reading is brought back to zero. The reading of the voltmeter connected across the hot-wire will then change which may be noted. By calibration in a stream of known velocity, the voltmeter reading is correlated with the velocity of flow of air or gas. The constant temperature hot-wire anemometer may be conveniently used for velocity measurement even when the velocity fluctuations are large.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

A hot-wire anemometer requires frequent calibration against a known velocity. The calibration should be carried out in the same fluid as that whose velocity is to be measured, because heat transfer depends on the fluid density also. Moreover it is a delicate device being subjected to breakage. However, despite these limitations it has widespread use in air or gas flow studies. Since it responds very rapidly to fluctuations of velocity it is widely used in conjunction with oscilloscopes and similar electronic instruments for measuring the intensity of turbulence.

A hot-wire anemometer is mainly used for measuring the velocity of flow of air or gas. It has not been possible to use it successfully with liquids because bubbles and some small solid particles present in the liquid tend to collect on the wire and spoil the calibration.

Recently a *hot film anemometer* has been developed, in which a thin platinum film fused to a glass support is used in place of the hot-wire. This instrument has a better mechanical strength but slightly inferior frequency response as compared with hot-wire anemometer.

(c) Measurement of Flow Direction. Often it is desired to determine both the magnitude as well as the direction of the velocity of flow. The pitot tube is a good instrument used for measuring the magnitude of the velocity of flow, but it cannot be used for determining the direction of the velocity. Moreover it has been observed that a pitot tube is very insensitive to its alignment with the direction of flow, as it has only about 1 per cent error at an angle of 20° to the direction of flow. As such certain other instruments have been developed which may be used to determine both the magnitude as well as the direction of the velocity of flow. These are (*i*) *Pitot cylinder* and (*ii*) *Pitot sphere* which are shown in Fig. 19.7.

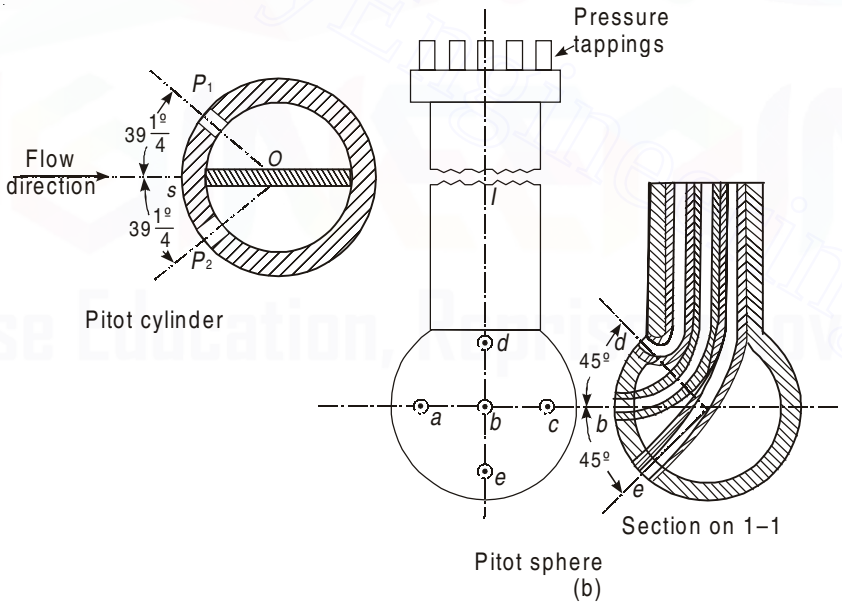


Figure 19.7 *(a) Pitot cylinder; (b) Pitot sphere*

A *Pitot cylinder* consists of a cylindrical tube, with two small pressure taps P_1 and P_2. The axis of the cylinder is perpendicular to the direction of flow. Each pressure tap is connected to one side of a differential pressure gage. For determining the direction of flow the cylinder is rotated about its axis in the stream, until the pressures at P_1 and P_2 are equal as indicated by a null reading on the differential

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

pressure gage. In this null position the stagnation point S is mid-way between P_1 and P_2 and therefore the bisector of the angle P_1OP_2 gives the direction of flow. If the pitot cylinder is to be used for the determination of the direction of flow only, then the angle P_1OP_2 may have any value. But if it is used for determining the magnitude of the velocity also, then for an ideal fluid each of the pressure taps P_1 and P_2 should be at an angle of 30° from S, so that in null position pressure recorded at P_1 or P_2 would be the static pressure of the undisturbed stream. However, for the fluids in actual practice this angle between P_1 or P_2 and S is to be about $39\frac{1}{4}°$. Thus if the angle P_1OP_2 is $78\frac{1}{2}°$, then in the null position the pressure reading from the tap P_1 or P_2 gives the static pressure. Now if the cylinder is rotated about its axis through $39\frac{1}{4}°$, then one of the pressure taps will be in line with the direction of flow. In this position the pressure reading at this tap gives the stagnation pressure. The difference between the stagnation and the static pressure heads is equal to the velocity head $\frac{V^2}{2g}$ (Sec. 7.14) from which the velocity of flow V can be computed. A Pitot cylinder is suitable for determination of both magnitude and direction of velocity in a two dimensional flow.

A *Pitot sphere* is another instrument which may be used to determine the magnitude and direction of velocity of flow in a three-dimensional flow. As shown in Fig. 19.7 (b) it consists of a small hollow sphere 8 to 15 mm in diameter having five pressure taps *a*, *b*, *c*, *d* and *e*. The three taps *a*, *b* and *c* are located on the horizontal great circle, each being inclined at 50° to the other. The taps *d*, *b* and *e* are located on the vertical great circle passing through tap *b* and each of these taps is inclined at 45° to the other. All the taps are connected to manometers. The sphere is fixed on the end of a hollow stem which can be rotated about its axis and the angle of rotation may be read off on fixed horizontal graduated disc by means of an indicating pointer attached to the stem. The stem can also be rotated in a vertical plane and the angle of tilt may be read off on a vertical graduated arc attached to the stem.

For measuring the velocity of flow the sphere is introduced in the flowing fluid with its stem perpendicular to the direction of the longitudinal axis of the flow passage. It is then so adjusted that the plane containing the taps *d*, *b* and *e*; and the zero of the horizontal graduated disc are parallel to the longitudinal axis of the flow passage. The stem is then rotated about its axis until the manometers connected to the taps *a* and *c* indicate the same reading. In this position, the velocity vector lies in the vertical plane passing through the taps *d*, *b* and *e*. The angle of rotation is obtained from the horizontal graduated disc. The stem is then tilted by rotating it in the vertical plane, until the manometers connected to the taps *d* and *e* indicate the same reading. In this position the velocity vector would be along the radius of the sphere passing through the central tap *b*. The angle of tilt is obtained from the vertical graduated arc. Thus the direction of the velocity vector is known. In order to determine the magnitude of the velocity, the pressure reading at the tap *b* is recorded when the instrument is in tilted position, which gives the stagnation pressure. By subtracting the static pressure head from the stagnation pressure head the velocity head can be known from which the velocity of flow can be computed.

Another type of Pitot sphere which is useful for the determination of the direction of flow only, has also been developed in which only three pressure taps are provided on the sphere which are situated at the vertices of an equilateral triangle. It is held in the flowing stream and so adjusted that the pressure at all the three taps are equal. The velocity of flow is then parallel to the radius of the sphere

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

passing through the centroid of the triangle.

Both the Pitot cylinder and the Pitot sphere should be properly calibrated in a stream of known velocity prior to these being used for the measurement of the velocity of flow.

ILLUSTRATIVE EXAMPLES

Example 19.1. *A test plane is described as having attained a flight speed of Ma = 2 at an attitude of 16 km where the temperature is approximately –56.5°C (or 216.65 K). Assuming k = 1.4 and R = 287 J/kg.K, determine the speed of the air plane.*

Solution

From Eq. 19.23 sonic velocity is given by

$$C = \sqrt{kRT}$$
$$= \sqrt{1.4 \times 287 \times 216.65}$$
$$= 295 \text{ m/s}$$

The speed of the test plane is

$$V = 2 \times 295 \times \frac{3600}{1000} = 2124 \text{ km/h}$$

Example 19.2. *To what change in pressure a liquid of specific gravity 1.2 should be subjected to cause reduction in volume by one percent? The velocity of sound in the liquid is 1500 m/s.*

Solution

From Eq. 19.16, we have

$$C^2 = \frac{dp}{d\rho} \quad \ldots(i)$$

and from Eq. 19.17, we have

$$\frac{dv}{v} = -\frac{d\rho}{\rho} \quad \ldots(ii)$$

Multiplying Eqs (i) and (ii), the change in pressure dp is given by

$$dp = -\rho C^2 \frac{dv}{v}$$

$$\rho = 1.2 \times 1000 = 1200 \text{ kg/m}^3; C = 1500 \text{ m/s; and } \frac{dv}{v} = -0.99$$

Thus by substituting, we get

$$dp = \left[-1200 \times (1500)^2 \times (-0.99)\right] \text{N/m}^2$$
$$= 2.673 \times 10^9 \text{ N/m}^2 = 2.673 \times 10^6 \text{ kN/m}^2$$

Example 19.3. *Air flows isentropically around a submerged object. At section 1 in the approaching stream the pressure $p_1 = 101.043$ kN/m^2, the density $\rho_1 = 1.226$ kg/m^3 and the velocity $V_1 = 135$ m/s. At point 2 near the object, the pressure p_2 is observed to be 39.24 kN/m^2. Calculate (a) the temperature ratio between these two points and the Mach number at each point. Take k = 1.4 and R = 287 J/kg. K.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

By combining Eq. 19.2 and the equation of state the following expression can be readily obtained for such a flow

$$\frac{T_1}{T_2} = \left(\frac{p_1}{p_2}\right)^{\frac{k-1}{k}} = \left(\frac{101.043}{39.24}\right)^{\frac{1.4-1}{1.4}} = 1.31$$

Since $p_1 = \rho_1 R T_1$

$$T_1 = \frac{101.043 \times 10^3}{1.226 \times 287} = 287.17 \text{ K}$$

$$\therefore \quad T_2 = \frac{287.17}{1.31} = 219.21 \text{ K}$$

Also $$\left(\frac{\rho_1}{\rho_2}\right) = \left(\frac{p_1}{p_2}\right)^{\frac{1}{k}} = \left(\frac{101.043}{39.24}\right)^{\frac{1}{1.4}} = 1.965$$

$$\therefore \quad \rho_2 = \frac{1.226}{1.965} = 0.624 \text{ kg/m}^3$$

Applying Eq. 19.12 between the sections 1 and 2 with $z_1 = z_2$ and substituting the known values, we get

$$\frac{V_2^2 - (135)^2}{2 \times 9.81} = \frac{1.4}{1.4-1}\left(\frac{101.043}{1.226} - \frac{39.24}{0.624}\right)\frac{10^3}{9.81}$$

$$\therefore \quad V_2 = 393.64 \text{ m/s}$$

$$C_1 = \sqrt{kRT_1}$$

$$= \sqrt{1.4 \times 287 \times 287.17} = 339.68 \text{ m/s}$$

$$\therefore \quad Ma_1 = \frac{V_1}{C_1} = \frac{135}{339.68} = 0.397$$

$$C_2 = \sqrt{kRT_2}$$

$$= \sqrt{1.4 \times 287 \times 219.21} = 296.78 \text{ m/s}$$

$$\therefore \quad Ma_2 = V_2 C_2 = \frac{393.64}{296.78} = 1.326$$

Example 19.3. *Air discharges from a large tank, in which the pressure is 700 kN/m² and temperature 40°C, through a convergent nozzle of 25 mm tip diameter. Calculate the flow rates when the pressure outside the jet is (a) 200 kN/m², and (b) 550 kN/m² and the barometric pressure is 101 kN/m². Also calculate the pressure, temperature, velocity and sonic velocity at the nozzle tip for these flow rates. Take k = 1.4 and R = 286.8 J/kg.K.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

From the equation of state the mass density ρ_1 for the air in the tank is obtained as

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{(700+101)\times 10^3}{286.8\times(273.15+40)} = 8.919 \text{ kg/m}^3$$

From Eq. 19.39, critical pressure ratio is

$$\left(\frac{p_2}{p_1}\right)_c = \left(\frac{2}{2.4}\right)^{3.5} = 0.528$$

(a) $$\left(\frac{p_2}{p_1}\right) = \frac{301}{801} = 0.376 < 0.528$$

Therefore in this case the flow rate is given by Eq. 19.40 as

$$G = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

$$= \frac{\pi}{4}\times(0.025)^2\times(1.4\times 801\times 10^3\times 8.919)^{\frac{1}{2}}\left(\frac{2}{2.4}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 0.898 \text{ kg/s}$$

As explained in Sec. 19.11 for this case

$$\frac{p_2}{p_1} = 0.528$$

$$\therefore \quad p_2 = (0.528\times 801) = 422.93 \text{ kN/m}^2 \text{ (abs)}$$

From Eq. 19.36, we have

$$\frac{V_2^2}{2} = \frac{p_1}{\rho_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]$$

By substituting, we get

$$\frac{V_2^2}{2} = \frac{801\times 10^3}{8.919}\times\frac{1.4}{0.4}\left[1-(0.528)^{\frac{0.4}{1.4}}\right]$$

$$\therefore \quad V_2 = 323.82 \text{ m/s}$$

which is also the sonic velocity at the nozzle tip in this case.

From Eq. 19.2 and the equation of state

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$T_2 = T_1\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}$$

$$= 313.15(0.528)^{\frac{0.4}{1.4}} = 261 \text{ K}$$

(b) $$\left(\frac{p_2}{p_1}\right) = \frac{651}{801} = 0.813 > 0.528$$

Therefore in this case the flow rate is given by Eq. 19.38 as

$$G = A_2\left(\frac{2k}{k-1}p_1\rho_1\right)^{1/2}\left[\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_2}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2}$$

By substituting, we get

$$G = \frac{\pi}{4}\times(0.025)^2\left(\frac{2\times1.4}{1.4-1}\times801\times10^3\times8.919\right)^{1/2}$$

$$\times\left[(0.813)^{\frac{2}{1.4}} - (0.813)^{\frac{1.4+1}{1.4}}\right]^{1/2}$$

$$= 0.718 \text{ kg/s}$$

In this case since

$$\frac{p_2}{p_1} = 0.813$$

$\therefore$ $p_2 = (0.813 \times 801) = 651.21 \text{ kN/m}^2 \text{ (abs)}$

From Eq. 19.36, we have

$$\frac{V_2^2}{2} = \frac{p_1}{\rho_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]$$

By substitution, we get

$$\frac{V_2^2}{2} = \frac{801\times10^3}{8.919}\times\frac{1.4}{0.4}\left[1-(0.813)^{\frac{0.4}{1.4}}\right]$$

$\therefore$ $V_2 = 190 \text{ m/s}$

From Eq. 19.37, we have

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{V_2}{C_2} = \left[\frac{2}{k-1}\left\{\left(\frac{p_1}{p_2}\right)^{\frac{k-1}{k}} - 1\right\}\right]^{1/2}$$

By substitution, we get

$$\frac{190}{C_2} = \left[\frac{2}{1.4-1}\left\{\left(\frac{1}{0.813}\right)^{\frac{0.4}{1.4}} - 1\right\}\right]^{1/2}$$

$\therefore$ Sonic velocity at the nozzle tip is

$$C_2 = 344.22 \text{ m/s}$$

From Eq. 19.2 and the equation of state

$$T_2 = T_1\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}$$

$$= 313.15(0.813)^{\frac{0.4}{1.4}} = 295.16 \text{ K}$$

Example 19.4. *A tank contains air at a gage pressure of 34 kN/m^2 and a temperature of 32.2°C. The local barometric pressure is 101 kN/m^2. Air discharges out of the tank into atmosphere through a convergent nozzle. The exit opening of the nozzle is 25 mm in diameter. The coefficient of discharge C_d for the nozzle may be taken as unity. Assuming insentropic flow, what is the mass rate of air flow through the nozzle. Take k = 1.4 and R = 287 J/kg.K.*

Solution

From the equation of state the mass density ρ_1 for the air in the tank is obtained as

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{(101+34)\times 10^3}{287\times(273.15+32.2)} = 1.54 \text{ kg/m}^3$$

The pressure ratio

$$\frac{p_2}{p_1} = \frac{101}{135} = 0.748$$

which is greater than the critical pressure ratio, and hence the mass rate of flow of air is given by Eq. 19.38 as

$$G = A_2\left(\frac{2k}{k-1}p_1\rho_1\right)^{1/2}\left[\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_2}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2}$$

By substituting, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$G = \frac{\pi}{4}(0.025)^2 \left(\frac{2\times1.4}{1.4-1}\times135\times10^3\times1.54\right)^{1/2} \times\left[\left(\frac{101}{135}\right)^{\frac{2}{1.4}}-\left(\frac{101}{135}\right)^{\frac{1.4+1}{1.4}}\right]^{1/2}$$

$$= 0.136 \text{ kg/s}.$$

Example 19.5. *What will be the mass rate of flow if the pressure in the tank of Example 19.4 is 343.5 kN/m^2? Other data remain the same.*

Solution

The density of air corresponding to a pressure of 343.5 kN/m^2 and a temperature of 32.2°C is

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{(101+343.5)\times10^3}{287\times(273.15+32.2)} = 5.072 \text{ kg/m}^3$$

The pressure ratio

$$\frac{p_2}{p_1} = \frac{101}{444.5} = 0.227$$

which is less than the critical pressure ratio. Therefore the velocity of air flow at the nozzle exit is sonic and the mass rate of flow is given by Eq. 19.40 as

$$G = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

By substitution, we get

$$G = \frac{\pi}{4}(0.025)^2 \times (1.4\times444.5\times10^3\times5.072)^{1/2}\times\left(\frac{2}{1.4+1}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 0.505 \text{ kg/m}^3$$

Example 19.6. *The exit area to throat area ratio for a convergent-divergent nozzle is 2.5, the throat being 12.5 × 10^{-4} m^2. Air enters this nozzle from a reservoir where the absolute pressure is 686.7 kN/m^2 and the temperature is 24°C. Air discharges from the exit area into the atmosphere where the local barometric pressure is 101 kN/m^2. Assume the flow to be isentropic with R = 287 J/kg.K and k = 1.4. Calculate (a) the mass rate of air flow through the nozzle, (b) the exit pressure, (c) the exit temperature, (d) the exit Mach number, and (e) the exit velocity for the following two conditions:*

(*i*) *Sonic velocity at the throat, with divergent section acting as a nozzle;*

(*ii*) *Sonic velocity at the throat, with divergent section acting as a diffuser.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Applying energy equation between the throat (2) and the exit section (3) of the nozzle, we have

$$\frac{p_2}{\rho_2}\frac{k}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = \frac{V_3^2}{2} - \frac{V_2^2}{2}$$

or $$\frac{p_2}{\rho_2}\frac{2}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = R\left[\frac{V_3^2 T_3}{kRT_3} - \frac{V_2^2 T_2}{kRT_2}\right]$$

or $$\frac{p_2}{\rho_2}\frac{2}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = \frac{p_3}{\rho_3}\left(Ma^2\right)_3 - \frac{p_2}{\rho_2}\left(Ma^2\right)_2$$

since $$p = \rho RT \text{ and } \frac{V^2}{kRT} = Ma^2$$

or $$\frac{2}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = \left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\left(Ma^2\right)_3 - \left(Ma^2\right)_2$$

since $$\left(\frac{\rho_2}{\rho_3}\right) = \left(\frac{p_2}{p_3}\right)^{\frac{1}{k}}$$

Solving for (p_3 / p_2), we get

$$\frac{p_3}{p_2} = \left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_3}\right]^{\frac{k}{k-1}} \quad \text{...(i)}$$

By combining the above equation with Eq. 19.2 and the equation of state, the following equation for the temperature ratio may be readily obtained.

$$\frac{T_3}{T_2} = \left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_3}\right] \quad \text{...(ii)}$$

Again by introducing Eq. (i), the equation of state and Eq. 19.2 in the above noted energy equation the following equation for the velocity ratio may be obtained.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{V_3}{V_2} = \frac{(Ma)_3}{(Ma)_2}\left[\frac{1+\frac{k-1}{2}\left(Ma^2\right)_2}{1+\frac{k-1}{2}\left(Ma^2\right)_3}\right]^{1/2} \quad \ldots(iii)$$

By combining Eq. (iii) with the continuity equation the following equation for the area ratio may be obtained.

$$\frac{A_3}{A_2} = \frac{(Ma)_2}{(Ma)_3}\left[\frac{1+\frac{k-1}{2}\left(Ma^2\right)_3}{1+\frac{k-1}{2}(Ma^2)_2}\right]^{\frac{k+1}{2(k-1)}} \quad \ldots(iv)$$

Now using Eq. (iv) to relate conditions at the throat and at the nozzle exit by setting $Ma_2 = 1$ and $(A_3/A_2) = 2.5$, we get

$$2.5 = \frac{1}{(Ma)_3}\left[\frac{1+\frac{1.4-1}{2}\left(Ma^2\right)_3}{1+\frac{1.4-1}{2}(1)^2}\right]^{\frac{1.4+1}{2(1.4-1)}}$$

Solving the above equation by trial and error, we get

$$(Ma)_3 = 2.44 \text{ or } (Ma)_3 = 0.24$$

The former value is for the divergent section acting as a nozzle and the later value corresponds to the condition when the divergent section acts as a diffuser.

(a) When sonic velocity occurs at the throat, the mass rate of flow is the same for both conditions and is determined by using Eq. 19.40.

Thus

$$G = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

in which

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{686.7\times10^3}{287\times(273.15+24)} = 8.052 \text{ kg/m}^3$$

$\therefore$

$$G = 12.5 \times 10^{-4}\sqrt{1.4\times686.7\times10^3\times8.052}\left(\frac{2}{1.4+1}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 2.013 \text{ kg/m}^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) When $(Ma)_3 = 2.44$

(b) By applying Eq. (i) between points 1 in the reservoir and 3 at the exit of the nozzle with $(Ma)_1 = 0$, we get

$$\frac{p_3}{686.7 \times 10^3} = \left[\frac{1}{1+\frac{1.4-1}{2}(2.44)^2}\right]^{\frac{1.4}{1.4-1}}$$

$$\therefore \quad p_3 = 44.128 \text{ kN/m}^2 \text{ absolute}$$

(c) Again by applying Eq. (ii) between points 1 and 3 the exit temperature may be calculated by considering

$$(Ma)_1 = 0 \text{ and } T_1 = (273.15 + 24) = 297.15 \text{ K}$$

Thus $$\frac{T_3}{297.15} = \frac{1}{1+\frac{1.4-1}{2}(2.44)^2}$$

$$\therefore \quad T_3 = 135.6 \text{ K}$$

(d) The exit velocity is given by

$$V_3 = (Ma)_3 \sqrt{kRT_3}$$

$$\therefore \quad V_3 = (2.44) \sqrt{1.4 \times 287 \times 135.6} = 569.5 \text{ m/s}$$

(ii) When $(Ma)_3 = 0.24$

(b) The exit pressure:

$$p_3 = \frac{686.7 \times 10^3}{10^3}\left[\frac{1}{1+\frac{1.4-1}{2}(0.24)^2}\right]^{\frac{1.4}{1.4-1}}$$

$$= 659.7 \text{ kN/m}^2$$

(c) The exit temperature:

$$T_3 = \frac{297.15}{1+\frac{1.4-1}{2}(0.24)^2} = 293.8 \text{ K}$$

(d) The exit velocity:

$$V_3 = 0.24 \sqrt{1.4 \times 287 \times 293.8} = 82.5 \text{ m/s}$$

Example 19.7. *Conditions immediately upstream from a compression shock in an air flow are given as follows: $p_u = 34.3$ kN/m^2 absolute, $T_u = 0°$ C and $V_u = 1000$ m/s. Calculate the conditions immediately downstream from the shock.*

Solution

Assuming $k = 1.4$ and $R = 287$ J/kg.K

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$C_u = \sqrt{kRT_u}$$

$$= \sqrt{1.4 \times 287 \times 273.15} = 331.3 \text{ m/s}$$

$$(Ma)_u = \frac{1000}{331.3} = 3.02$$

From Eq. 19.50, we have

$$\frac{p_d}{34.3} = \frac{2 \times 1.4 \times (3.02)^2 - (1.4 - 1)}{(1.4 + 1)}$$

$$\therefore \quad p_d = 359.3 \text{ kN/m}^3 \text{ absolute}$$

From Eq. 19.51, we have

$$\frac{1000}{V_d} = \frac{(1.4 + 1)(3.02)^2}{(1.4 - 1)(3.02)^2 + 2}$$

$$\therefore \quad V_d = 258 \text{ m/s}$$

From Eq. 19.52, we have

$$(Ma)_d^2 = \frac{(1.4 - 1)(3.02)^2 + 2}{2 \times 1.4(3.02)^2 - (1.4 - 1)}$$

$$\therefore \quad (Ma)_d = 0.474$$

By combining Eqs 19.50 and 19.51, we obtain

$$\frac{T_d}{T_u} = \frac{p_d \rho_u}{p_u \rho_d} = \frac{2k(Ma)_u^2 - (k-1)}{(k+1)} \times \frac{(k-1)(Ma)_u^2 + 2}{(k+1)(Ma)_u^2}$$

$$\therefore \quad T_d = (273.15) \times \frac{359.3}{34.3} \times \frac{258}{1000} = 738 \text{ K}$$

Example 19.8. *Air at 30°C at a pressure of 294.3 kN/m² absolute flows into another reservoir through an orifice 50 × 10⁻⁴ m² in area. If the pressure in the receiver is 147.15 N/m² absolute, what is the discharge? Given R = 287.14 J/kg.K, k = 1.4 and coefficient of discharge C_d = 0.65.*

Solution

The density of air corresponding to a pressure of 294.3 kN/m² and a temperature of 30°C is given by equation of state as

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{294.3 \times 10^3}{287.14 \times (273.15 + 30)} = 3.38 \text{ kg/m}^3$$

The pressure ratio

$$\frac{p_2}{p_1} = \frac{147.15}{294.3} = 0.5$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which is less than the critical pressure ratio. Therefore the mass rate of flow is given by Eq. 19.40 as

$$G = C_d A_2 \sqrt{k p_1 \rho_1} \left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

By substituting, we get

$$G = 0.65 \times 50 \times 10^{-4} \times (1.4 \times 294.3 \times 10^3 \times 3.38)^{1/2} \times \left(\frac{2}{1.4+1}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 2.22 \text{ kg/m}^3$$

SUMMARY OF MAIN POINTS

1. When the density changes are appreciable the compressibility of the fluid needs to be taken into account in the analysis of such fluid flow phenomenon.
2. The density ρ of a particular gas is related to its absolute pressure p and absolute temperature T by the *equation of state,* which for a perfect gas is given as

   $$p = \rho RT$$

   where

   R = gas constant

   The absolute temperature is expressed in kelvin (K) and it is given by

   $$T° \text{ (abs)} = TK = 273.15 + \tau° C$$
3. A gas may be compressed or expanded by either *isothermal process* or *adiabatic process.*

   An isothermal procss is that in which the temperature is held constant and it is governed by Boyles, law according to which

   $$\frac{p}{\rho} = \text{constant}$$

   An adiabatic process is governed by the following law

   $$\frac{p}{\rho^k} = \text{constant}$$

   where

   k = adiabatic index.

   The adiabatic index is given by

   $$k = \frac{C_p}{C_v}$$

   where

   C_p = specific heat at constant pressure; and

   C_v = specific heat at constant volume.

   For air and other diatomic gases in the most ranges of temperature and pressure $k = 1.4$.
4. For ne dimensional steady flow of compressible fluid the equation of continuity is given as

   $$\rho AV = \text{constant}$$

   and in diffrential form continuity equation is given as

   $$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dV}{V} = \text{constant}$$
5. The energy equation for flow of compressible fluid under isothermal condition is given as

   $$\frac{p}{\rho g} \log_e p + \frac{V^2}{2g} + z = \text{constant}$$

   The energy equation for flow of compressible fluid under adiabatic condition is given as

   $$\frac{p}{\rho g} \frac{k}{k-1} + \frac{V^2}{2g} + z = \text{consant}$$
6. The velocity of sound wave is given by

   (i) In terms of Bulk modulus

   $$C = \sqrt{\frac{dp}{d\rho}} = \sqrt{\frac{K}{\rho}}$$

   (ii) For isothermal process

   $$C = \sqrt{\frac{p}{\rho}} = \sqrt{RT}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) For adiabatic process

$$C = \sqrt{\frac{kp}{\rho}} = \sqrt{kRT}$$

7. Mach number M_a (or N_M) is given by

$$M_a = \frac{V}{C}$$

For $M_a < 1$ the flow is sub-sonic flow
For $M_a > 1$ the flow is super-sonic flow
For $M_a = 1$ the flow is sonic flow

8. In a sub-sonic flow the elastic waves will travel in the forward direction ahead of the object.
In a sonic flow both the object and the elastic waves will travel side by side.
In a super-sonic flow the object moves ahead of the elastic waves.

9. Mach angle α is given by

$$\sin \alpha = \frac{C}{V} = \frac{1}{M_a}$$

10. Stagnation point is defined as the point in a fluid stream where the velocity of flow is reduced to zero. The pressure, density and temperature at the stagnation point are called stagnation pressure, stagnation density and stagnation temperature which are given by

$$p_s = p_2 = p_1\left[1 + \frac{k-1}{2}M_{a_1}^2\right]^{\frac{k}{k-1}}$$

$$\rho_s = \rho_2 = \rho_1\left[1 + \frac{k-1}{2}M_{a_1}^2\right]^{\frac{k}{k-1}}$$

$$T_s = T_2 = T_1\left[1 + \frac{k-1}{2}M_{a_1}^2\right]$$

Also $$T_s = \frac{1}{R}\frac{p_s}{\rho_1}$$

11. The dimension less stagnation pressure is given by

$$\frac{p_s - p_1}{\frac{\rho_1 V_1^2}{2}} = \frac{p_2 - p_1}{\frac{\rho_1 V_1^2}{2}} = 1 + \frac{1}{4}M_{a_1}^2 + \frac{2-k}{24}M_{a_1}^4 + \ldots$$

12. Area velocity relationship for compressible fluid flow through a pipe of varying cross-section is given by

$$\frac{dA}{A} = \frac{dV}{V}\left[M_a^2 - 1\right]$$

(i) For sub-sonic flow, i.e., $Ma < 1$

$$\frac{dV}{V} > 0;\ \frac{dA}{A} < 0 \text{ (convergent nozzle)}$$

$$\frac{dV}{V} < 0;\ \frac{dA}{A} > 0 \text{ (divergent diffuser)}$$

(ii) For super-sonic flow, i.e., $Ma > 1$

$$\frac{dV}{V} > 0;\ \frac{dA}{A} > 0 \text{ (divergent nozzle)}$$

$$\frac{dV}{V} < 0;\ \frac{dA}{A} < 0 \text{ (convergent diffuser)}$$

(iii) For sonic flow , i.e., $M_a = 1$

$\frac{dA}{A} = 0$ (straight flow passage since dA must be zero)

13. The velocity of flow of compressible fluid through a convergent nozzle is given by

$$V_2 = \sqrt{\frac{2k}{k-1}\frac{p_1}{\rho_1}\left[1 - \left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]}$$

14. The maximum mass flow rate G_{max} of a compressible fluid through a convergent nozzle is given by

$$G_{max} = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

15. The mass rate of flow G of a compressible fluid through a convergent divergent nozzle is given by

$$G = \left(\frac{2k}{k-1}p_1\rho_1\right)^{1/2} A_3\left[\left(\frac{p_3}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_3}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2}$$

16. When a super-sonic flow is converted to a subsonic flow normal shock waves may occur.
The relation between the pressures on the upstream and downstream of normal shock waves developed in a pipe is given by the following equations:
(i) Equation of Rayleigh line

$$p_u + \frac{(G/A)^2}{\rho_u} = p_d + \frac{(G/A)^2}{\rho_d}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(ii) Equation of Fanno line

$$\frac{(G/A)^2}{2\rho_u^2}+\frac{k}{k-1}\frac{p_u}{\rho_u}=\frac{(G/A)^2}{2\rho_d^2}+\frac{k}{k-1}\frac{p_d}{\rho_d}$$

(iii) Rankine-Hugoniot Equations

$$\frac{p_d}{p_u}=\frac{(k+1)\frac{\rho_d}{\rho_u}-(k-1)}{(k+1)-(k-1)\frac{\rho_d}{\rho_u}}$$

$$\frac{\rho_d}{\rho_u}=\frac{V_u}{V_d}=\frac{(k-1)+(k+1)\frac{p_d}{p_u}}{(k+1)+(k-1)\frac{p_d}{p_u}}$$

17. The mass rate of flow of comprssible fluid G through a venturimeter or orifice meter is given by

$$G = CA_2\rho_2\left[\frac{\frac{2k}{k-1}\frac{p_1}{\rho_1}\left\{1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right\}}{1-\left(\frac{A_2}{A_1}\right)^2\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}}}\right]$$

where

C = coefficient of the meter;
A_1 = area at inlet; and
A_2 = area at the throat.

PROBLEMS

19.1 A volume of air at an atmospheric pressure of 101.04 kN/m² [1.03 kg(f)/cm²] absolute and 24°C is compressed adiabatically to a gage pressure of 3.09×10^7 N/m² [315 kg(f)/cm²]. Calculate the final volume.

$$\left[Ans.\ \left(\frac{1}{59.75}\right)\text{of original volume}\right]$$

19.2 Calculate the pressure on the nose of a modern airplane moving at 800 km/h through still air and the Mach number, when atmospheric pressure and air temperature are 68.7 kN/m²(0.7kg(f)/cm²] absolute and – 1°C respectively. Take mass density of air as 1.295 kg/m³.

[*Ans.* 106.34 kN/m²; 0.82]

19.3 Oxygen flows through a 300 mm circular duct at the rate of 7.22 kN per minute [736 kg(f) per minute]. Gages at points 1 and 2 indicate pressures of 480.7 kN/m² [4.9 kg(f)/cm²] and 48.07 kN/m² [0.49 kg(f)/cm²] respectively. Calculate the Mach number of the flow at points 1 and 2. Assume t = 25°C, R = 263 J/kg.K and k = 1.4. [*Ans.* 0.085; 0.85]

19.4 Air at an absolute pressure of 101.04 kN/m² [1.03 kg(f)/cm²] and 15°C with zero velocity expands isentropically to Ma = 0.50. Calculate (a) the final pressure, (b) the final density, and (c) the temperature. Take R = 287 J/kg.K.

[*Ans.* (a) 85.18 kN/m²; (b) 1.08 kg/m³; (c) 1.28°C]

19.5 Solve Problem 19.4 if the air at an absolute pressure of 101.04 kN/m² and 15°C, with an initial Mach number of 0.50 expands isentropically to a final Mach number of 1.0.

[*Ans.* (a) 63.32 kN/m²; (b) 0.875 kg/m³; (c) – 21°C]

19.6 Air flows isentropically in a variable area duct. At section 1, V_1 = 64 km/hour, p_1 = 101.04 kN/m² absolute and t_1 =22°C. At section 2, p_2 = 68.7 kN/m² absolute. Determine (a) the velocity and temperature at section 2 and (b) the area ratio of these two sections. Take R = 287 J/kg K.

[*Ans.* (a) 249.4 m/s, – 8.8° C; (b) 10.65]

19.7 Derive the relationship

$$\frac{T_2}{T_1}=\left(\frac{p_2}{p_1}\right)^{\frac{(k-1)}{k}}$$

and specifically state the conditions for which the equation is valid.

19.8 Air flows isentropically in a flow passage with increasing velocity in the direction of flow. At section 1, p_1 = 274.68 kN/m² [2.8 kg(f)/cm²] absolute, t_1 = 180°C and V_1 = 150 m/s. At section 2, the air flow just attains the sonic velocity. Calculate for section 2, (a) the pressure, (b) the temperature, (c) the velocity, and (d) the density of air. Take R = 287 J/kg.K.

[*Ans.* (a) 145.11 kN/m²; (b) 104.48° C; (c) 389.53 m/s; (d) 1.339 kg/m³]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

19.9 Considering the variation of velocity with area, prove that

$$\frac{dA}{dV} = \frac{A}{V}(Ma^2 - 1)$$

and explain variation of velocity with change in area for the subsonic and supersonic velocities.

19.10 Nitrogen at an absolute pressure of 123.61 kN/m^2 and 13°C flows in a 300 mm diameter pipe at a velocity of 12 m/s when it enters a 300 mm × 200 mm gradual reducer. Assuming isentropic flow, calculate for the 200 mm section, (a) the velocity, (b) the pressure, and (c) the temperature of nitrogen. Take R = 296.26 J/kg.K [30.2 kg(f)–m/kg(m) deg C abs] and k = 1.4.

[*Ans.* (a) 27.12 m/s; (b) and (c) pressure and temperature are almost the same as that at 300 mm section]

[**Hint.** $(Ma)_1 = 0.035$,

$$\frac{A_2}{A_1} = \frac{(Ma)_1}{(Ma)_2}\left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_1}\right]^{\frac{k+1}{2(k-1)}}$$

from which $(Ma)_2 = 0.0791$. Since $(Ma)_2$ is very small, no appreciable change in pressure and temperature at 200 mm section occurs]

19.11 An ideal gas with k = 2 and R = 225.63 J/kg.K [23 kg(f)-m/kg (m) deg C abs] flows through an isentropic convergent-divergent nozzle. The pressure and temperature at the nozzle inlet are respectively 412.02 kN/m^2 [4.2 kg(f)/cm^2] absolute and 27°C. Air discharges into the atmosphere where the barometric pressure is 740 mm of mercury. For the condition of complete expansion of air as it leaves the exit, calculate (a) the temperature in K at the throat and exit, and (b) the sonic velocities at the inlet, throat and exit sections of the nozzle.

[*Ans.* (a) 200 K, 146.9 K; (b) 368 m/s, 300.4 m/s, 257.5 m/s]

19.12 A pitot tube is used to measure the air velocity in the test section of a wind tunnel. During a test run, the manometer connected to the stagnation tube indicates a total gage pressure of 112 mm of water when the static gage pressure reads 22 mm of water. The barometric pressure is 101 kN/m^2 and the temperature of the atmosphere is 24°C . Calculate the air velocity in the test section. [*Ans.* 4.28 m/s]

19.13 Show that the celerity C of a sound wave in a fluid is given by

$$C^2 = \left(\frac{dp}{d\rho}\right)$$

Hence show that in a perfect gas $C = \sqrt{kRT}$ where k is the adiabatic index, R the gas constant and T the absolute temperature. Also show that for a liquid $C = \left(\sqrt{K/\rho}\right)$ where K is the bulk modulus of elasticity and ρ the density of the fluid.

19.14 Define pressure coefficient. A Prandtl pitot tube attached to the nose of an aircraft gave a pressure coefficient of 1.04. Determine the Mach number of the flying aircraft.

[*Ans.* 0.4]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Impact of Free Jets

Chapter 20

20.1 INTRODUCTION

A jet of fluid emerging from a nozzle has some velocity and hence it possesses a certain amount of kinetic energy. If this jet strikes an obstruction placed in its path, it will exert a force on the obstruction. This impressed force is known as impact of the jet and it is designated as hydrodynamic force, in order to distinguish it from the forces due to hydrostatic pressure. Since a dynamic force is exerted by virtue of fluid motion, it always involves a change of momentum, unlike a force due to hydrostatic pressure that implies no motion. Hence the impluse-momentum principle may be utilized to evaluate the hydrodynamic force exerted on a body by a fluid jet. In this chapter, different cases of forces exerted by free jets on stationary and moving plates or vanes of different shapes have been discussed, which forms an important application of the impulse-momentum principle as mentioned in Chapter 8.

20.2 FORCE EXERTED BY FLUID JET ON STATIONARY FLAT PLATE

(a) Flat Plate Normal to the Jet. Let a jet of diameter d and velocity V issue from a nozzle and strike a flat plate as shown in Fig. 20.1 (a). The plate is held stationary and perpendicular to the centre line

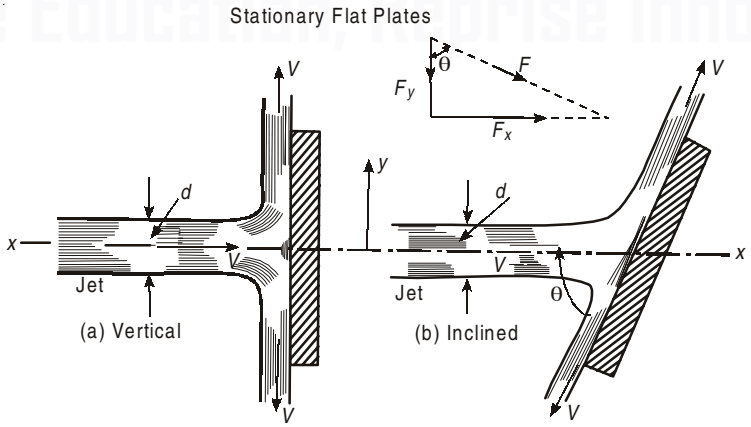


Figure 20.1 *Fluid jet striking stationary flat plate*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the jet. The jet after striking the plate will leave it tangentially i.e., the jet will get deflected through 90°. If the plate is quite smooth the friction between the jet and the plate may be neglected. Further if there is no energy loss in the flow because of impact of the fluid jet, and the difference in elevation between the incoming and outgoing jets is neglected; the application of Bernoulli's equation indicates that the jet will move on and off the plate with the same velocity V. However, if some energy loss occurs the velocity of the fluid leaving the plate will be slightly less than V.

The quantity of fluid striking the plate $Q = (\pi d^2/4) \times V = aV$, where a is the area of cross-section of the jet. Thus the mass of fluid issued by the jet per second is $m = \rho Q = \rho aV$; where ρ represents the mass density of the fluid. Since $\rho = (w/g)$, where w is the specific weight of the fluid, the mass m may also be expressed as $m = (waV/g)$.

After striking the plate since the jet gets deflected through 90°, the component of the velocity of the jet leaving the plate, in the original direction of the striking jet will be zero. Therefore by applying the impulse-momentum equation, the force F exerted by the stationary plate on the jet of fluid in the direction normal to the plate may be determined as

$$-F = \frac{waV}{g}(0 - V)$$

or

$$F = \frac{waV^2}{g} \qquad ...(20.1)$$

The sign for the force F has been considered as negative because the force exerted by the plate on the jet is in the negative x-direction. Further the force which the jet exerts on the plate is equal and opposite to the force exerted by the plate on the jet, hence it is equal to F acting in the positive x-direction and its magnitude is given by Eq. 20.1.

(b) Flat Plate Inclined at Angle θ to the Jet. Let a stationary smooth plate be held inclined at an angle θ to the direction of flow of jet of diameter d and velocity V, as shown in Fig. 20.1 (b). After impact the fluid will slide away in all directions over the plate and it will leave the plate tangentially. It is assumed that there exists no frictional resistance at the plate and there are no impact losses. Further if the small difference in elevation between the incoming and outgoing jets is neglected, the application of Bernoulli's equation then yields that velocity of the divided jets developed after impact will be the same as the velocity of jet before impact i.e., after impact the velocity of jet parallel to the surface of the plate will also be equal to V.

The mass of fluid striking the plate per second is

$$m = \rho Q = \rho aV = (waV/g)$$

where all the notations are same as defined earlier.

The component of the velocity of the jet before impact in the direction normal to the plate = V (sin θ).

Further since the jet after impact leaves the plate tangentially, the component of the velocity of the leaving jet in the direction normal to the plate becomes zero.

Therefore by the applications of the impulse-momentum equation, the thrust exerted by the jet in the direction normal to the plate may be expressed as

$$F = \frac{waV}{g}[(V \sin \theta) - 0]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$F = \frac{waV^2}{g}\sin\theta \quad ...(20.2)$$

There being no resultant thrust acting in the direction tangential to the plate the thrust F given by Eq. 20.2 is the resultant dynamic thrust exerted by the jet.

The dynamic thrust F can be resolved into two components viz., component F_x parallel to the direction of the jet and component F_y normal to the direction of the jet. Thus

$$F_x = F\sin\theta = \frac{waV^2}{g}\sin^2\theta \quad ...(20.3)$$

$$F_y = F\cos\theta = \frac{waV^2}{g}\sin\theta\cos\theta \quad ...(20.4)$$

The division of discharge that results in this case may be computed by applying the condition that there is no resultant force acting in the direction parallel to the plate, because the pressure everywhere is atmospheric and there is no frictional resistance.

Thus if Q_1 and Q_2 represent the discharge flowing in the two directions then according to momentum equation

$$\left(\frac{wQ_1V}{g} - \frac{wQ_2V}{g}\right) - \frac{w}{g}QV\cos\theta = 0$$

or $$(Q_1 - Q_2) - Q\cos\theta = 0 \quad ...(i)$$

Also $$Q_1 + Q_2 = Q \quad ...(ii)$$

Solving Eqs (i) and (ii) the values of Q_1 and Q_2 are obtained as

$$Q_1 = \frac{Q}{2}(1+\cos\theta) \quad ...(20.5)$$

$$Q_2 = \frac{Q}{2}(1-\cos\theta) \quad ...(20.6)$$

Alternatively the expression for the resultant force F may also be obtained by applying the momentum equation along the x and y directions. Thus if F is the force exerted by the plate on the jet and F_x and F_y are its components along the x and y directions respectively then according to the momentum equation

$$-F_x = [\rho Q_1 V\cos\theta - \rho Q_2 V\cos\theta] - \rho QV$$

or $$F_x = \rho V[Q - (Q_1 - Q_2)\cos\theta]$$

and $$F_y = [\rho Q_1 V\sin\theta - \rho Q_2 V\sin\theta]$$

or $$F_y = \rho V(Q_1 - Q_2)\sin\theta$$

For F_x the negative sign has been considered because this component of the force exerted by the plate on the jet is in the negative x-direction.

$\therefore$ $$F = \sqrt{F_x^2 + F_y^2}$$

$$= \sqrt{(\rho V)^2[(Q_1 - Q_2)^2 + Q^2 - 2Q(Q_1 - Q_2)\cos\theta]}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting for Q_1 and Q_2 from Eqs. 20.5 and 20.6 in the above expression, it becomes

$$F = \rho QV \sin\theta = \frac{waV^2}{g} \sin\theta$$

which is same as Eq. 20.2.

Further if α is the angle of inclination of F with x-direction then

$$\tan\alpha = \frac{F_y}{F_x} = \frac{\rho V(Q_1 - Q_2)\sin\theta}{\rho V[Q - (Q_1 - Q_2)\cos\theta]}$$

Again substituting for Q_1 and Q_2 from Eqs. 20.5 and 20.6 in the above expression, it becomes

$$\tan\alpha = \cot\theta = \tan(90° - \theta)$$

which means that the resultant force F is acting in the direction normal to the plate.

It may however be noted that for a given inclination of the plate the magnitude of the resultant force F as well as its inclination α will depend on the magnitudes of Q_1 and Q_2. Thus if

$$Q_1 = Q_2, \text{ then}$$
$$F = F_x = \rho QV; F_y = 0; \text{ and } \alpha = 0$$

i.e. for equal division of discharge the resultant force F will be acting in the direction of the jet.

In both the cases discussed above, since the plate is held stationary, the work done by the force exerted by the jet on the plate is zero.

20.3 FORCE EXERTED BY FLUID JET ON MOVING FLAT PLATE

(a) Flat Plate Normal to Jet. As shown in Fig. 20.2 (a) a flat plate held normal to the jet is assumed to move with a velocity u in the same direction as the jet. In this case absolute velocity of the jet V, will not be the effective velocity with which the jet strikes the plate, because as the jet is about to strike

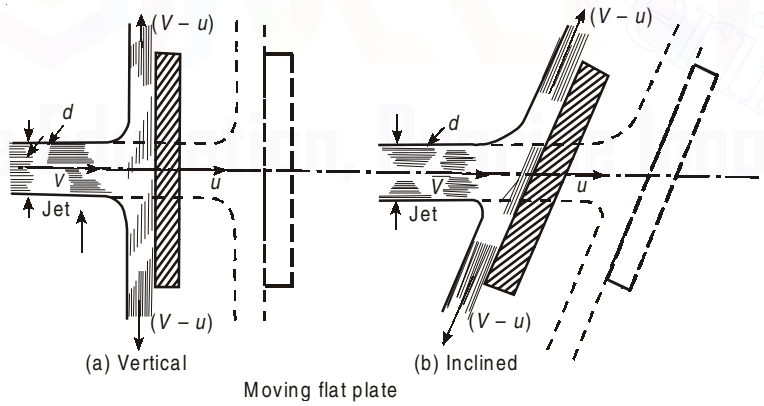


Figure 20.2 *Fluid jet striking moving flat plate*

the plate, the plate has also moved away from the jet with a velocity u, and only that mass of fluid which really overtakes the plate will be striking it to cause the impinging action.

This type of problem may therefore be analyzed by applying the principle of relative motion to the whole system. This is done by bringing the moving plate into a stationary state before the impulse-

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

momentum equation is applied. For this a velocity equal in magnitude to that of the plate but in opposite direction, i.e., $-u$, is applied to the whole system, thereby bringing the plate to rest and making the effective velocity with which the jet strikes the plate equal to $(V-u)$, which is the velocity of the jet relative to the plate.

Thus the quantity of fluid striking the plate per second is

$$Q\left(\frac{\pi d^2}{4}\right)(V-u) = a\,(V-u)$$

and the mass of fluid striking the plate is

$$m = \rho Q = \rho a\,(V-u) = \left[\frac{wa(V-u)}{g}\right]$$

Again if there is no frictional resistance offered by the plate to the jet, as also there are no impact losses and the small difference in elevation between the incoming and outgoing jets is neglected, the application of Bernoulli's equation indicates that the jet leaving the plate will have the velocity equal to $(V-u)$. Further since the jet, after impact, gets deflected through 90°, the component of the velocity of the leaving jet in the direction normal to the plate is equal to zero. Therefore by applying the impulse-momentum equation the dynamic thrust exerted by the jet on the plate in the direction normal to the plate is obtained as

$$F = \left[\frac{wa(V-u)}{g}\right][(V-u)-0]$$

or

$$F = \frac{wa(V-u)^2}{g} \qquad ...(20.7)$$

The work done by the jet on the plate per second

$$= (F \times u) = \frac{wa(V-u)^2 u}{g}$$

The above discussed case will not be practically feasible because the distance between the plate and nozzle will be constantly increasing at the rate of u per second, which in turn would require a continuous lengthening of the jet at this rate in order to strike the plate. This difficulty may however be overcome, if instead of a single plate, there is a continuous series of plates, arranged at a fixed distance apart so that each plate moving with a velocity u, may appear successively before the jet in the same position, and all moving in the same direction as the jet. This condition may be accomplished if a number of flat smooth plates are mounted radially at equal spacing on the periphery of a large wheel, which is capable of rotating in a vertical plane.

Accordingly as shown in Fig. 20.3, let a jet of diameter d and velocity V strike the wheel tangentially at its lower edge, such that the bottom-most vertical plate is impinged at its centre. The impact of jet will push the plate in the same direction as the jet and its position will be occupied by another plate in succession. Likewise the wheel will commence revolving at a constant angular speed. Corresponding to this angular speed, let u be the tangential velocity of the wheel along the centre of the plates, which will be the velocity of the plates as well.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

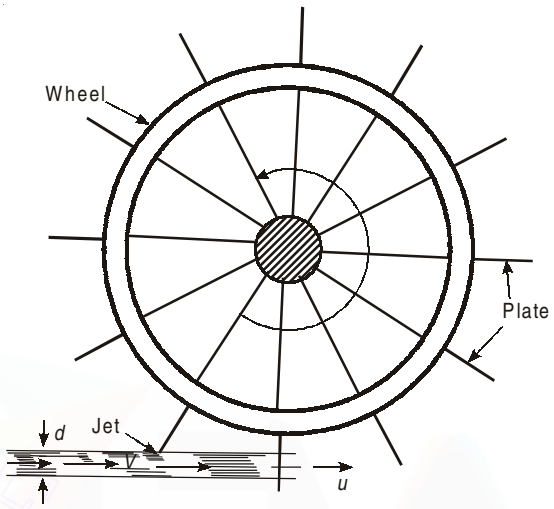


Figure 20.3 *Flat plates mounted on the periphery of a wheel*

In this case since the plates face the jet normally one after the other it is as if one plate is always in the vertical position to intercept the jet. Hence the entire fluid issuing from the nozzle will strike the plates when all the plates are considered. Therefore the mass of fluid striking the plates is

$$m = \rho\left(\frac{\pi d^2}{4}\right)V = \left(\frac{waV}{g}\right)$$

Again the effective velocity of the jet before impact will be $(V - u)$ which is the relative velocity of the jet with respect to the plate. Further since the jet will leave the plate tangentially, the component of the velocity of the jet leaving the plate, in the direction normal to the plate is equal to zero. Therefore by applying the impulse-momentum equation to the flow system, the force exerted by the jet in the direction normal to the plate is given by

$$F = \frac{waV}{g}[(V - u) - 0]$$

or

$$F = \frac{waV}{g}(V - u) \qquad ...(20.8)$$

The work done by the jet per second will be the output of the jet.

Thus, output of jet = $(F \times u)$

$$= \frac{waV(V - u)u}{g}$$

The kinetic energy supplied by the jet per second will be input of the jet to the wheel.

Thus, input of jet

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

∴ Efficiency of the wheel is

$$\eta = \frac{\text{Output of jet}}{\text{Input of jet}}$$

or
$$\eta = \frac{\frac{waV}{g}(V-u)u}{\frac{1}{2}\frac{waV}{g}(V^2)}$$

or
$$\eta = \frac{2u(V-u)}{V^2} \qquad ...(20.9)$$

For a given jet velocity V, the efficiency is maximum if

$$\left(\frac{d\eta}{du}\right) = 0$$

Thus
$$\frac{d\eta}{du} = \frac{2(V-2u)}{V^2} = 0$$

or
$$u = \frac{V}{2}, \text{ since } \frac{2}{V^2} \neq 0$$

Hence for the efficiency of the wheel to be maximum,

$$u = \left(\frac{V}{2}\right)$$

and the corresponding maximum efficiency is

$$\eta_{max} = \frac{2u(2u-u)}{4u^2} = 0.5 \text{ or } 50\%.$$

(b) Flat Plate Inclined at Angle θ to Jet. Let a smooth plate held inclined at an angle θ to the direction of flow of jet, be moving with velocity u in the same direction as the jet, as shown in Fig. 20.2 (b). As stated earlier the effective velocity with which the jet strikes the plate will be the velocity of the jet relative to the plate i.e., $(V-u)$.

Thus the mass of fluid striking the plate per second is

$$m = \rho\left(\frac{\pi d^2}{4}\right)(V-u) = \left[\frac{wa(V-u)}{g}\right]$$

where all the notations are as defined earlier. Further the component of the velocity $(V-u)$ in the direction normal to the plate is $[(V-u)\sin\theta]$.

Again if there are no losses due to friction on the plate as well as due to impact and if the small differece in elevation is neglected the jet will come off the plate with velocity $(V-u)$. But as it leaves the plate tangentially, the component of this velocity in the direction normal to the plate will be zero.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Therefore by applying the impulse-momentum equation the force exerted on the plate in the direction normal to the plate is given by

$$F = \frac{wa(V-u)}{g}[(V-u)\sin\theta - 0]$$

or

$$F = \frac{wa(V-u)^2}{g}\sin\theta \qquad ...(20.10)$$

The normal thrust F can be resolved in two components viz., F_x parallel to the direction of the jet and F_y normal to the direction of jet. Thus,

$$F_x = F\sin\theta = \frac{wa(V-u)^2}{g}\sin^2\theta \qquad ...(20.11)$$

$$F_y = F\cos\theta = \frac{wa(V-u)^2}{g}\sin\theta\cos\theta \qquad ...(20.12)$$

The work done by the jet per second

$$= (F_x \times u) = \frac{wa(V-u)^2}{g}u\sin^2\theta$$

It is obvious from the above expression that for $\theta = 90°$ the same result as for a flat pate normal to the jet will be obtained.

20.4 FORCE EXERTED BY A FLUID JET ON STATIONARY CURVED VANE

On the basis of the direction of the jet striking the curved vane or plate, two distinct cases as discussed below may be considered.

(a) Jet Striking a Symmetrical Stationary Curved Vane at the Centre. As shown in Fig. 20.4, let a horizontal jet of cross-sectional area a and velocity V be striking a smooth symmetrical curved vane at its centre on the concave side. After striking the vane the jet will divide itself and the divided jets will glide over the vane and will leave the vane at its both the ends. The vane being smooth the velocity of the leaving jets will also be V. Let 2θ be the angle between the two tangents drawn to the vane at its outlet tips. Thus the jet after striking, will be deflected on each side through the angle of $(180°-\theta)$. Accordingly the component of the velocity of the leaving jet in the direction of flow of incoming jet = $(V\cos(180°-\theta)] = (-V\cos\theta)$. Therefore the application of the impulse-momentum equation gives the force exerted on the plate in the direction of flow of jet as

$$F = \frac{waV}{g}[V-(-V\cos\theta)]$$

or

$$F = \frac{waV^2}{g}(1+\cos\theta) \qquad ...(20.13)$$

When $\theta = 90°$(or $180°-\theta = 90°$) the curved vane will be reduced to a flat plate, and the expression for F then becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F = \frac{waV^2}{g} \qquad \ldots(20.14)$$

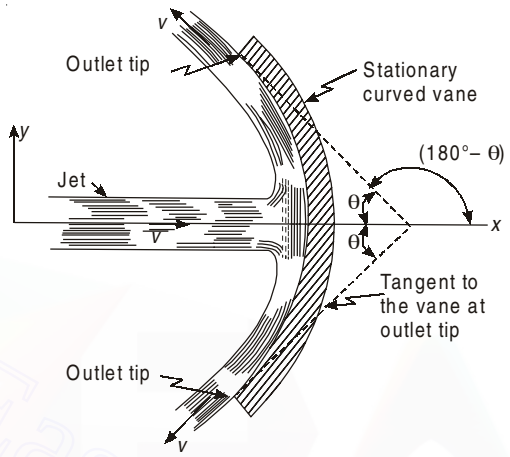


Figure 20.4 *Jet striking a stationary symmetrical curved vane at its centre*

which is same as previously obtained expression in the case of a flat plate held fixed in position. By comparing Eqs 20.13 and 20.14 it will be observed that the force exerted by a jet in its direction of flow, on a curved vane is always greater than that exerted on a flat plate, because

$$\frac{waV^2}{g}(1+\cos\theta) > \frac{waV^2}{g}$$

When $\theta = 0$ (or $180° - \theta = 180$), the vane becomes semicircular and the jets leaving at the outlet tips of the vane will be parallel to the incoming jet, but their directions will be opposite to the direction of the incoming or striking jet. The expression for F then becomes

$$F = \frac{2waV^2}{g} \qquad \ldots(20.15)$$

which indicates that the force exerted by a jet in its direction of flow, on a semicircular vane is twice that exerted on a flat plate, both plates being fixed in position.

From the foregoing discussion it is thus observed that as the angle of deflection of jet increases from 90° (for a flat plate) to 180 (for a semicircular plate) the value of F also increases from $\left(\frac{waV^2}{g}\right)$ to $\left(\frac{2waV^2}{g}\right)$.

Obviously on account of the symmerty the net force acting on the vane in the direction normal to the direction of flow of jet will be equal to zero. Hence in this case the resultant thrust acting on the vane will be equal to F.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) Jet Striking an Unsymmetrical Stationary Curved Vane Tangentially at one of the Tips. Let a jet of cross-sectional area a and velocity V strike tangentially at one of the tips, called inlet tip, of a smooth curved vane as shown in Fig. 20.5. The jet after gliding over the inner surface of the vane will leave it tangentially at its other tip called outlet tip. Let the tangent at the inlet tip make an angle θ with the horizontal x-axis and that at the outlet tip make an angle ϕ with this axis. The jet will therefore get deflected through an angle $[180 - (\theta + \phi)]$. The components of the velocity of jet at the inlet and the outlet tips of the vane in the x-direction will be $(V \cos \theta)$ and $(-V \cos \phi)$ respectively. Therefore by applying the impulse-momentum equation, the force exerted by the jet on the vane in the x-direction is obtained as

$$F_x = \frac{waV}{g}[V \cos \theta - (-V \cos \phi)]$$

or
$$F_x = \frac{waV^2}{g}(\cos \theta + \cos \phi) \qquad \ldots(20.16)$$

If the curvature of the vane at the outlet tip is such that angle ϕ is more than 90°, then the component of the velocity of the jet at the outlet tip will be $(V \cos 180° - \phi)$. Accordingly the expression for the force F_x is again same as given by Eq. 20.16.

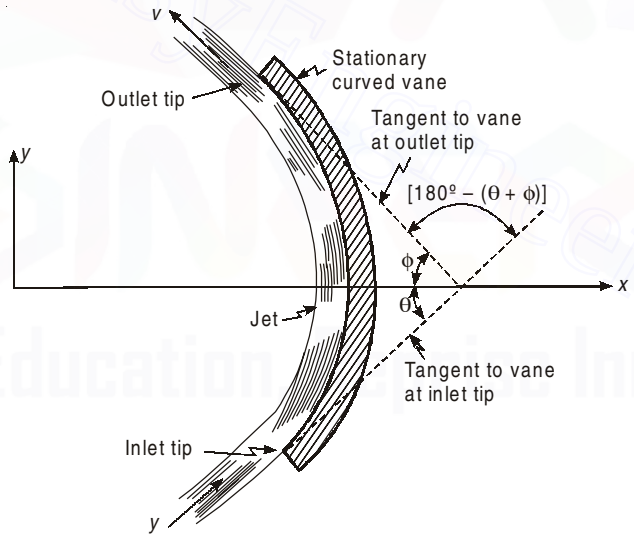


Figure 20.5 *Jet striking tangentially at one of the tips of a stationary unsymmetrical curved vane*

The components of the velocity of jet at the inlet and the outlet tips of the vane in the y-direction will be $(V \sin \theta)$ and $(V \sin \phi)$ respectively. The application of impulse-momentum equation then gives the force exerted by the jet on the vane in the y-direction as

$$F_y = \frac{waV}{g}(V \sin \theta - V \sin \phi)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$F_y = \frac{waV^2}{g}(\sin\theta - \sin\phi) \quad \ldots(20.17)$$

The resultant thrust exerted by the jet on the vane will be $F = \sqrt{F_x^2 + F_y^2}$ and its inclination with the x -axis is

$$\alpha = \tan^{-1}\left(\frac{F_y}{F_x}\right) \quad \ldots(20.18)$$

If the curvature of the vane is same at both inlet and outlet tips i.e., $\theta = \phi$, as shown in Fig. 20.6, then the expression for F_x is reduced to

$$F_x = \frac{2waV^2}{g}\cos\theta \quad \ldots(20.19)$$

and the value of F_y becomes zero. As such in this case F_x represents the resultant thrust on the vane.

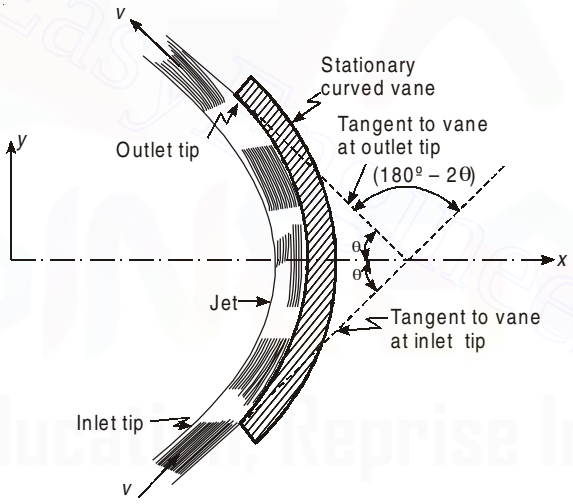


Figure 20.6 *Jet striking tangentially at one of the tips of a stationary symmetrical curved vane*

Further if $\theta = \phi = 0$ [or $180° - (\theta + \phi) = 180°$], the vane becomes semicircular and therefore the incoming and the outgoing jets will be parallel to the horizontal axis of symmetry but the two jets will be in opposite direction. The expression for F_x then becomes

$$F_x = \frac{2waV^2}{g} \quad \ldots(20.20)$$

which is same as obtained previously in section 20.4 for a semicircular vane.

In the expressions derived above the velocity of jet has been assumed to be same at both inlet and outlet tips of the vane. It will hold good if there is no friction on the vane, there are no impact losses and the small difference in elevation between the inlet and the outlet ends of the vane is neglected.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, if some frictional resistance is offered to the jet as it glides over the vane and/or if there occurs an impact loss, the velocity of jet at outlet end will not be the same as that at the inlet end. Under these circumstances

$$F_x = \left(\frac{waV_1}{g}\right)(V_1 \cos\theta + V_2 \cos\phi) \qquad \ldots(20.21)$$

$$F_y = \left(\frac{waV_1}{g}\right)(V_1 \sin\theta - V_2 \sin\phi) \qquad \ldots(20.22)$$

where V_1 and V_2 represent the velocities of the jet at the inlet and the outlet ends of the vane respectively, a is the cross-sectional area of the jet at inlet and the other notations are same as defined earlier.

In both these cases since the vane is held stationary the work done is equal to zero.

20.5 FORCE EXERTED BY A FLUID JET ON MOVING CURVED VANE

(a) Jet Striking Symmetrical Moving Curved Vane at the Centre. Let the curved vane of section 20.4 (a) be allowed to move with a velocity u in the direction of jet as shown in Fig. 20.7. Thus as stated earlier the effective velocity with which the jet strikes the vane will be the velocity of jet relative to the plate i.e., $(V - u)$. Accordingly the mass of fluid striking the vane per second will be $m = [\rho a\,(V-u)] = [wa\,(V-u)/g]$.

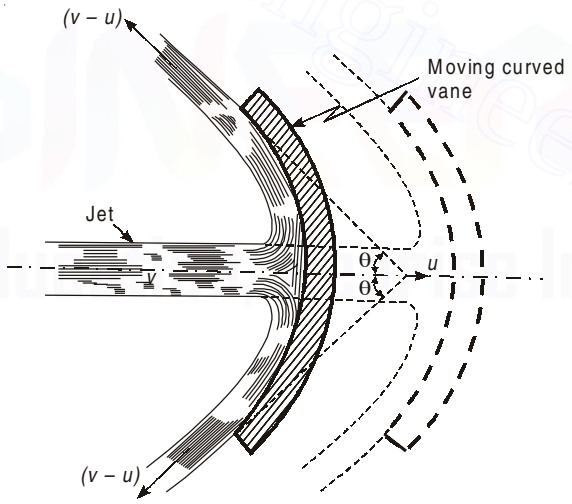


Figure 20.7 *Jet striking moving symmetrical curved vane at its centre*

Further in accordance with the assumptions stated earlier the jet leaving the vane will have its velocity equal to the effective velocity of the incoming jet i.e., $(V-u)$. Thus the component of the velocity of the jet leaving the vane in the direction of the incoming jet is $[(V-u)\cos(180-\theta)]$ or $[-(V-u)\cos\theta]$. Therefore by applying the impulse-momentum equation the force exerted by the jet on the plate in the direction of flow of incoming jet is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$F = \frac{wa(V-u)}{g}[(V-u) - (V-u)\cos(180° - \theta)]$$

or
$$F = \frac{wa(V-u)^2}{g}(1+\cos\theta) \quad \ldots(20.23)$$

When $\theta = 90°$ (or $180° - \theta = 90°$), the above obtained expressions is reduced to

$$F = \frac{wa(V-u)^2}{g} \quad \ldots(20.24)$$

which is the expression for the force exerted by a jet on a moving flat plate as derived in Sec. 20.3 (a) A comparison of Eqs. 20.23 and 20.24 indicates that the force exerted by a jet on a moving curved vane is greater than that on a moving flat plate.

When $\theta = 0$ (for $180° - \theta = 180$), the vane becomes semicircular and the expression for the force exerted by the jet in its direction of flow becomes

$$F = \frac{2wa(V-u)^2}{g} \quad \ldots(20.25)$$

The work done by the jet on the vane per second

$$= (F \times u)$$

$$= \frac{wa(V-u)^2}{g}(1+\cos\theta)\,u$$

which will be the output of the jet in this case.

The kinetic energy supplied by the jet per second

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2 = \frac{waV^3}{2g}$$

which will be the input of the jet. Therefore efficiency of the jet is

$$\eta = \frac{\text{Output of jet}}{\text{Input of jet}}$$

$$= \left[\frac{wa(V-u)^2(1+\cos\theta)u/g}{(waV^3/2g)}\right]$$

$$= \frac{2u(V-u)^2(1+\cos\theta)}{V^3}$$

For a jet of constant velocity V, striking a vane of constant angle θ, the efficiency is maximum if

$$(d\eta/du) = 0.$$

Thus
$$\frac{d\eta}{du} = \frac{2(V^2 - 4uV + 3u^2)(1+\cos\theta)}{V^3} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since $\frac{2(1+\cos\theta)}{V^3} \neq 0$ being constant for a given system,

$$V^2 - 4uV + 3u^2 = 0$$

or $$(V-u)(V-3u) = 0$$

i.e., $$V = u \text{ or } V = 3u$$

Evidently if $V = u$ the work done by the jet will be equal to zero, hence for maximum η, $V = 3u$ and then

$$\eta_{max} = \frac{2u[(3u)^2 - 2u(3u) + u^2](1+\cos\theta)}{(3u)^3}$$

or $$\eta_{max} = \frac{8}{27}(1+\cos\theta) = \frac{16}{27}\cos^2\frac{\theta}{2}$$

For a semicircular vane since $\theta = 0$ the maximum efficiency becomes $\eta_{max} = 0.592$ which is slightly less than 0.6.

Obviously the case discussed above will not be practically feasible because the distance between the vane and the nozzle issuing the jet will be constantly increasing at the rate of u per second, which would required a continuous lengthening of the jet. This difficulty may however be overcome if instead of a single curved vane, there is a series of such vanes arranged at equal spacing on the periphery of a large wheel which is capable of rotating in a vertical plane. Thus as the wheel rotates, each vane will become normal to the jet in turn, so that the entire fluid of the incoming jet issued from the nozzle will be utilized in striking the vanes. Thus for a jet of cross-sectional area a and velocity V, the mass of water striking the wheel per second is $m = (\rho aV) = (waV/g)$.

Further if u is the tangential velocity of the wheel along the centre of the vanes, the effective velocity of the jet before impact will be the relative velocity of the jet with respect to that of the vane, i.e., $(V-u)$. Since the velocity of the jet leaving the vane is also equal to $(V-u)$ and it leaves the vane tangentially, the component of the velocity of the leaving jet in the direction of the incoming jet will be $[(V-u)\cos(180°-\theta)]$. Therefore by applying the impulse-momentum equation the force exerted by the jet on the vanes in the direction of flow of jet is given by

$$F = \frac{waV}{g}[(V-u) - (V-u)\cos(180°-\theta)]$$

or $$F = \frac{waV(V-u)}{g}(1+\cos\theta) \qquad \ldots(20.26)$$

The work done on the wheel per second will be the output of the jet. Thus output of the jet

$$= (F \times u)$$

$$= \frac{waV(V-u)}{g}(1+\cos\theta)\,u$$

Again the kinetic energy of the jet per second will be the input of the jet. Thus input of the jet

$$= \frac{1}{2}\frac{waV}{g}(V^2) = \frac{waV^3}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Therefore the efficiency of the wheel is

$$\eta = \frac{\text{Work done per second}}{\text{Kinetic energy of jet per second}}$$

or
$$\eta = \frac{2u(V-u)(1+\cos\theta)}{V^2}$$

For a jet of constant velocity V, striking a given wheel, the efficiency will be maximum when $(d\eta/du) = 0$. Thus

$$\frac{d\eta}{du} = \frac{2(V-2u)(1+\cos\theta)}{V^2} = 0$$

Since $\frac{2(1+\cos\theta)}{V^2} \neq 0$, being constant for a given system

$$V - 2u = 0, \quad \text{or } V = 2u.$$

Hence for maximum efficiency $V = 2u$ and by substitution the value of maximum efficiency is obtained as

$$\eta_{max} = \frac{2u(2u-u)(1+\cos\theta)}{(2u)^2}$$

or
$$\eta_{max} = \frac{1}{2}(1+\cos\theta)$$

For $\theta = 90°$, the curved vanes are reduced to flat plates mounted on the periphery of a wheel. In this case $\eta_{max} = 0.5$ or 50%, which is same as obtained previously.

For $\theta = 0$ the curved vanes will become semicircular. Then the value of $\eta_{max} = 1$ or 100%. This is however a theoretical value of maximum efficiency for a wheel provided with semicircular curved vanes mounted on its periphery. Similar is the principle employed in the case of bucket of a Pelton wheel. The bucket of a Pelton wheel is double semi-ellipsoidal in shape. The jet of water impinges at the centre of the bucket and deflects through 160° to 170° depending upon the particular design.

In the above derived expressions it has been assumed that no loss of energy either due to impact or due to frictional resistance takes places as the jet strikes the vane and it glides over the same. However, if some loss of energy occurs, the relative velocity at the outlet end (or tip) of the vane will be less than that at the inlet end. Thus, if the relative velocity at the outlet end is $k(V-u)$, where k is a coefficient having a value less than 1, Eq. 20.23 becomes

$$F = \frac{wa(V-u)^2}{g}(1+k\cos\theta) \qquad ...(20.27)$$

Similarly Eq. 20.26 may also be written as

$$F = \frac{waV(V-u)}{g}(1+k\cos\theta) \qquad ...(20.28)$$

The other expressions may also be modified accordingly.

(b) Jet Striking an Unsymmetrical Moving Curved Vane Tangentially at One of the Tips. Let the curved vane of section 20.4 (b) be moving with a velocity u in the x-direction as shown

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

in Fig. 20.8 when a jet of cross-sectional area a and velocity V impinges on the vane as before. Since in this case the vane is also moving, as stated earlier, the effective velocity of the jet entering the vane will be equal to the relative velocity of the jet with respect to the vane. Thus, if V_r represents the relative velocity between the jet and the vane at the entrance, it is the velocity with which the jet actually enters the vane at its inlet tip and its magnitude may be found by subtracting the vectors V and u. Referring to Fig. 20.8 draw ab to represent the velocity V in magnitude and direction to some convenient scale. Next draw ac to represent the velocity u in magnitude and direction to the same

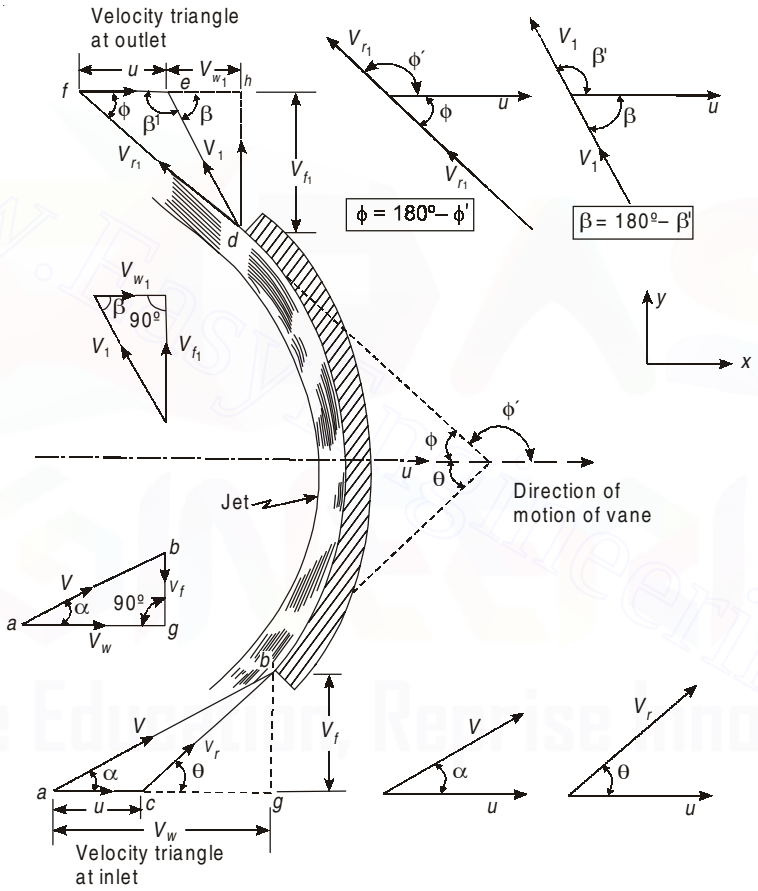


Figure 20.8 *Jet striking tangentially at one of the tips of a moving curved vane*

scale. Then cb represents the relative velocity V_r between the jet and the vane in magnitude and direction to the same scale. In order that the jet may enter the vane smoothly without shock, cb must be parallel to the tangent to the vane at its inlet tip. The triangle abc is known as velocity triangle at inlet tip, in which the absolute velocity of the jet V is the resultant of the velocity of the vane u and the relative velocity V_r.

Since the jet is assumed to impinge on the vane tangentially, there is no loss of energy because of impact. Further the friction loss of the flow passing along the smooth surface of the vane is almost

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

equal to zero, after entry the jet will glide on the inside surface of the vane with velocity V_r, if the small difference in elevation between the two ends of the vane is neglected. Thus the jet will leave the vane at the outlet tip *d*, with relative velocity V_{r1} which will be equal to V_r. However, if the loss of energy due to friction cannot be neglected, V_{r1} will be less than V_r. The absolute velocity of the leaving jet may be found by drawing the triangle of velocities at outlet tip of the vane as indicated below.

Referring to Fig. 20.8, draw *df* to represent the relative velocity V_{r1} in magnitude and direction to some convenient scale. Again if the jet is to leave the vane without shock (or without causing the impact loss), V_{r1} should be parallel to the tangent to the vane at its outlet tip. Next draw *fe* to represent to the same scale, the velocity of the vane u in magnitude and direction. Then *de* represents (to the same scale) the absolute velocity V_1 of the jet coming out at the outlet tip. Again the absolute velocity of the leaving jet V_1 is the resultant of the relative velocity of the leaving jet V_{r1} and the velocity of the vane u; and the triangle *def* is called the velocity triangle at the outlet tip of the vane.

The velocity of the entering jet V may be resolved into two components, one parallel to and other perpendicular to the direction of the motion of the vane. Thus, *ag* represents the component of V parallel to the direction of the motion of the vane, which is commonly represented by V_w , and is known as velocity of whirl at inlet. Further *bg* represents the component of V perpendicular to the direction of the motion of the vane, which is usually represented by V_f and is known as velocity of flow at inlet. Similarly the absolute velocity V_1 of the jet leaving the vane also may be resolved into two components along the same directions. Thus, *eh* represents the components of V_1 parallel to the direction of the motion of the vane, which is represented by V_{w1}, known as velocity of whirl at outlet and *dh* represents the component of V_1 perpendicular to the direction of the motion of the vane, which is represented by V_{f1}, known as velocity of flow at outlet.

In the velocity triangle *abc* at inlet $\angle bac$ is the angle at which the jet strikes the vane. This angle is usually represented by α and it is the angle between the directions of the velocities V and u. Also $\angle bcg$ generally represented by θ is the angle between the directions of the velocities V_r and u. As indicated earlier since the direction of V_r is parallel to the tangent to the vane at its inlet tip, θ denotes the angle of the vane at inlet.

Similarly in the velocity triangle *def* at outlet, $\angle deh$ is the angle between the direction of the absolute velocity of the jet leaving the vane and the direction of motion of the vane. It is generally represented by β. Also $\angle dfe$ is the angle between the directions of the velocities V_{r1} and u, which is represented by ϕ. Again, since the direction of V_{r1} is parallel to the tangent to the vane at its outlet tip, ϕ denotes the angle of the vane at outlet.

Actually from the usual convention the angles measured in the anticlockwise direction are considered as positive. Hence as shown in Fig. 20.8 the angle between the velocities u and V_1 is β' but for the sake of convenience the angle between these velocities is considered as β, where $\beta = (180° - \beta')$. Similarly the angle between the velocities u and V_{r1} is ϕ' but again the angle of the vane at the outlet tip is considered as ϕ, where $\phi = (180° - \phi')$.

This problem may also be analyzed by applying the principle of relative motion to the whole system. For this the moving vane is brought to stationary state by applying a velocity equal in magnitude to that of the vane but in opposite direction, i.e., $-u$ on the whole system. This will then reduce the problem to that of a stationary curved vane subjected to an impinging action of a jet entering tangentially at one of the tips with an effective velocity equal to V_r and leaving tangentially at the other tip with an effective velocity V_{r1}.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus the mass of fluid striking the vane per second is

$$m = (\rho\, aV_r) = (waV_r/g) = (W/g)$$

The component of velocity V_r at inlet in the direction of motion of the vane

$$= (V_r \cos\theta) = (V_w - u) \qquad [\text{From } \Delta abg]$$

Similarly the component of velocity V_{r1} at outlet in the direction of the motion of the vane

$$= (-V_{r1} \cos\phi) = -(u + V_{w1}) \qquad [\text{From } \Delta dfh]$$

The negative sign has been introduced in the above expression because the direction of this component is opposite to that of the motion of the vane. Moreover this expression will hold good as long as β is an acute angle. However, when $\beta = 90°$, $V_{w1} = 0$, in which case $(-V_{r1} \cos\phi) = -(u)$; and when β is an obtuse angle

$$(-V_{r1} \cos\phi) = -(u - V_{w1}).$$

Therefore by applying the impulse-momentum equation the force exerted on the vane in the direction of motion of the vane is obtained for different values of β as noted below.

When β is an acute angle

$$F_w = \frac{waV_r}{g}[(V_w - u) - \{-(u + V_{w1})\}]$$

or

$$F_w = \frac{waV_r}{g}(V_w + V_{w1}) \qquad ...(20.29)$$

when $\beta = 90°$

$$F_x = \frac{waV_r}{g}[(V_w - u) - (-u)]$$

or

$$F_x = \frac{waV_r}{g}(V_w) \qquad ...(20.30)$$

When β is an obtuse angle

$$F_x = \frac{waV_r}{g}[(V_w - u) - \{-(u - V_{w1})\}]$$

or

$$F_x = \frac{waV_r}{g}(V_w - V_{w1}) \qquad ...(20.31)$$

Therefore, the general expression for F_x becomes

$$F_x = \frac{waV_r}{g}(V_w \pm V_{w_1}) \qquad ...(20.32)$$

The work done per second by the jet on the vane

$$= (F_x \times u) = \frac{waV_r}{g}(V_w \pm V_{w_1})\, u \qquad ...(20.33)$$

The work done per second is also equal to the change in kinetic energy of the jet per second.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hence, the work done per second

$$= \left[\frac{1}{2}\frac{waV_r}{g}(V^2) - \frac{1}{2}\frac{waV_r}{g}(V_1^2)\right]$$

$$= \left(\frac{waV_r}{2g}\right)(V^2 - V_1^2) \qquad ...(20.34)$$

Equating the work done represented by Eqs. 20.33 and 20.34, we obtain

$$\frac{waV_r}{g}(V_w \pm V_{w1})\,u = \frac{waV_r}{2g}(V^2 - V_1^2)$$

or $$2(V_w \pm V_{w1})u = (V^2 - V_1^2)$$

It may, however, be stated that the above expression will hold good only if $V_r = V_{r1}$ (i.e., when there is no loss of energy).

The work done by the jet on the vane will be the output of the jet, and the initial kinetic energy of the jet will be the input. Thus, the efficiency of the jet η may be expressed as

$$\eta = \frac{\left(\frac{waV_r}{g}\right)(V_w \pm V_{w1})u}{\frac{1}{2}\left(\frac{waV}{g}\right)(V^2)} \qquad ...(20.35)$$

Similarly the efficiency of the jet may also be expressed as

$$\eta = \frac{\frac{1}{2}\left(\frac{waV_r}{g}\right)(V^2 - V_1^2)}{\frac{1}{2}\left(\frac{waV}{g}\right)V^2} \qquad ...(20.36)$$

As explained earlier the case discussed above is also not practically feasible. But the corresponding practical case may be that of a series of such vanes fixed radially to the rim of a wheel. Such types of wheels are usually provided for radial flow hydraulic turbines.

A *radial flow* is one in which a fluid particle during its flow through the vane of a rotating wheel, remains in a plane normal to the axis of rotation and its distance from the axis of rotation is continuously changing. On the other hand in case of *tangential flow,* considered in sections 20.3 and 20.5 ; the distance of a fluid particle from the axis of rotation remains same, as it flows through the vanes of rotating wheel. As such in the case of tangential flow the action of jet on the vanes can be determined by applying impulse-momentum equation as indicated earlier. But in the case of radial flow, since the distance of a fluid particle from the axis of rotation varies along its path of flow, the same has to be taken into account while computing the action of jet on the vanes. Hence instead of finding the force by impulse-momentum equation, the action of jet on the vanes for radial flow is determined by evaluating the torque or turning moment produced on the vanes. The torque may be evaluated on the basis of angular momentum principle indicated in Sec. 8.8 of Chapter 8.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

20.6 TORQUE EXERTED ON A WHEEL WITH RADIAL CURVED VANES

Figure 20.9 shows a part of the wheel of a radial flow hydraulic turbine. For the sake of convenience only one vane out of the series of the vanes is shown. Due to the impact of the jet on the vanes, the wheel rotates. Now if R and R_1 are the radii of the wheel at the inlet and the outlet ends of the vane respectively, then since the entire wheel rotates at the same angular velocity ω, the tangential velocities at the inlet and the outlet tips of the vane will be different. Thus if u and u_1 are the tangential velocities at the inlet and the outlet tips of the vane then

$$u = \omega R = \frac{2\pi N}{60} R \text{ and } u_1 = \omega R_1 = \frac{2\pi N}{60} R_1$$

where N represents the speed of the wheel in r.p.m. The velocity triangles at the inlet and the outlet tips of the vane may thus be obtained in the same manner as discussed earlier in section 20.5 (b).

The tangential component or the component of absolute velocity V of the incoming jet in the direction of motion u is

$$V \cos \alpha = V_w$$

Similarly the tangential component of the absolute velocity V_1 of the leaving jet in the direction of motion u_1 is

$$-V_1 \cos \beta = (-V_{w1})$$

The negative sign has been introduced because as shown in Fig. 20.9 the jet leaves the vane in the direction opposite to the motion of the wheel.

Further if W is the weight of fluid striking the vanes per second then the mass of fluid striking the vane per second is

$$m = \rho Q = \left(\frac{W}{g}\right)$$

The tangential momentum of the fluid striking the vanes at inlet

$$= \left(\frac{WV_w}{g}\right)$$

As such the angular momentum at inlet

$$= \left(\frac{WV_w}{g}\right) R$$

Similarly the tangential momentum of the fluid at outlet of the vanes

$$= \left(-\frac{WV_{w_1}}{g}\right)$$

and hence the angular momentum at outlet

$$= \left(-\frac{WV_{w_1}}{g}\right) R_1$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

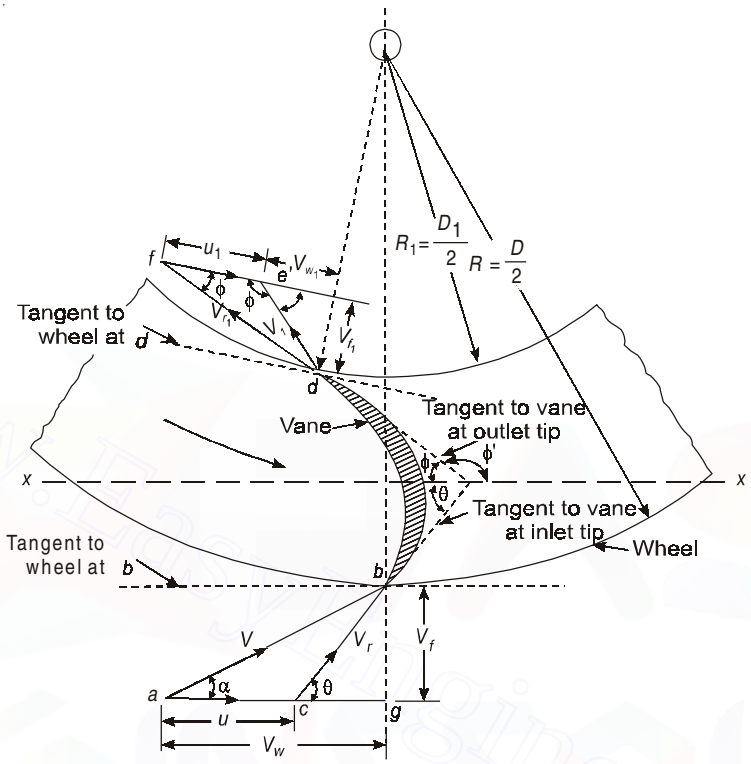


Figure 20.9 *Curved vane mounted radially on a wheel*

Therefore, applying Eq. 8.29 (Chapter 8) the torque T exerted by the fluid on the wheel to revolve it is obtained as

$$T = \left[\frac{WV_w}{g}R - \left(-\frac{WV_{w1}}{g}R_1\right)\right]$$

or
$$T = \frac{W}{g}(V_w R + V_{w1}R_1)$$

Further the work done on the wheel per second or the power delivered to the wheel

$$= \text{Torque} \times \text{angular velocity} = T \times \omega$$

$$= \frac{W}{g}(V_w R + V_{w1}R_1)\,\omega$$

But
$$u = \omega R \text{ and } u_1 = \omega R_1$$

$\therefore$ Work done on the wheel per second or the power delivered to the wheel

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{W}{g}(V_w u + V_{w1} u_1) \quad ...(20.37)$$

If the jet leaves in the direction as that of the motion of the wheel, V_{w1} will be positive (which corresponds to β being an obtuse angle) and then Eq. 20.37 becomes:

Work done on the wheel per second

$$= \frac{W}{g}(V_w u - V_{w1} u_1) \quad ...(20.38)$$

By combining Eqs 20.37 and 20.38 the general expression for the work done on the wheel per second may be written as

Work done on wheel per second

$$= \frac{W}{g}(V_w u \pm V_{w1} u_1) \quad ...(20.39)$$

For radial discharge at the outlet tip of the vane β = 90° and V_{w1} = 0. Then from Eq. 20.39, work done on wheel per second

$$= \frac{W}{g}(V_w u) \quad ...(20.40)$$

Further from Eq. 20.39, work done on the wheel per second per unit weight of fluid

$$= \frac{1}{g}(V_w u \pm V_{w1} u_1) \quad ...(20.41)$$

If there is no loss of energy, the work done on the wheel per second is also equal to the change in kinetic energy of the jet per second. Hence, the work done per second

$$= \frac{W}{2g}(V^2 - V_1^2) \quad ...(20.42)$$

Equating the work done represented by Eqs 20.39 and 20.42, we obtain

$$\frac{W}{g}(V_w u \pm V_{w1} u_1) = \frac{W}{2g}(V^2 - V_1^2)$$

or $$2\,(V_w u \pm V_{w1}\, u_1) = (V_2^2 - V_1^2) \quad ...(20.43)$$

The work done by the jet on the wheel will be the output of the system and the initial kinetic energy of the jet will be the input. Thus the efficiency of the system may be expressed as

$$\eta = \frac{W(V_w u \pm V_{w1} u_1)/g}{WV^2/2g}$$

or $$\eta = \frac{2(V_w u \pm V_{w1} u_1)}{V^2} \quad ...(20.44)$$

Similarly the efficiency of the system may also be expressed as

$$\eta = \frac{W(V^2 - V_1^2)\,2g}{WV^2/2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\eta = \frac{V^2 - V_1^2}{V^2} = \left[1 - \left(\frac{V_1}{V}\right)^2\right] \quad ...(20.45)$$

It follows from Eq. 20.45 that for a given angle α and the initial jet velocity V, the efficiency will be maximum when V_1 has a minimum value. However, V_1 cannot be zero as in that case the incoming jet will not move out of the vane. Further Eq. 20.44 indicates that the efficiency will be maximum when β is an acute angle so that there will be a positive sign between V_w and V_{w1}. In other words, it means that for the efficiency to be maximum, V_1 should be in the direction opposite to that of u_1. Thus, for maximum efficiency the velocities V_1, V_{r1} and u_1 should be colinear, so that $V_{r1} = (V_1 - u_1)$ and V_{r1} is in the direction of V_1. But this condition requires that $\phi = 0$ and $V_{w1} = V_1$, which is, however, not possible to accomplish in actual practice, because in practice ϕ will have some value other than zero. Hence, the above indicated condition for maximum efficiency is only theoretical, but it may, however, be stated that smaller is the value of ϕ, higher will be the efficiency.

ILLUSTRATIVE EXAMPLES

Example 20.1. *A jet of water 75 mm diameter having a velocity of 20 m/s, strikes normally a flat smooth plate. Determine the thrust on the plate (a) if the plate is at rest, (b) if the plate is moving in the same direction as the jet with a velocity of 5 m/s. Also find the work done per second on the plate in each case and the efficiency of the jet when the plate is moving.*

Solution

(a) From Eq. 20.1 the normal thrust on a stationary flat plate is given as

$$F = \left(\frac{waV^2}{g}\right)$$

Area of jet $$a = \frac{\pi}{4}\left(\frac{75}{1000}\right)^2 = 0.004\,42 \text{ m}^2$$

Velocity of jet $V = 20$ m/s

$$\therefore \quad F = \frac{9810 \times (0.00442) \times (20)^2}{9.81}$$

$$= 1\,768 \text{ N}$$

Since in this case plate is at rest, $u = 0$

$\therefore$ Work done $= (F \times u) = 0$

(b) From Eq. 20.7 the normal thrust on a moving flat plate is given as

$$F = \frac{wa(V-u)^2}{g}$$

$$= \frac{9810 \times (0.00442) \times (20-5)^2}{9.81} = 994.5 \text{ N}$$

Work done per second by the jet on the plate

$$= (F \times u)$$

$$= (994.5 \times 5) = 4\,972.5 \text{ N.m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus in this case, output

$$= 4\,972.5 \text{ N. m/s}$$

Input = Kinetic energy of the issuing jet

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

$$= \frac{1}{2}\left(\frac{9810 \times 0.00442}{9.81}\right)(20)^3$$

$$= 17\,680 \text{ N.m/s}$$

$$\therefore \quad \eta \text{ of jet} = \frac{4972.5}{17680} \times 100 = 28.13\%$$

Example 20.2. *A 50 mm diameter jet having a velocity of 25 m/s, strikes a flat plate, the normal of which is inclined at 30° to the axis of the jet. Calculate the normal force exerted on the plate (a) when the plate is stationary, (b) when the plate is moving with a velocity of 10 m/s in the direction of the jet, (c) when the plate is moving with a velocity of 8 m/s parallel to itself and in the direction of the normal to its surface. Find also the work done and the efficiency of the jet when the plate is moving.*

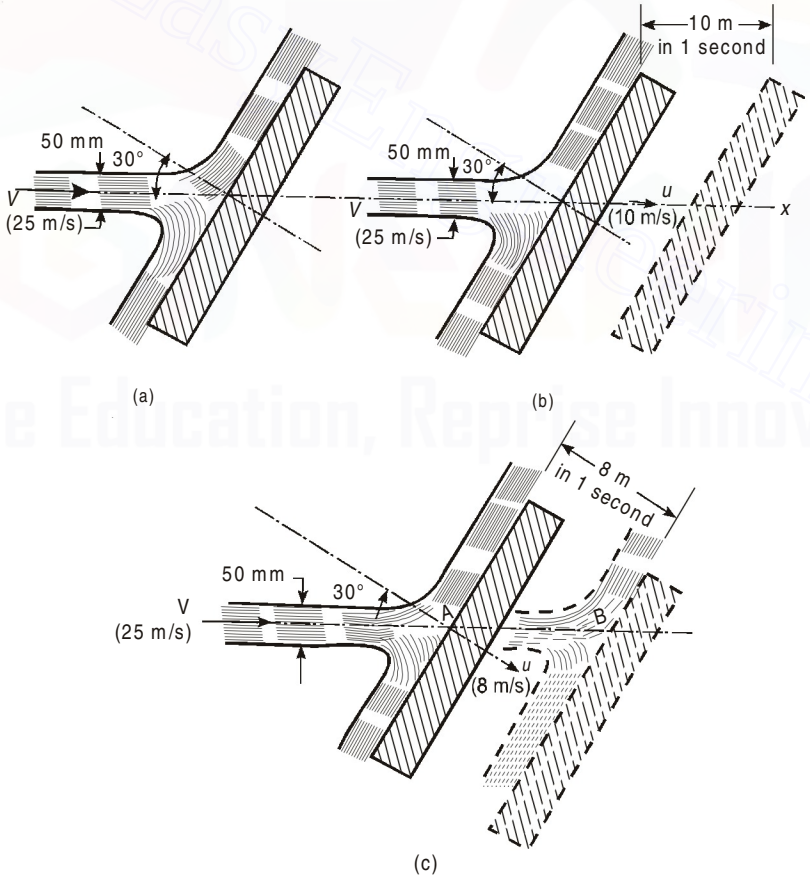


Figure Ex. 20.2

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

(a) From Eq. 20.2 the normal force on an inclined stationary flat plate is given as

$$F = \left(\frac{waV^2}{g}\right)\sin\theta$$

Area of jet $\alpha = \frac{\pi}{4}\left(\frac{50}{1000}\right)^2 = 0.00196 \text{ m}^2$

Velocity of jet $V = 25 \text{ m/s}$

Angle of inclination

$\theta = (90° - 30°) = 60°$

Thus $F = \frac{9810\times(0.00196)\times(25)^2}{9.81} \times \left(\frac{\sqrt{3}}{2}\right)$

$= 1060.88 \text{ N}$

(b) From Eq. 20.10 for an inclined plate moving in the same direction as the jet, the normal force is given as

$$F = \frac{wa(V-u)^2}{g}\sin\theta$$

$$= \frac{9810\times(0.00196)\times(25-10)^2}{9.81} \times \left(\frac{\sqrt{3}}{2}\right)$$

$= 381.92 \text{ N}$

The component of the force in the direction of the jet is

$F_x = F(\sin\theta)$

$= 381.92 \times \left(\frac{\sqrt{3}}{2}\right) = 330.75 \text{ N}$

Work done by the jet on the plate

$= F_x \times u$

$= 330.75 \times 10$

$= 3307.5 \text{ N.m/s}$

Energy supplied = kinetic energy of the issuing jet

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

$$= \frac{9810\times(0.00196)\times(25)^3}{2\times9.81}$$

$= 15\,312.5 \text{ N.m/s}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore$ η of the jet $= \dfrac{3307.5}{15312.5} \times 100 = 21.6\%$

(c) Figure Ex. 20.2 shows the arrangement of the plate for this case.

Mass of fluid issued by the jet per second

$$= \frac{waV}{g}$$

Mass of fluid used to extend the jet per second

$$= \frac{wa(AB)}{g} = \frac{wa}{g}\left(\frac{u}{\cos\theta}\right)$$

$\therefore$ Mass of fluid actually striking the plate per second

$$= \frac{wa}{g}\left(V - \frac{u}{\cos\theta}\right)$$

Initial component of velocity normal to the plate = $V \cos\theta$

Final velocity normal to the plate = u

Change of velocity normal to the plate = $(V \cos\theta - u)$

$\therefore$ Force normal to the plate

$$F = \frac{wa}{g}\left(V - \frac{u}{\cos\theta}\right)(V\cos\theta - u)$$

or

$$F = \frac{wa(V\cos\theta - u)^2}{g\cos\theta}$$

In this case $\theta = 30°$

Thus substituting the given values, we get

$$F = \frac{9810 \times (0.00196)(25\cos 30° - 8)^2}{9.81 \times (\cos 30°)}$$

$$= 421.73 \text{ N}$$

Work done per second

$$= (F \times u)$$

$$= (421.73 \times 8) = 3373.84 \text{ N.m}$$

Energy supplied per second

$$= \text{kinetic energy of the jet}$$

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

$$= \frac{9810 \times (0.00196) \times (25)^3}{2 \times 9.81}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 15\,312.5 \text{ N.m}$$

$$\therefore \quad \eta \text{ of the jet} = \frac{3373.84}{15312.5} = 22.03\ \%$$

Note. The problem can also be solved by adopting the principle of relative motion, in which case a velocity equal to (u/cos θ) may be applied to the entire system in the direction opposite to that of the jet so that the plate becomes stationary.

Example 20.3. *A jet of water 75 mm in diameter having velocity of 20 m/s strikes a series of the flat plates arranged around the periphery of a wheel such that each plate appears successively before the jet. If the plates are moving at a velocity of 5 m/s, compute the force exerted by the jet on the plate, the work done per second on the plate and the efficiency of the jet.*

Solution

From Eq. 20.8 the force exerted on the plate is given as

$$F = \frac{waV}{g}(V-u)$$

Area of jet $\quad \alpha = \frac{\pi}{4} \times \left(\frac{75}{1000}\right)^2 = 0.00442 \text{ m}^2$

Velocity of jet $\quad V = 20 \text{ m/s}$

Thus $\quad F = \frac{9810 \times (0.00442) \times 20 \times (20-5)}{9.81}$

$$= 1\,326 \text{ N}$$

Work done $= (F \times u)$

$$= (1\,326 \times 5) = 6\,630 \text{ N.m/s}$$

From Eq. 20.9 the efficiency of the jet is given as

$$\eta = \frac{2u(V-u)}{V^2}$$

$$= \frac{2 \times 5(20-5)}{(20)^2} \times 100 = 37.5\%$$

Example 20.4. *A metal plate of 10 mm thickness and 200 mm square is hung so that it can swing freely about the upper horizontal edge. A horizontal jet of water of 20 mm diameter impinges with its axis perpendicular and 50 mm below the edge of the hinge, and keeps it steadily inclined at 30° to the vertical. Find the velocity of the jet if the specific weight of the metal is 75.54 kN/m³.*

Solution

Weight of the plate

$$= \frac{(10 \times 200 \times 200) \times 75.54 \times 10^3}{(1000)^3}$$

$$= 30.22 \text{ N}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Area of the jet,

$$\alpha = \frac{\pi}{4} \times \left(\frac{20}{1000}\right)^2$$
$$= 0.000\ 314\ \text{m}^2$$

Let the velocity of the jet be V m/s, then from Eq. 20.2 the force exerted by the jet in the direction normal to the plate is given as

$$F = \left(\frac{waV^2}{g}\right) \sin\theta$$

As shown in Fig. Ex. 20.4 $\theta = 60°$, thus by substitution, we get

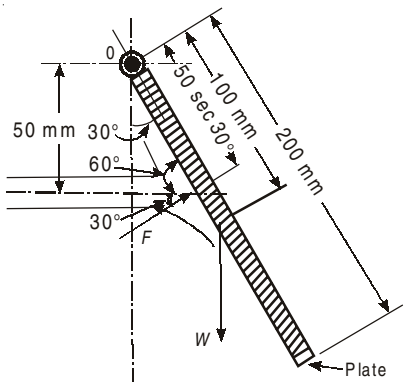


Figure Ex. 20.4

$$F = \frac{9810 \times (0.000314) \times (\sqrt{3})}{9.81 \times 2} V^2$$
$$= 0.272\ V^2$$

Taking moments of all the forces about the hinge 0, we have

$$30.22 \times \left(\frac{100}{1000}\right)(\sin 30°) = 0.272\ V^2 \left(\frac{50}{1000}\right)(\sec 30°)$$

$\therefore$ Velocity of jet $V = 9.81$ m/s

Example 20.5. *A jet of water moving at 20 m/s impinges on a symmetrical curved vane shaped to deflect the jet through 120° (that is the vane angles θ and ϕ are each equal to 30°). If the vane is moving at 5 m/s, find the angle of the jet so that there is no shock at inlet. Also determine the absolute velocity of exit in magnitude and direction, and the work done.*

Solution

From the velocity triangle at inlet, as shown in Fig. Ex. 20.5, applying the sine rule, we have

$$\frac{V}{\sin(180° - 30°)} = \frac{u}{\sin(30° - \alpha)} = \frac{V_r}{\sin\alpha}$$

Also $V = 20$ m/s , $u = 5$ m/s

Thus considering the first two terms of the above equation, we have

$$\frac{20}{\sin 150°} = \frac{5}{\sin\ (30° - \alpha)}$$

or $\sin\ (30° - \alpha) = 0.125$

or $\alpha = 22°49'$

Now considering the first and the third terms of the above equation, we have

$$\frac{20}{\sin 150°} = \frac{V_r}{\sin\ (22°49')}$$

$\therefore$ $V_r = 15.51$ m/s

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

It is assumed that the relative velocity of flow remains constant, that is,

$$V_{r1} = V_r = 15.51 \text{ m/s}$$

From the outlet velocity triangle applying the cosine rule, we have

$$V_1^2 = u_1^2 + V_{r1}^2 - 2u_1 V_{r1} (\cos 30°)$$

Since $u_1 = u = 5$ m/s

$$V_1^2 = (5)^2 + (15.51)^2 - 2(5 \times 15.51)(0.866)$$

$$\therefore \quad V_1 = 11.46 \text{ m/s}$$

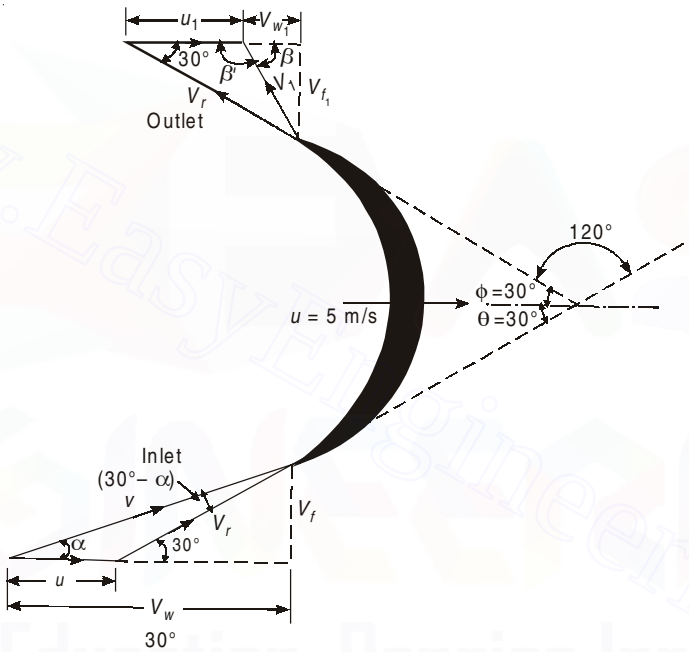


Figure Ex. 20.5

Again from the outlet velocity triangle

$$\tan \beta = \frac{V_{r_1} (\sin 30°)}{V_{r_1} (\cos 30°) - u_1}$$

or $$\tan \beta = \frac{15.51 \times 0.5}{(15.51 \times 0.866) - 5} = 0.9197$$

$$\therefore \quad \beta = 42° 36'$$

and $$\beta' = (180° - \beta) = 137° 24'$$

From Eq. 20.41, work done per unit weight of fluid

$$= \frac{1}{g} [V_w u \pm V_{w1} u_1]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_w = V(\cos\alpha)$$
$$= 20(\cos 22°49') = 18.44 \text{ m/s}$$
$$V_{w1} = V_1(\cos\beta)$$
$$= 11.45(\cos 42°36') = 8.43 \text{ m/s}$$

Since the direction of V_{w1} is opposite to that of the motion of vane and $u = u_1 = 5$ m/s. Thus work done

$$= \frac{1}{g}[V_w + V_{w1}]u$$
$$= \frac{5}{9.81}[18.44 + 8.43]$$
$$= 13.70 \text{ N.m/N}$$

Example 20.6. *A jet of water 50 mm in diameter having a velocity of 20 m/s, impinges tangentially on a series of vanes, which when stationary deflect the jet through an angle of 120°. Calculate the magnitude of the resultant force on the vanes when they are (a) stationary, (b) moving with a velocity of 10 m/s in the same direction as the jet. Also determine the work done per second and the efficiency in case (b).*

Solution

In this case the vanes may be set in either of the two ways as shown in Fig. Ex. 20.6.

(a) (i) Let the axis of symmetry of the vane be horizontal as shown in Fig. Ex. 20.6 (a). In this case the tangents at inlet and outlet will make an angle of 30° with the horizontal. Then from Eq. 20.19, we have

$$F_x = 2\left(\frac{wav^2}{g}\right)\cos\theta$$

Area of jet $$\alpha = \frac{\pi}{4}\left(\frac{50}{1000}\right)^2 = 0.00196 \text{ m}^2$$

Thus $$F_x = \frac{2\times 9810\times 0.00196\times(20)^2}{9.81} \times \left(\frac{\sqrt{3}}{2}\right)$$
$$= 1357.93 \text{ N}$$

Further $$F_y = 0$$

Thus the resultant force acting on the vane is

$$F = F_x = 1357.93 \text{ N}$$

which is acting in the direction of horizontal axis or it is inclined at an angle of 30° to the direction of the jet.

(ii) Let the tangent at the inlet tip of the vane be horizontal, that is, let the axis of symmetry be inclined at an angle of 30° to the horizontal as shown in Fig. Ex. 20.6 (b).

Thus in this case the tangent at outlet tip of the vane will make an angle of 60° with the horizontal. Hence in this case $\theta = 0°$ and $\phi = 60°$.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

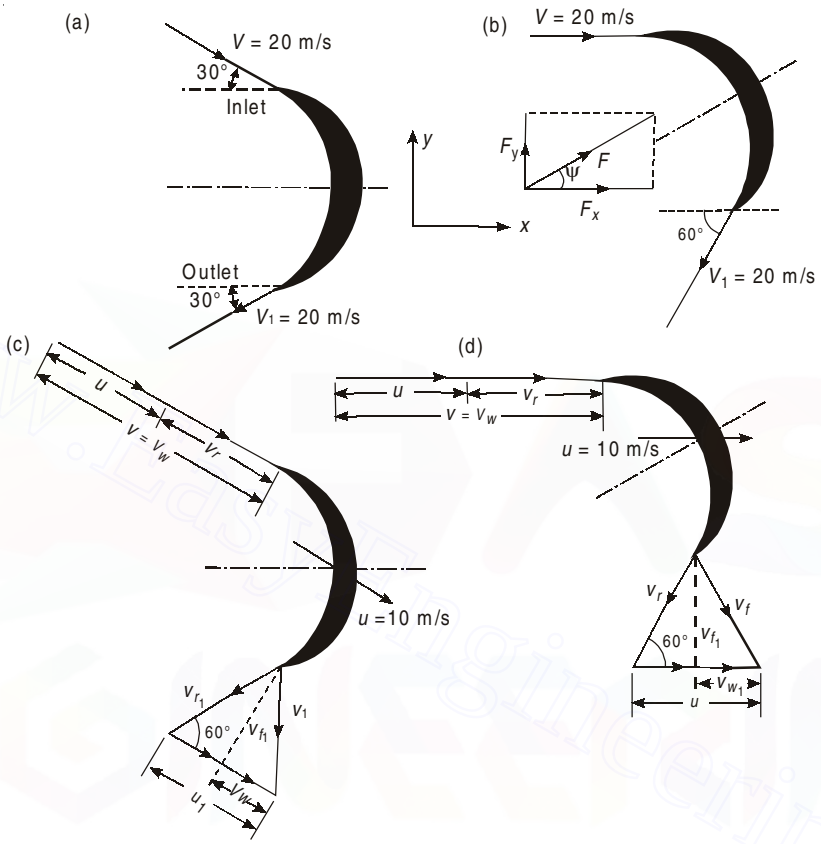


Figure Ex. 20.6

From Eq. 20.16, we have

$$F_x = \frac{waV^2}{g}[\cos\theta + \cos\phi]$$

$$= \frac{9810\times(0.00196)\times(20)^2}{9.81} \times (1 + \cos 60°)$$

$$= 1176 \text{ N}$$

Again from Eq. 20.17, we have

$$F_y = \frac{waV^2}{g}[\sin\theta - \sin\phi]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{waV^2}{g}[0 - \sin 60°]$$

$$= \frac{9810 \times (0.00196) \times (20)^2}{9.81} \times \left(-\frac{\sqrt{3}}{2}\right)$$

$$= -678.96 \text{ N}$$

The negative sign for F_y is, however, not significant and the same may be ignored.

The resultant force exerted by the jet on the vane is

$$F = \sqrt{F_x^2 + F_y^2}$$

$$= \sqrt{(1176)^2 + (678.96)^2} = 1357.93 \text{ N}$$

If ψ is the angle made by the resultant force with the direction of the jet, then

$$\tan \psi = \frac{F_y}{F_x}$$

$$= \frac{678.96}{1176} = 0.5773$$

$$\therefore \quad \psi = 30°$$

(b) When the vanes are moving in the same direction as the jet then the velocity triangles at the inlet and outlet tips of the vanes for both the settings of vanes are shown in Fig. Ex. 20.6 (c) and (d). Since in both the settings of vanes the velocity triangles are exactly same, the case shown in Fig. Ex. 20.6 (d) only is discussed. From the velocity triangle at inlet (which is a straight line), we have

$$V_r = (V - u) = (20 - 10) = 10 \text{ m/s}$$

$$V_w = V = 20 \text{ m/s}$$

Assuming that the relative velocity remains constant, we have

$$V_r = V_{r1} = 10 \text{ m/s}$$

Moreover, at the outlet tip of the vane

$$\phi = 60°, u_1 = u = 10 \text{ m/s}$$

and $V_{r1} = 10$ m/s

hence the outlet velocity triangle is an equilateral triangle.

Thus $V_1 = 10$ m/s

and $V_{w1} = V_1 (\cos 60°) = 5$ m/s

It is assumed that the vanes are so arranged that the vanes appear successively before the jet, hence the mass of water striking the vanes per second

$$= \frac{waV}{g} = \frac{9810 \times (0.00196) \times (20)}{9.81} = 39.2 \text{ kg}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus the component of the force in the x-direction is

$$F_x = \frac{waV}{g}[V_w - V_{w1}]$$

$$= 39.2\,(20 - 5) = 588 \text{ N}$$

Similarly the component of force in the y-direction is

$$F_y = \frac{waV}{g}[0 - V_1 \sin 60°]$$

$$= 39.2\left(-10 \times \frac{\sqrt{3}}{2}\right) = -339.48 \text{ N}$$

The negative sign for F_y is, however, not significant and the same may be ignored.
The resultant force exerted by the jet on the vane is

$$F = \sqrt{F_x^2 + F_y^2}$$

$$= \sqrt{(588)^2 + (339.48)^2} = 678.96 \text{ N}$$

If ψ is the angle between the resultant force and the direction of jet, then

$$\tan \psi = \frac{F_y}{F_x} = \frac{339.48}{588} = 0.5773$$

$$\therefore \quad \psi = 30°$$

Hence the resultant force is inclined at an angle of 30° to the direction of the jet.
From Eq. 20.39 the work done per second

$$= \frac{waV}{g}(V_w u - V_{w1} u_1)$$

$$= 39.2\,(20 - 5) \times 10 = 5\,880 \text{ N.m}$$

From Eq. 20.44 the efficiency is given as

$$\eta = \frac{2(V_w u - V_{w1} u_1)}{V^2}$$

$$= \frac{2(20 - 5) \times 10}{(20)^2} \times 100 = 75\%$$

Example 20.7. *A jet of water having a velocity of 45 m/s impinges without shock a series of vanes moving at 15 m/s, the direction of motion of the vanes being inclined at 20° to that of the jet. The relative velocity at outlet is 0.9 of that at inlet, and the absolute velocity of the water at exit is to be normal to motion of the vanes. Find: (a) vane angles at entrance and exit; (b) work done on vanes per unit weight of water supplied by the jet; and (c) the hydraulic efficiency.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

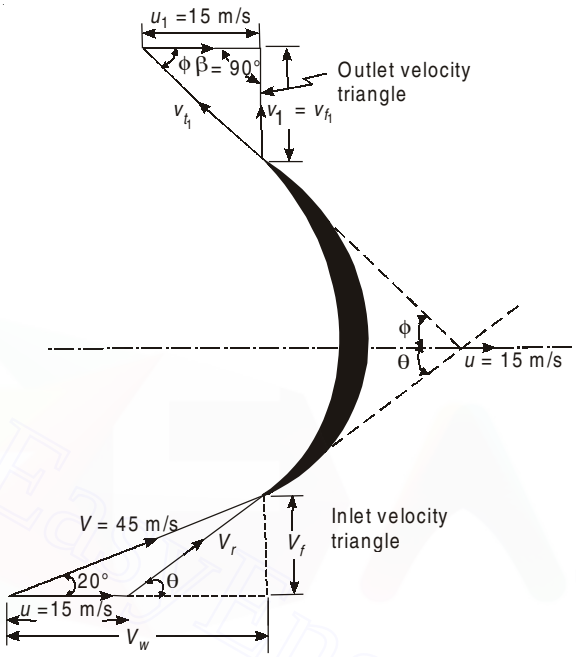


Figure Ex. 20.7

Applying the cosine rule at the inlet velocity triangle, we have

$$V_r^2 = (45)^2 + (15)^2 - 2(45 \times 15)(\cos 20°)$$

$$\therefore \quad V_r = 31.33 \text{ m/s}$$

Thus $V_{r1} = 0.9\, V_r = (0.9 \times 31.33) = 28.2$ m/s

From inlet velocity triangle

$$\tan \theta = \frac{45 (\sin 20°)}{45 (\cos 20°) - 15} = 0.5641$$

$$\therefore \quad \theta = 29° 26'$$

From outlet velocity triangle since

$$\beta = 90°$$

$$\cos \phi = \frac{u_1}{V_{r_1}} = \frac{15}{28.2} = 0.531\,9$$

$$\therefore \quad \phi = 57° 52'$$

i.e., the vane angles at entrance and exit are 29°26′ and 57°52′ respectively.

Work done on the vane per unit weight of water

$$= \frac{1}{g}(V_w u) \qquad \text{(since } V_{w1} = 0\text{)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{1}{9.81} \times (45 \cos 20°) \times 15$$

$$= 64.66 \text{ N.m/N}$$

Hydraulic efficiency

$$\eta_h = \frac{\text{Work done/second}}{\text{K.E of jet/second}}$$

K.E. of jet per unit weight of water per second

$$= \frac{(45)^2}{2 \times 9.81} = 103.21 \text{ N.m/N}$$

$$\therefore \quad \eta_h = \frac{64.66}{103.21} \times 100 = 62.65\ \%$$

Example 20.8. *A jet discharges 0.15 m^3/s of water with velocity of 70 m/s impinges without shock on a series of curved vanes which move in the same direction as the jet. The shape of each vane is such that it would deflect the jet through an angle of 150°. Surface friction reduces the relative velocity by 8 percent as the water passes across the vanes and there is a further windage loss equivalent to* $\left(\frac{0.5\,u^2}{2g}\right)$ *N.m per N of water, u being the vane velocity. Find:*

(a) the velocity of the vanes corresponding to maximum efficiency;

(b) the value of this efficiency;

(c) the corresponding force on the vanes in, and at right angles to the direction of their motion ; and

(d) the power of this arrangement.

Solution

Let u be the required velocity of the vanes, then from Fig. Ex. 20.8,

$$V_r = (V - u) = (70 - u)$$

and $$V_{r1} = 0.92\, V_r = 0.92\,(V - u)$$

Further $$V_w = V = 70 \text{ m/s}$$

and $$V_{w1} = [u - V_{r1} (\cos 30°)]$$

$$= \left[u - 0.92(V - u)\left(\sqrt{3}/2\right)\right]$$

$\therefore$ Work done by water per second

$$= \frac{W}{g}[V_w u - V_{w1} u_1]$$

$$= \frac{W}{g}\left[V - \left\{u - 0.92(V - u)\left(\sqrt{3/2}\right)\right\}\right]u$$

$$= \frac{W}{g}[1.797\,(V - u)u\,]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Therefore useful work done per second

$$= \frac{W}{g}[1.797(V-u)u] - (0.5)\frac{Wu^2}{2g}$$

Kinetic energy of jet

$$= \frac{WV^2}{2g}$$

$\therefore$ Efficiency $\eta = \dfrac{\text{Useful work done/second}}{\text{Kinetic energy of jet}}$

$$= \frac{2[1.797(V-u)u] - 0.5u^2}{V^2}$$

$$= 3.594\left(\frac{u}{V} - \frac{u^2}{V^2}\right) - 0.5\frac{u^2}{V^2}$$

$$= 3.594\left(\frac{u}{V}\right) - 4.094\left(\frac{u}{V}\right)^2$$

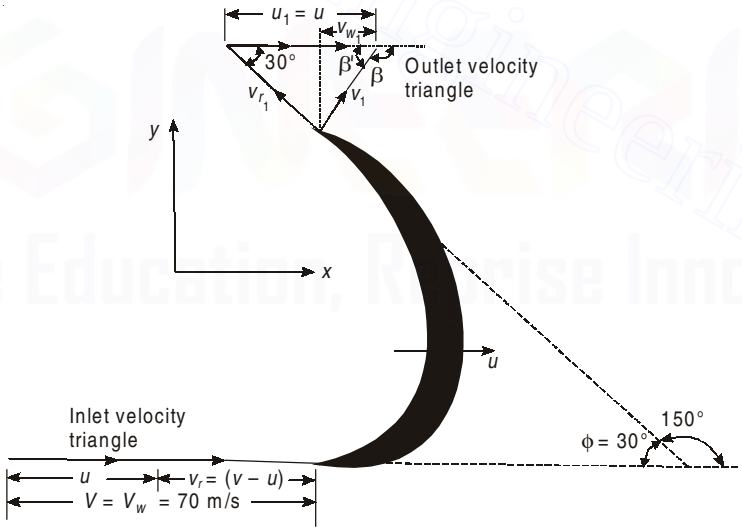


Figure Ex. 20.8

Now if $\dfrac{u}{V} = n$, then

$$\eta = n\,(3.594 - 4.094\,n)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For maximum efficiency

$$\frac{d\eta}{dn} = 0$$

Thus $3.594 - 2(4.094)n = 0$

or $$n = \frac{u}{V} = \frac{3.594}{8.188} = 0.439$$

∴ For maximum efficiency

$$u = (0.439 \times 70) = 30.73 \text{ m/s}$$

Maximum efficiency

$$\eta_{max} = 0.439\,[3.594 - (4.094)\,(0.439)]$$
$$= 0.789 \text{ or } 78.9\%$$

Force on the vane in the direction of motion of the vane is

$$F_x = \frac{W}{g}[V_w - V_{w1}]$$

$$= \frac{9810 \times 0.15}{9.81}\left[70 - 30.73 + 0.92\,(70 - 30.73)\left(\frac{\sqrt{3}}{2}\right)\right.$$

$$= 10\,583.7 \text{ N}$$

Force on the vane at right angles to the direction of the motion of the vane is

$$F_y = \frac{W}{g}(V_{f1} - V_f) = \frac{W}{g}V_{f1}, \text{ since } V_f = 0$$

or $$F_y = \frac{9810 \times 0.15}{9.81}[V_{r1} \sin 30°]$$

$$= \frac{9810 \times 0.15}{9.81}\left[0.92(70 - 30.73) \times \frac{1}{2}\right]$$

$$= 2\,709.6 \text{ N}$$

The power of the arrangement

$$= \text{Useful work done per second}$$

$$= \frac{W}{g}[1.797\,(V - u)u] - 0.5\frac{Wu^2}{2g}$$

$$= \frac{9810 \times 0.15}{9.81}\left[1.797(70 - 30.73) \times 30.73 - \frac{0.5}{2} \times (30.73)^2\right]$$

$$= 289\,872 \text{ W} = 289.872 \text{ kW}$$

Example 20.9. *The rotor of an inward flow hydraulic turbine has a diameter over the tips of the blades of 1m. The diameter at the bottom of the blades is 0.6 m. The speed is 300 r.p.m. The water is supplied through fixed vanes at 10° to the tangent to the outer circumference of the rotor, the velocity of water being 10 m/s. Find the blade angles at entry and exit, so that the water may enter and leave the moving blades without shock, the water leaves the blades with the velocity entirely radial and equal to 3.5 m/s. Also find the velocity of the water relative to the blades at the exit.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

With reference to Fig. Ex. 20.9 the following data are given

$\alpha = 10°$, $R = 0.5$ m, $R_1 = 0.3$ m

$V = 10$ m/s, $\beta = 90°$, $V_1 = V_{f1} = 3.5$ m/s

$N = 300$ r.p.m.

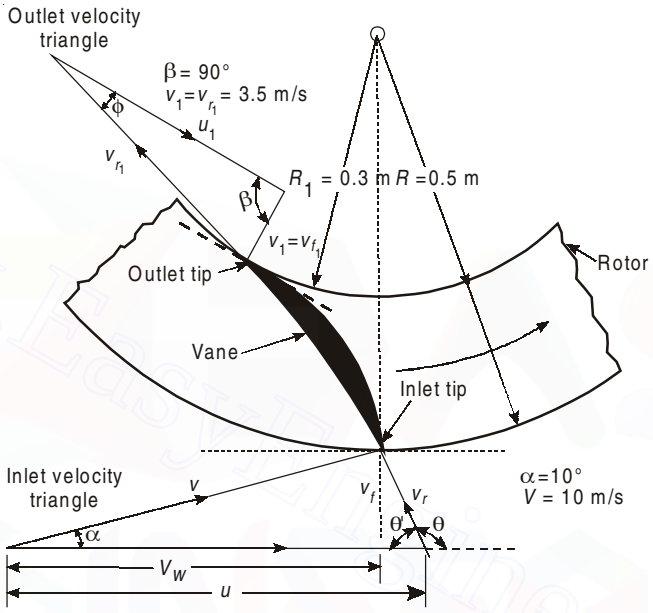


Figure Ex. 20.9

Now $$u = \frac{2\pi NR}{60}$$

$$= \frac{2\times\pi\times300\times0.5}{60} = 15.708 \text{ m/s}$$

and $$u_1 = \frac{2\pi NR_1}{60}$$

$$= \frac{2\times\pi\times300\times0.3}{60} = 9.425 \text{ m/s}$$

From the outlet velocity triangle, we have

$$\tan\phi = \frac{V_1}{u_1} = \frac{3.5}{9.425} = 0.371\,4$$

$\therefore$ $$\phi = 20.38°$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Again $V_{r1} = V_1 \text{ cosec } \phi$

$= 3.5 \text{ (cosec } 20.38°) = 10.05 \text{ m/s}$

From the velocity triangle at inlet tip of the blade, we have

$V_w = V \cos \alpha = 10 (\cos 10°) = 9.848 \text{ m/s}$

$V_f = V \sin \alpha = 10 (\sin 10°) = 1.736 \text{ m/s}$

Now $$\tan \theta' = \frac{V_f}{u - V_w} = \frac{1.736}{15.708 - 9.848} = 0.2962$$

or $\theta' = 16.50°$

$\therefore \quad \theta = (180° - \theta') = (180° - 16.50°) = 163.50°$

Example 20.10. *A jet of water 15 mm in diameter impinges on a fixed cup which deflects the jet by 165° as shown in Fig. Ex. 20.10. If the reaction of the cup was found to be 26.5 N, when the discharge from the jet was 980 N per minute, calculate the ratio of (a) actual to theoretical force of the jet, (b) velocity at outlet to velocity at inlet.*

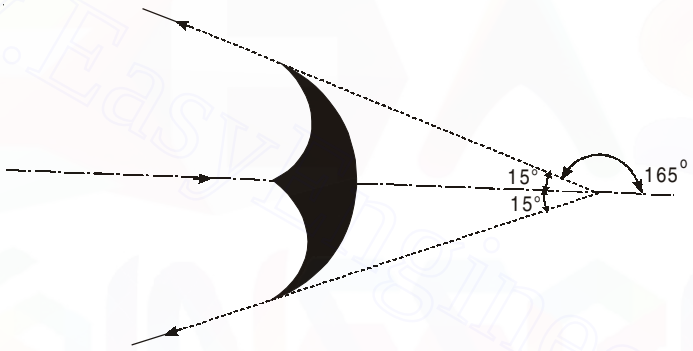


Figure Ex. 20.10

Solution

$$\text{Discharge from jet} = \frac{980}{9810 \times 60} = 0.001\,66 \text{ m}^3/\text{s}$$

$$\therefore \quad \text{Velocity of jet} = \frac{0.00166 \times 4 \times (1000)^2}{\pi \times (15)^2} = 9.39 \text{ m/s}$$

For theoretical force it is assumed that the velocity is not reduced as the water passes round each cup. Change of velocity in direction of jet = 9.39 (1 – cos 165°) = 18.46 m/s

Hence theoretical force

= (mass flowing/second) × (change of velocity)

$$= \frac{980}{9.81 \times 60} \times 18.46 = 30.74 \text{ N}$$

$$\therefore \quad \frac{\text{Actual force}}{\text{Theoretical force}} = \frac{26.5}{30.74} = 0.862$$

If V_1 is the actual velocity at the outlet, then

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\text{Actual force} = (\text{Mass flowing/second}) \times (\text{change of velocity})$$

or $$26.5 = \frac{980}{9.81\times 60}(9.39 - V_1 \cos 165°)$$

Solving for V_1, we get

$$V_1 = 6.76 \text{ m/s}$$

$$\frac{\text{Outlet Velocity}}{\text{Inlet Velocity}} = \frac{6.76}{9.39} = 0.72$$

SUMMARY OF MAIN POINTS

1. The force exerted by a jet of water of cross-sectional area a and velocity V on stationary flat plate in the direction of the jet is given by

(i) For the plate held normal to the jet

$$F = \frac{w\,aV^2}{g} = \frac{wQV}{g} \quad (\text{since } Q = aV)$$

(ii) For the plate held inclined at an angle θ to the jet

$$F = \frac{w\,aV^2}{g}\sin\theta = \frac{wQV}{g}\sin\theta$$

The division of discharge Q that results in this case is given as

$$Q_1 = \frac{Q}{2}(1+\cos\theta)$$

$$Q_2 = \frac{Q}{2}(1-\cos\theta)$$

2. The force exerted by a jet of water of cross-sectional area a and velocity V on a flat plate moving with velocity u in the dirction of the jet is given by

(i) For the plate held normal to the jet

$$F = \frac{wa(V-u)^2}{g}$$

(ii) For the plate held inctined at an angle θ to the jet

F = 968a jet of water of cross-sectional area a and velocity V, striking the wheel tangentially at its lower edge is given by

$$F = \frac{waV}{g}(V-u)$$

The work done by the jet per second will be the output of the jet. Thus output of the jet = $\frac{waV}{g}(V-u)u$

The efficiency of the wheel is given by

$$\eta = \frac{2u(V-u)}{V^2}$$

For maximum efficiency $u = \frac{V}{2}$, and η_{max} = 0.5 or 50%.

4. The force exerted by a jet of water of cross-sectional area a and velocity V on a stationary curved vane in the direction of the jet is given by

(i) For the jet striking a symmetrical vane at the centre

$$F = \frac{waV^2}{g}(1+\cos\theta)$$

where

θ = angle of the vane tips.

(ii) For the jet striking an unsymmetrical vane tangentially at one of the tips

$$F_x = \frac{waV^2}{g}(\cos\theta + \cos\phi)$$

$$F_y = \frac{waV^2}{g}(\sin\theta - \sin\phi)$$

The resultant thrust extered by the jet will be $F = \sqrt{F_x^2 + F_y^2}$ and its inclination with the x-axis is

$$\alpha = \tan^{-1}\left(\frac{F_y}{F_x}\right)$$

where

θ = angle of the vane at the inlet tip; and

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ϕ = angle of the vane at the outlet tip.

For semicircular vane $\theta = \phi = 0$, and hence $F = F_x$

$$= \frac{2waV^2}{g} \text{ ; and } F_y = 0$$

5. The force exrted by a jet of water of cross-sectional area a and velocity V on a curved vane moving with velocity u in the direction of the jet is given by

(i) For the jet striking a symmetrical vane at the centre

$$F = \frac{wa(V-u)^2}{g}(1+\cos\theta)$$

The work done by the jet on the vane per second

$$= F \times u$$

$$= \frac{wa(V-u)^2}{g}(1+\cos\theta)u$$

The efficiency of the jet is given by

$$\eta = \frac{2u(V-u)^2(1+\cos\theta)}{V^3}$$

$$\eta_{max} = \frac{8}{27}(1+\cos\theta)$$

For a series of symmetrical curved vanes mounted on the periphery of a wheel the force exerted by the jet is given by

$$F = \frac{waV(V-u)}{g}(1+\cos\theta)$$

The work done by the jet per second will be the output of the jet. Thus output of the jet = $\frac{waV(V-u)}{g}(1+\cos\theta)u$ .The efficiency of the wheel is given by

$$\eta = \frac{2u(V-u)(1+\cos\theta)}{V^2}$$

$$\eta_{max} = \frac{1}{2}(1+\cos\theta)$$

(ii) For the jet striking an unsymmetrical vane tangentially at one of the tips

$$F_x = \frac{waV_r}{g}\left(V_w \pm V_{w_1}\right)$$

Work done per second = $\frac{waV_r}{g}\left(V_w \pm V_{w_1}\right)u$

6. For a series of unsymmetrical curved vanes mounted radially on the periphery of a wheel the work done by the jet of water per second

$$= \frac{W}{g}\left(V_w u \pm V_{w_1} u_1\right)$$

The efficiency of the system in this case is given by

$$\eta = \frac{2\left(V_w u \pm V_{w_1} u_1\right)}{V^2}$$

where

V_w = velocity of whirl at inlet;
V_{w1} = velocity of whirl at outlet;
u = tangential velocity of vane at inlet;
u_1 = tangential velocity of vane at outlet; and
V = absolute velocity of jet at inlet.

PROBLEMS

20.1 A jet of water 50 mm in diameter having a velocity of 20 m/s, strikes normally a flat smooth plate. Determine the thrust on the plate (a) if the plate is at rest; (b) if the plate is moving in the same direction as the jet with a velocity of 8 m/s. Also find the work done per second on the plate and the efficiency of the jet when the plate is moving.

[*Ans.* (a) 784 N {79.92 kg (f)}; (b) 282.24 N {28.77 kg(f)}, 2 257.92 N.m {230.16 kg (f) – m}, 28.80%]

20.2 Show that the efficiency of a free jet striking normally as series of flat plates mounted on the periphery of a wheel never exceeds 50%.

20.3 A 75 mm diameter jet having a velocity of 30 m/s strikes a flat plate, the normal of which is inclined at 45° to the axis of the jet. Find the normal pressure on the plate (a) when the plate is stationary; (b) when the plate is moving with a velocity of 15 m/s in the direction of the jet and away from the jet. Also determine the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

power and the efficiency of the jet when the plate is moving.

[*Ans.* (a) 2812.87 N {286.7 kg(f)}; (b) 703.22 N {71.68 kg (f)}, 7 458.77 W (10.14 h.p.), 12.5%]

20.4 A rectangular plate, weight 60 N is suspended vertically by a hinge on the top horizontal edge. The c.g. of the plate is 100 mm from the hinge. A horizontal jet of water 20 mm diameter, whose axis is 150 mm below the hinge impinges normally on the plate with velocity 5 m/s. Find the horizontal force, applied at the c.g. to maintain the plate in its vertical position. Find the alteration of the velocity of the jet if the plate is deflected through 30° and the same force continues to act at the c.g. of the plate.

[*Ans.* 11.78 N{1.2 kg (f)}, 4.24 m/s]

20.5 A jet of water 75 mm diameter and with velocity of 20 m/s flows tangentially on to a stationary vane which deflects the water through 120. What is the magnitude and direction of the resultant force on the vane? If the jet flows on to a series of vanes moving in the direction of the jet with velocity 12 m/s, find: (a) the force on the system of vanes in the direction of motion; (b) the work done per second; and (c) the efficiency.

[*Ans.* (a) 3 062.27 N {312.16 kg (f)} at 30° to jet; (b) 1060.8 N {108.13 kg (f)}; 12 729.6 N.m {1297.6 kg (f) – m}; (c) 72%]

20.6 A jet of water moving at 15 m/s impinges on a symmetrical concave vane shaped to deflect the jet through 140°. If the vane is moving at 6 m/s, find the angle of the jet so that there is no shock at inlet. Also determine the absolute velocity of exit in magnitude and direction and the work done per unit weight of water.

[*Ans.* 12.14°; 4.13 m/s, 49.8°; 104 N.m/N {10.6 kg(f) - m/kg (f)}]

20.7 A jet of water having a velocity of 60 m/s impinges without shock on a series of vanes moving at 30 m/s. The direction of motion of vanes being inclined at 20° to that of the jet. The angle between the inlet and outlet relative velocities is 120° and the ratio between these velocities is 0°9. Find: (a) the angle θ at the entry edge of the vane; (b) the work done by the vanes in N.m/N of water supplied by the jet ; (c) the hydraulic efficiency.

[*Ans.* (a) 37.9°; (b) 1627.55 N.m/N ; (c) 90.42%]

20.8 A jet having a velocity V strikes a single curved vane moving in the same direction as the jet with velocity u, so that the velocity of the jet relative to the vane is $(V - u)$. The vane causes the jet to be reversed in direction. Show that the maximum efficiency is obtained when $V = 3u$ and that this maximum efficiency is slightly less than 0.6.

20.9 Water enters a wheel consisting of curved vanes, outside diameter 1 m, inside diameter 0.5 m. The flow is inwards from the circumference. The supply jet is at 30° to the tangent to the outside circle with a velocity of 40 m/s. The water is to leave the wheel at 3.5 m/s at 120° to the tangent to the inner circle. Draw the velocity triangles for the inlet and outlet and find suitable blade angles if the wheel runs at 360 r.p.m. [*Ans.* 51.7°; 15.2°]

20.10 A wheel having radial blades is of 0.6 m radius at the outer tip of the blades and 0.3 m at the inner tip. Water enters the blades at the outer tip with a velocity of 30 m/s at the angle of 30° to tangent and leaves the blades with a velocity of flow of 4 m/s. The blades have an angle of 40° at entrance and 35° at exit. Find the work done per newton of water entering the wheel, the speed of the wheel, and the efficiency.

[*Ans.* 22 N.m/N ; 129 r.p.m ; 48%]

20.11 Water impinges on a smooth moving curved vane under the following conditions: Velocity of vane = 15 m/s; direction of jet at entry to vane = 30° to the direction of the vane; velocity of jet= 30 m/s. At the exit side the vane makes an angle of 170° with the direction of motion of the vane (i.e., it is curved backwards). Find: (a) the vane angle at entry so that the water strikes the vanes tangentially; (b) the absolute velocity of the water after leaving the vane and its direction ; (c) the work done on the vanes per unit weight of water supplied.

[*Ans.* (a) 53° 48′ ; (b) 4.62 m/s at 135° 42′ to tangent at exit ; (c) 44.79 N.m/N]

20.12 A free jet of water of velocity V strikes against a series of curved semi-circular vanes tangentially. The vanes are moving in the direction of the jet with a velocity equal to 0.6V. Assuming the relative velocity of water is reduced by 10% by moving over the vanes, show that the vanes have an efficiency of 91.20%.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

20.13 The following data refer to a water wheel comprising a series of flat plates mounted on the periphery of the wheel radially. Diameter of wheel (upto centre of plates) = 1.22 m, diameter of free jet = 168 mm. Head at the beginning of the nozzle = 18.3 m, coefficient of velocity of the nozzle = 0.97, and speed of the wheel = 120 r.p.m. Determine (a) efficiency of the water wheel, and (b) power delivered to the water wheel.

[*Ans.* (a) 48.6%; (b) 68.82 kW (93.54 h.p.)]

20.14 A water wheel has a number of hemispherical vanes equally spaced on its periphery. A jet of diameter 300 mm having a velocity of 16 m/s impinges on the vanes which have a tangential velocity of 6 m/s. Determine the work done on the wheel.

[*Ans.* 135.717 kN. m/s {13 835 kg(f)-m/s}]

20.15 Water flows over series of curved vanes of a hydraulic turbine wheel, the diameter of which between inlet tips of vanes is 2 m and that between outlet tips is 1 m. The wheel rotates at 240 r.p.m. Jet of water enters at an angle of 30° to the tangent to wheel at inlet with a velocity of 40 m/s and leaves with a velocity of 10 m/s at an angle of 30° to the tangent to wheel at outlet tip. Find (a) vane angles at entry and exit; (b) work done on the wheel per newton of water; (c) hydraulic efficiency of wheel; (d) power developed by the wheel when the discharge flowing through it is 0.3 m^3/s.

[*Ans.* (a) 64° 34′ , 52° ; (b) 77.65 N.m/N;
(c) 95.2%; (d) 228.52 kW (310.6 h.p.)]

20.16 A 50 mm diameter jet having a discharge of 50 litres per second impinges without shock on a series of vanes which move in the same direction as the jet at 10 m/s. The shape of each vane is such that if stationary it would deflect the jet through an angle of 135°. Due to frictional resistance the relative velocity at outlet is 0.85 of that at inlet. Determine : (a) the magnitude and direction of the resultant force on vanes ; (b) the work done on the vanes; and (c) the efficiency of the system.

[*Ans.* (a) 1326.2 N {135.2 kg (f)} at 20° 34 ′ ;
(b) 12 416 N.m/s {1 265.6 kg (f) - m/s } ;
(c) 76.32%]

20.17 A 20 mm diameter jet having a velocity of 60 m/s impinges without shock on a series of vanes which move in the same direction as the jet. The shape of each vane is such that, if stationary, it would deflect the jet through an angle of 150°. Friction reduces the relative velocity by 10% as water flows across the vanes and there is a further windage loss given as $(0.4u^2/2g)$ N.m per N of water, where u is velocity of vanes. Find the velocity of vanes corresponding to an efficiency of 80%. Also determine the force on the vanes in, and at right angles, to the direction of their motion.

[*Ans.* 27 m/s ; 1106.3 N{112.77 kg(f)},
279.8 N {28.52 kg (f)}]

Apprise Education, Reprise Innovations

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hydraulic Turbines

Chapter 21

21.1 INTRODUCTION

Hydraulic (or water) turbines are the machines which use the energy of water (hydro-power) and convert it into mechanical energy. As such these may be considered as hydraulic motors or primemovers. The mechanical energy developed by a turbine is used in running an electric generator which is directly coupled to the shaft of the turbine. The electric generator thus develops electric power, which is known as *hydroelectric power.* Since the generation of hydro-electric power is relatively cheaper than the power generated by other sources such as coal, oil, etc., nowadays a number of hydro-electric and multipurpose projects have been undertaken in our country in order to harness more and more power from the available water potential.

The idea of utilising hydraulic energy to develop mechanical energy has been in existence for more than 2000 years. In the earlier days of water-power development, water wheels made of wood, were widely applied which used either the energy of falling water (i.e., potential energy) or the kinetic energy of the flowing stream of water. One of the types of water wheels formerly used was the *overshot wheel.* It consisted of a series of buckets attached to the periphery of a wheel, the diameter of which was equal to the available head. Water was permitted to enter the buckets at the top, and the imbalance created by the weight of the water caused the wheel to rotate. The buckets were designed to empty themselves when they reached to bottom of the wheel. The overshot wheel, when properly designed, had very good efficiency, but it could not be built to handle large quantity of water. Another type of water wheel formerly used was the *undershot wheel,* which used the kinetic energy of the water. An earlier type of undershot wheel consisted of a series of a straight blades attached to the periphery of a wheel and so placed that a swiftly moving stream of water used to strike the blades on the underside of the wheel. The efficiency of this type of wheel was low. As such an improvement on the straight blade type of undershot wheel was suggested by Poncelet, who instead of straight blades designed curved blades so that water strikes the blades of the wheel, practically without shock. This type of wheels were called Poncelet wheels.

However, these water wheels utilized small heads and were capable of producing small power. Moreover, these wheels had a low efficiency and they used to run very slowly and hence these could not be directly coupled to the modern fast running electric generators for the purpose of power generation. As such the water wheels have been completely replaced by the modern type of hydraulic (or water) turbines, which may operate under any head and practically any desired speed thereby enabling the generator to be coupled directly.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In general a water turbine consists of a wheel called *runner* (*or rotor*) having a number of specially designed *vanes or blades or buckets.* The water possessing a large amount of hydraulic energy when strikes the runner, it does work on the runner and causes it to rotate. The mechanical energy so developed is supplied to the generator coupled to the runner, which then generates electrical energy.

21.2 ELEMENTS OF HYDROELECTRIC POWER PLANTS

One of the essential requirements of the hydroelectric power generation is the availability of a continuous source of water with a large amount of hydraulic energy. Such a source of water may be made available if a natural lake or a reservoir may be found at a higher elevation or an artificial reservoir may be formed by constructing a dam across a river. Figure 21.1 shows a general layout of a hydroelectric power plant, in which an artificial storage reservoir formed by constructing a dam has

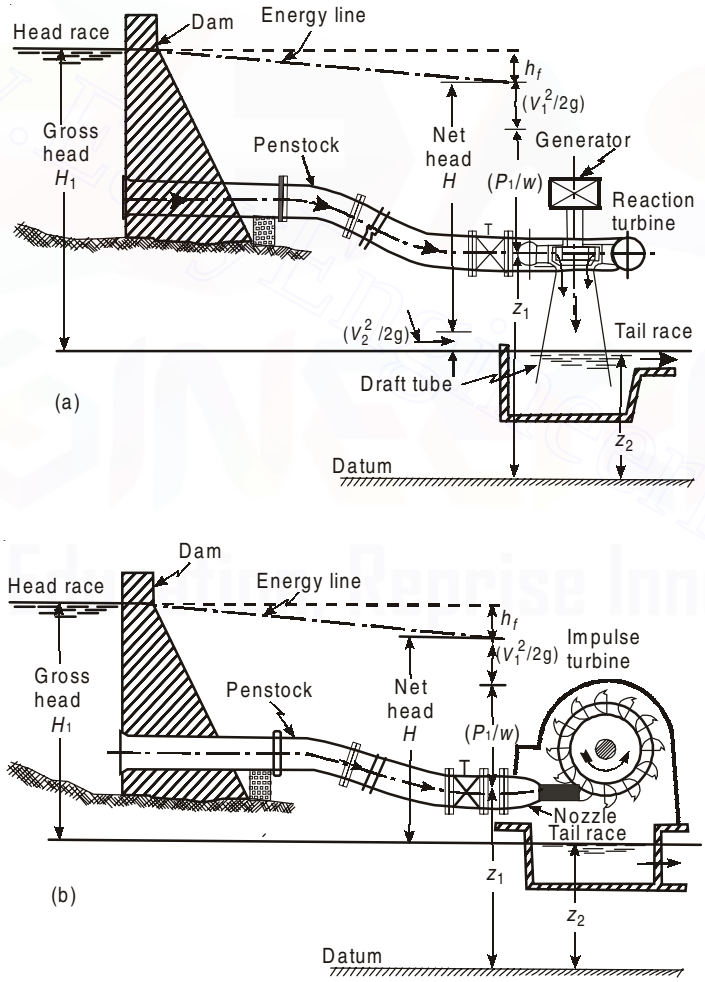


Figure 21.1 *General layout of a hydro-electric power plant*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

been shown. The water surface in the storage reservoir is known as *head race level* or simply *head race.* Water from the storage reservoir is carried through penstock or canals to the power house. *Penstocks* are the pipes of large diameter, usually made of steel, wood or reinforced concrete, which carry water under pressure from the storage reservoir to the turbine. In some installations smaller reservoirs known as forebays are also provided. A *forebay* is essentially a storage reservoir at the head of the penstocks. The purpose of a forebay is to temporarily store water when it is not required by the turbine and supply the same when required. When the power house is located just at the base of the dam no forebay is required to be provided since the reservoir itself serves the same purpose. However, if the power house is situated away from the storage reservoir and water is carried to the power house through a canal, then a forebay may be provided. In that case water from the reservoir is first led into forebay which in turn distributes it to penstocks through which it is supplied to the turbines. Furthermore, where the power house is located across a canal, a forebay may be provided by enlarging the canal just ahead of the power house.

The water passing through the turbine is discharged to the tail race. The *tail race* is the channel which carries water (known as tail water) away from the power house after it has passed through the turbine. It may be a natural stream channel or a specially excavated channel entering the natural stream at some point downstream from the power house. The water surface in the tail race channel is known as tail race level or simply tail race.

21.3 HEAD AND EFFICIENCIES OF HYDRAULIC TURBINES

(a) **Heads.** The head acting on a turbine may be defined in two ways as follows:

(i) **Gross head** is defined as the difference between the head race level and the tail race level when no water is flowing. As such the gross head is often termed as static head or total head and it may be represented by H_1 as shown in Fig. 21.1.

(ii) **Net or effective head** is the head available at the entrance to the turbine. It is obtained by subtracting from the gross head all the losses of head that may occur as water flows from the head race to the entrace of the turbine. The losses of head are mainly due to friction occurring in penstocks, canal etc. Thus if H represents the net head and h_f is the total loss of head between the head race and the entrance of the turbine then

$$H = H_1 - h_f \qquad ...(21.1)$$

For a *reaction* (*or encased*) *turbine* as shown in Fig. 21.1 (a) the net head is equal to the difference between (1) the pressure head at the entrance to the turbine plus the velocity head in the penstock at this point plus the elevation of this point above the assumed datum, and (2) the elevation of the tail water plus the velocity head in the draft tube at its exit. Thus

$$H = \left(\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1\right) - \left(Z_2 + \frac{V_2^2}{2g}\right) \qquad ...(21.2)$$

For an *impulse turbine* as shown in Fig. 21.1 (b) the net head is equal to the difference between (1) the pressure head at the entrance to the nozzle plus the velocity head in the penstock at this point plus the elevation of this point above the assumed datum and (2) the elevation of the tail water. Thus

$$H = \left(\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1\right) - Z_2 \qquad ...(21.3)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Hydroelectric power plants are usually classified according to the heads under which they work as high head, medium head and low head plants. High head plants are those which are working under heads more than about 250 m ; while low head plants are those which are working under heads less than about 60 m, and medium head plants are those which are working under heads ranging from 60 m to 250 m.

(b) Efficiencies. The various energy (or head) losses that may occur in a hydroelectric power plant with reaction and impulse turbine units are shown in Fig. 21.2. Accordingly the various efficiencies of the turbines may be expressed as follows.

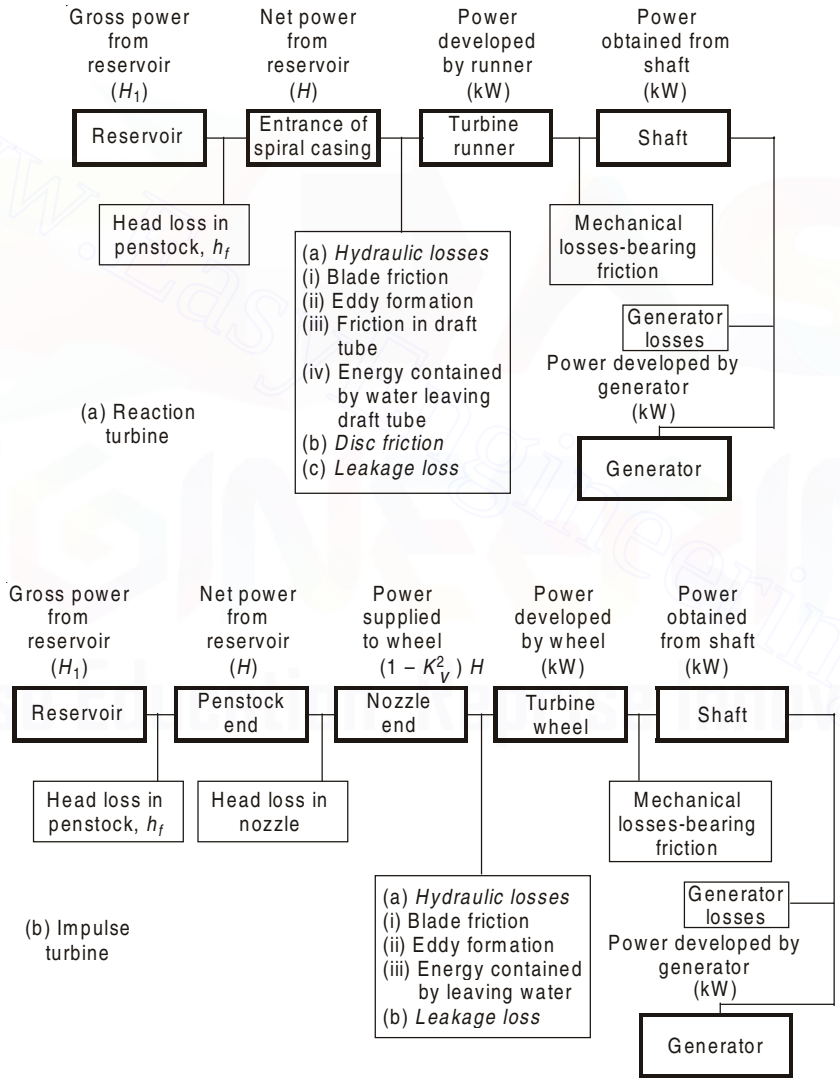


Figure 21.2 *Losses of energy in hydroelectric installations*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*i*) ***Hydraulic efficiency*** η_h**.** The hydraulic efficiency of the turbine is the ratio of the power developed by the runner to the net power supplied by the water at the entrance to the turbine. These two powers differ by the amount of the hydraulic losses. That is

$$\eta_h = \frac{\text{Power developed by the runner}}{\text{Net power supplied at the turbine entrance}}$$

or
$$\eta_h = \frac{\text{Power developed by the runner}}{[w(Q+\Delta Q)H]} \quad \ldots(21.4)$$

where Q is the quantity of water actually striking the runner, ΔQ is the quantity of water that is discharged directly to the tail race without striking the runner, and H is the net head available at the entrance to the turbine. However, if ΔQ is negligibly small, Eq. 21.4 becomes

$$\eta_h = \frac{\text{Power developed by the runner}}{(wQH)} \quad \ldots(21.5)$$

In SI units the powers are usually expressed in kilowatts (kW). However, in metric units the powers are expressed in metric horse power. In the later case the power supplied at the turbine entrance is termed as water horse power (W.H.P.).

(*ii*) ***Mechanical efficiency*** η_m**.** The mechanical efficiency of the turbine is the ratio of the power available at the turbine shaft to the power developed by the runner. These two powers differ by the amount of the mechanical losses viz., bearing friction. That is

$$\eta_m = \frac{\text{Power available at the turbine shaft}}{\text{Power developed by the runner}}$$

or
$$\eta_m = \frac{P}{\text{Power developed by the runner}} \quad \ldots(21.6)$$

where P is the power available at the turbine shaft. In metric units since the power is expressed in horse power (metric), the power available at the turbine shaft is termed as Shaft Horse Power, S.H.P., or Brake Horse Power, B.H. P.

(*iii*) ***Volumetric efficiency*** h_v**.** The volumetric efficiency is the ratio of the quantity of water actually striking the runner and the quantity of water supplied to the turbine. These two quantities differ by the amount of water that slips directly to the tail race without striking the runner. That is

$$\eta_v = \frac{Q}{(Q+\Delta Q)} \quad \ldots(21.7)$$

(*iv*) **O*verall efficiency*** η_o**.** The overall efficiency of the turbine is the ratio of the power available at the turbine shaft to the power supplied by the water at the entrance to the turbine. That is

$$\eta_o = \frac{\text{Power available at the turbine shaft}}{\text{Net power supplied at the turbine entrance}}$$

or
$$\eta_o = \frac{P}{[w(Q+\Delta Q)H]} \quad \ldots(21.8)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

It is evident from Eq. 21.8 that overall efficiency of the turbine is

$$\eta_o = \eta_h \times \eta_m \quad \ldots(21.9)$$

if η_h is given by Eq. 21.4 in which the volumetric efficiency is implied ; and

$$\eta_o = \eta_h \times \eta_v \times \eta_m \quad \ldots(21.9\ a)$$

if η_h is given by Eq. 21.5.

21.4 CLASSIFICATION OF TURBINES

Hydraulic turbines may be classified according to several considerations as discussed below.

According to the action of the water flowing through the turbine runners the turbines may be classified as impulse turbines and reaction turbines. In an *impulse turbine,* all the available energy of water is converted into kinetic energy or velocity head by passing it through a contracting nozzle provided at the end of the penstock. The water coming out of the nozzle is formed into a free jet which impinges on a series of buckets of the runner thus causing it to revolve. The runner revolves freely in air. The water is in contact with only a part of the runner at a time, and throughout its action on the runner and in its subsequent flow to the tail race, the water is at atmospheric pressure. A casing is however provided on the runner to prevent splashing and to guide the water discharged from the buckets to the tail race. Some of the impulse turbines are Pelton wheel, Turgo-impulse wheel, Girard turbine, Banki turbine, Jonval turbine etc. Out of these turbines only *Pelton wheel* is predominantly used at present, which has been described later.

In a *reaction turbine,* at the entrance to the runner, only a part of the available energy of water is converted into kinetic energy and a substantial part remains in the form of pressure energy. As water flows through the runner the change from pressure to kinetic energy takes place gradually. As such the pressure at the inlet to the turbine is much higher than the pressure at the outlet and it varies throughout the passage of water through the turbine. For this gradual change of pressure to be possible the runner in this case must be completely enclosed in an air-tight casing and the passage in entirely full of water throughout the operation of the turbine. The difference of pressure (or pressure drop) between the inlet and the outlet of the runner is called *reaction pressure,* and hence these turbines are known as reaction turbines. Some of the reaction turbines are Fourneyron, Thomson, Francis, Propeller, Kaplan, etc. Out of these the Francis and the Kaplan turbines are predominantly used at present which have been described later.

The turbines may also be classified according to the main direction of flow of water in the runner as (i) tangential flow turbine, (ii) radial flow turbine, (iii) axial flow turbine, and (iv) mixed flow turbine.

In a *tangential flow* turbine the water flows along the tangent to the path of rotation of the runner. Pelton wheel is a tangential flow turbine.

In a *radial flow* turbine the water flows along the radial direction and remains wholly and mainly in the plane normal to the axis of rotation, as it passes through the runner. A radial flow turbine may be either inward radial flow type or outward radial flow type. In an *inward radial flow* turbine the water enters at the outer circumference and flows radially inwards towards the centre of the runner. Old Francis turbine, Thomson turbine, Girard radial flow turbine etc., are some of the examples of inward radial flow turbine. In an *outward radial flow* turbine water enters at the centre and flows radially outwards towards the outer periphery of the runner. Fourneyron turbine is an example of outward radial flow turbine.

In an *axial flow* turbine the flow of water through the runner is wholly and mainly along the direction parallel to the axis of rotation of the runner. Jonval turbine, Girard axial flow turbine, Propeller turbine, Kaplan turbine etc., are some of the examples of axial flow turbines.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In *mixed flow* turbine, water enters the runner at the outer periphery in the radial direction and leaves it at the centre in the direction parallel to the axis of rotation of the runner. Modern Francis turbine is an example of the mixed flow type turbine.

On the basis of the head and quantity of water required, the turbines may be classified as (i) high head turbine, (ii) medium head turbine, and (iii) low head turbine.

High head turbines are those which are capable of working under very high heads ranging from several hundred metres to few thousand metres. These turbines thus require relatively less quantity of water. In general impulse turbines are high head turbines. In particular Pelton wheel has so far been used under a highest head of about 1770 m (5800 ft.).

Medium head turbines are those which are capable of working under medium heads ranging from about 60 m to 250 m. These turbines require relatively large quantity of water. Modern Francis turbines may be classified as medium head turbines.

Low head turbines are those which are capable of working under the heads less than 60 m. These turbines thus require a large quantity of water. Kaplan and other propeller turbines may be classified as low head turbines.

The turbines may also be classified according to their specific speed. The *specific speed* of a turbine is the speed of a geometrically similar turbine that would develop one kilowatt power when working under a head of one metre. However, in metric units the *specific speed* of a turbine is defined as the speed of a geometrically similar turbine that would develop one metric horse power when working under a head of one metre. On the basis of the specific speed the various turbines may be considered in the following three groups in which the values given in the brackets represent the range of specific speed in metric units.

(i) Specific speed varying from 8.5 to 30 (10 to 35) — Pelton wheel with single jet and upto 43 (50) for Pelton wheel with double jet.

(ii) Specific speed varying from 50 to 340 (60 to 400) —Francis turbine.

(iii) Specific speed varying from 255 to 860 (300 to 1000) — Kaplan and other propeller turbines.

The turbines may also be classified according to the disposition of their shafts. The turbines may be disposed with either vertical or horizontal shafts and hence these may be classified as turbines with *vertical disposition of shaft* and turbines with *horizontal disposition of shaft.* Out of the two types the turbines with vertical disposition of shaft are commonly adopted.

21.5 PELTON WHEEL

This is the only impulse type of hydraulic turbine now in common use. It is named after Lester A. Pelton (1829–1908), the American engineer who contributed much to its development in about 1880. It is well suited for operating under high heads.

Figure 21.3 shows the elements of a typical Pelton wheel installation. The runner consists of a circular disc with a number of buckets evenly spaced round its periphery. The buckets have a shape of double semi-ellipsoidal cups. Each bucket is divided into two symmetrical parts by a sharp-edged ridge known as splitter. One or more nozzles are mounted so that each directs a jet along a tangent to the circle through the centres of the buckets called the pitch circle. The jet of water impinges on the splitter, which divides the jet into two equal portions, each of which after flowing round the smooth inner surface of the bucket leaves it at its outer edge. The buckets are so shaped that the angle at the outlet tip varies from 10° to 20° (usually kept as 15°) so that the jet of water gets deflected through 160° to 170°. The advantage of having a double cup-shaped buckets is that the axial thrusts neutralise each

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

other, being equal and opposite, and hence the bearings supporting the wheel shaft are not subjected to any axial or end thrust. The back of the bucket is so shaped that as it swings downward into the jet no water is wasted by splashing. Further at the lower tip of the bucket a notch is cut which prevents the jet striking the preceding bucket being intercepted by the next bucket very soon, and it also avoids the deflection of water towards the centre of the wheel as the bucket first meets the jet. For low heads the buckets are made of cast iron, but for higher heads they are made of cast steel, bronze or stainless steel.

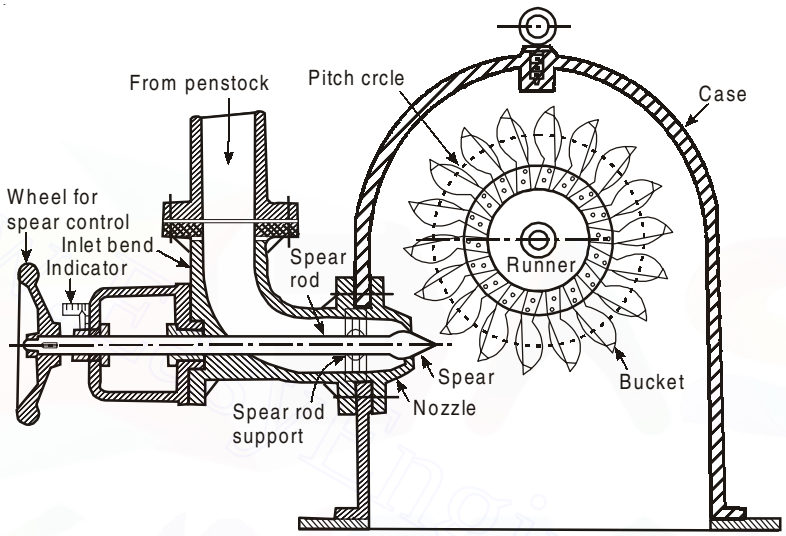


Figure 21.3 *Single jet Pelton wheel*

In order to control the quantity of water striking the runner, the nozzle fitted at the end of the penstock is provided with a spear or needle having a streamlined head which is fixed to the end of a rod as shown in Fig. 21.3. The spear may be operated either by a wheel (Fig. 21.3) in case of very small units or automatically by a governor (described later) in case of almost all the bigger units. When the shaft of the Pelton wheel is horizontal then not more than two jets are used. But if the wheel is mounted on a vertical shaft a larger number of jets (upto six) is possible.

A casing made of cast iron or fabricated steel plates is usually provided for a Pelton wheel as shown in Fig. 21.3. It has no hydraulic function to perform. It is provided only to prevent splashing of water, to lead water to the tail race and also to act as a safeguard against accidents.

Larger Pelton wheels are usually equipped with a small brake nozzle which when opened directs a jet of water on the back of the buckets, thereby bringing the wheel quickly to rest after it is shut down (as otherwise it would go on revolving by inertia for a considerable time).

21.6 WORK DONE AND EFFICIENCIES OF PELTON WHEEL

The transfer of work from the jet of water to the buckets may be determined by applying the momentum equation as indicated in Chapter 20. Figure 21.4 shows velocity triangles at the tips of the bucket of a Pelton wheel. Let

V = absolute velocity of jet before striking the bucket

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

V_1 = absolute velocity of jet leaving the bucket
u = absolute velocity of bucket considered along the direction tangential to the pitch circle
V_r = velocity of the incoming jet relative to the bucket
V_{r1} = velocity of the jet leaving the bucket relative to the bucket
V_w = velocity of whirl at inlet tip of the bucket
V_{w1} = velocity of whirl at outlet tip of the bucket
θ = angle through which the jet is deflected by the bucket (=180° – ϕ), where ϕ is the angle of the bucket at the outlet tip.

Since the velocities V and u are colinear, the velocity triangle at the inlet tip of the bucket is a straight line and thus

$$V_r = (V - u) \text{ and } V_w = V$$

At the outlet tip any one of the three velocity triangles as shown in Fig. 21.4 is possible depending upon the magnitude of u, corresponding to which it is a slow, medium or fast runner. As the inlet and

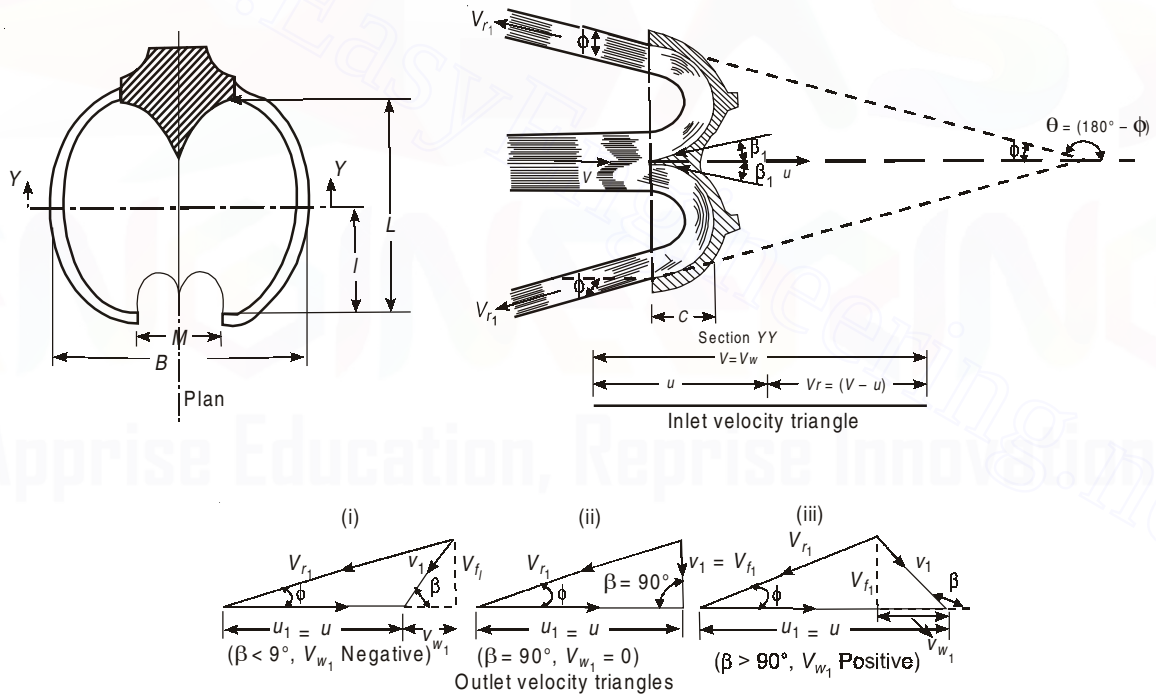


Figure 21.4 *Pelton wheel bucket and velocity-triangles*

the outlet tips of the bucket are at the same radial distance, the tangential velocity of the bucket at both the tips is same, i.e., $u = u_1$. Further the relative velocity V_{r1} with which the jet leaves the bucket will be somewhat less than the initial relative velocity V_r at the inlet tip. This is because, although the inner faces of the buckets are polished so as to minimize frictional losses as water flows over them, such losses cannot be completely eliminated. In addition to this some loss of energy will also take place as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the jet strikes the splitter. These losses of energy reduce the relative velocity between the jet and the bucket, and hence

$$V_{r1} = k(V_r) = k(V - u)$$

where k is a fraction slightly less than unity.

Now from the outlet velocity triangle (i) of Fig. 21.4

$$V_{w1} = (V_{r1}\cos\phi - u_1) = (kV_r\cos\phi - u)$$

and from the outlet velocity triangle (iii) of Fig. 21.4

$$V_{w1} = (u_1 - V_{r1}\cos\phi) = (u - kV_r\cos\phi)$$

where $\phi = (180° - \theta)$ is the angle of the bucket at the outlet tip.

However, if the losses are neglected then $k = 1$. Also for outlet velocity triangle (ii) $V_{w1} = 0$.

If W is the weight of water per second which strikes the buckets then, as shown in Chapter 20, the work done per second on the wheel is given as

$$\text{Work done} = \frac{W}{g}\left[V_w u \pm V_{w_1} u_1\right]$$

$$= \frac{W}{g}[V + kV_r\cos\phi - u]\,u$$

$$= \frac{W}{g}[(V - u)(1 + k\cos\phi)]\,u \qquad ...(21.10)$$

The energy supplied to the wheel is in the form of kinetic energy of the jet which is equal to $(WV^2/2g)$ per second.

Therefore the hydraulic efficiency of Pelton wheel is given by

$$\eta_h = \frac{(W/g)[(V - u)(1 + k\cos\phi)u]}{(WV^2/2g)}$$

or

$$\eta_h = \frac{2u(V - u)(1 + k\cos\phi)}{V^2} \qquad ...(21.11)$$

Equation 21.11 indicates that for $u = 0$, as well as for $u = V$, $\eta_h = 0$. Thus η_h will be maximum for some value of u in between 0 and V. For a given jet velocity V and the bucket tip angle ϕ at outlet, the condition for maximum efficiency may be obtained by considering $(d\eta_h/du) = 0$. Thus if k is assumed constant

$$\frac{d(h_h)}{du} = \frac{2(1 + k\cos\phi)}{V^2}(V - 2u) = 0$$

or

$$u = \left(\frac{V}{2}\right) = 0.5\,V \qquad ...(21.12)$$

Thus the hydraulic efficiency is maximum when the bucket speed is equal to half the velocity of the jet. Substituting this condition in Eq. 21.11, the maximum hydraulic efficiency is obtained as

$$(\eta_h)_{max} = \frac{1}{2}(1 + k\cos\phi) \qquad ...(21.13)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Now if $k = 1$, then Eq. 21.13 indicates that the maximum value of η_h will be equal to 1 or 100% when $\phi = 0$ or $\theta = 180°$ i.e., the buckets are so shaped that the jet gets deflected through 180°. This is however, theoretical maximum value of η_h. The actual maximum value of η_h will be slightly less and it varies from 0.9 to 0.94 (or 90 to 94%). This is so because the actual value of k is not equal to one but it is slightly less. Further in actual practice θ can not be made equal to 180°, because in that case the jet leaving the bucket will strike the back of the bucket just following it, thus exerting a retarding force on it. Hence in order that the outgoing jet keeps clear of the following bucket, the bucket tip angle ϕ at outlet is usually kept ranging from 10° to 20°, (the average value being 15°) so that the jet gets deflected through 160° to 170°. The angle ϕ is also known as side clearance angle.

If there is no loss of energy as the water flows over the buckets then the work done per second on the Pelton wheel (or the power output of the Pelton wheel) may also be expressed as

$$\text{Work done} = \frac{W}{2g}(V^2 - V_1^2) \qquad ...(21.14)$$

Accordingly the hydraulic efficiency of Pelton wheel may also be expressed as

$$\eta_h = \frac{[(W/2g)(V^2 - V_1^2)]}{(W/2g)V^2} = \frac{V^2 - V_1^2}{V^2} \qquad ...(21.15)$$

By substituting the value of V_1 obtained from the outlet velocity triangle, Eq. 21.15 becomes exactly similar to Eq. 21.11 with $k = 1$.

The loss of head as the water flows through the buckets of the Pelton wheel may be obtained by applying the Bernoulli's equation between the inlet and the outlet tips of the bucket. Thus

$$\frac{V^2}{2g} = \frac{V_w u \pm V_{w_1} u_1}{g} + \frac{V_1^2}{2g} + h_L$$

where h_L is the loss of head in the buckets.

From the velocity triangles at the inlet and the outlet tips of the bucket

$$\frac{V_w u \pm V_{w_1} u_1}{g} = \left[\frac{V^2 - V_1^2}{2g} + \frac{u^2 - u_1^2}{2g} + \frac{V_{r1}^2 - V_r^2}{2g}\right]$$

$$= \left[\frac{V^2 - V_1^2}{2g} + \frac{V_{r1}^2 - V_r^2}{2g}\right] (\text{since } u = u_1)$$

Thus by substitution, we get

$$h_L = \frac{V_r^2 - V_{r1}^2}{2g} = \frac{Vr^2}{2g}(1 - k^2)$$

$$= \frac{(V-u)^2}{2g}(1 - k^2) \qquad ...(21.16)$$

If P is the power available at the shaft then the mechanical efficiency of Pelton wheel is given by

$$\eta_m = \frac{P}{W\left[\frac{(V-u)u(1 + k\cos\phi)}{g}\right]} \qquad ...(21.17)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The volumetric efficiency is given by Eq. 21.7 and its value for Pelton wheel ranges from 97 to 99%.

The overall efficiency is given by Eq. 21.8 and for large Pelton wheels the overall efficiency of 85 to 90% may usually be achieved.

21.7 WORKING PROPORTIONS OF PELTON WHEEL

(i) The ideal velocity of jet usually known as spouting velocity = $\left(\sqrt{2gH}\right)$ where H is the net head. However, the actual velocity of the jet is slightly less, due to friction loss in the nozzle. Thus

$$V = (K_v \text{ or } C_v)\sqrt{2gH} \qquad ...(21.18)$$

where K_v or C_v is the coefficient of velocity for the nozzle with its value ranging from 0.97 to 0.99.

(ii) As obtained earlier for maximum η_h the velocity of wheel u at pitch circle is equal to 0.5V. However, in actual practice the maximum efficiency occurs when the value of u is about 0.46V. Moreover, it is convenient to express u in terms of H, in the form of an expression $u = K_u\left(\sqrt{2gH}\right)$, where K_u is known as *speed ratio*. Thus considering $V = 0.98\left(\sqrt{2gH}\right)$, we obtain

$$u = 0.46\ \text{V} = 0.45\left(\sqrt{2gH}\right) \qquad ...(21.19)$$

In practice the value of K_u ranges from 0.43 to 0.47.

(iii) Angle through which jet of water gets deflected in buckets = 165°, unless otherwise stated.

(iv) Least diameter d of the jet is given by

$$d = \left[\frac{4Q}{\pi K_v\left(\sqrt{2gH}\right)}\right]^{1/2} \qquad ...(21.20)$$

where Q is the discharge through the jet in m^3/s.

Taking $K_v = 0.98$, we obtain

$$d = 0.542\left(\frac{Q}{\sqrt{H}}\right)^{1/2} \text{ metres} \qquad ...(21.20\ a)$$

(v) *Mean diameter* or the pitch diameter D of the Pelton wheel may be obtained as follows. If the wheel rotates at N r.p.m., then $u = (\pi DN/60)$. Thus

$$D = \frac{60u}{\pi N} = \frac{60(K_u\sqrt{2gH})}{\pi N} \qquad ...(21.21)$$

(vi) The ratio of pitch diameter D of the wheel to the jet diameter d is known as *jet ratio* and is represented by m i.e., $m = (D/d)$. The jet diameter is an important parameter in the design of a Pelton wheel. For maximum efficiency the jet ratio should be from 11 to 14 and normally a jet ratio of 12 is adopted in practice. A smaller value of m results in either too close a spacing of the buckets or too few buckets for the whole jet to be used. A larger value of m results in a more bulky installation. However, in extreme cases a value of m as low as 7 and as high as 110 has been used.

(vii) Some of the main dimensions of the bucket of Pelton wheel as shown in Fig. 21.4 expressed in terms of the jet diameter are as noted below:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

B = (4 to 5) d; L = (2.4 to 3.2) d;
C = (0.81 to 1.05) d; l = (1.2 to 1.9) d;
M = (1.1 to 1.25) d; Angle ϕ = 10° to 20°
Angle β_1 = 5° to 8°.

(viii) The number of buckets for a Pelton wheel should be such that the jet is always completely intercepted by the buckets so that volumetric efficiency of the turbine is very close to unity. The number of buckets is usually more than 15. Certain empirical formulae have been developed for determining the number of buckets. One such formula which is widely used has been given by Taygun according to which the number of buckets Z is approximately given by

$$Z = \left(\frac{D}{2d} + 15\right) = (0.5\ m + 15) \qquad ...(21.22)$$

This equation has been found to hold good for all values of m ranging from 6 to 35.

21.8 DESIGN OF PELTON TURBINE RUNNER

A Pelton turbine runner is required to be designed to develop a known power P, when running at a known speed N r.p.m. under a known head H. The probable values of η_o, K_v, K_u and m are assumed. The various steps involved in the design are as follows:

(i) Determine the required discharge Q from the relation

$$P = \eta_0(WH) = \eta_0\,(wQH)$$

(ii) From Eq. 21.18 calculate the velocity V of the jet.

(iii) Calculate the total area of the jets required by using the relation $a = (Q / V)$.

(iv) Calculate the pitch circle diameter D from Eq. 21.21.

(v) Calculate the required diameter d of the jet from the relation $m = (D/d)$ and also calculate the corresponding area of the jet.

(vi) Obtain the number of jets required by dividing the total area of jets obtained in step (ii) by the area of each jet obtained in step (v).

(vii) The fractional number of jets obtained in step (vi) may be rounded up to the appropriate integral number and the corresponding diameter of each jet to be actually provided may be calculated. Care should however be taken to have the actual value of m very nearly equal to the assumed value.

(viii) Calculate the number of buckets to be provided from Eq. 21.22 and the bucket dimensions from the relevant equations.

21.9 MULTIPLE JET PELTON WHEEL

The power developed by a Pelton wheel provided with a single jet is usually quite low. This is so because on account of the restrictions of the jet velocity, wheel speed and the jet ratio, a single jet cannot be made big enough to develop any desired power. The amount of power developed by a single runner of a Pelton wheel turbine may however be increased by providing more than one jet spaced evenly around the same runner. The nozzles must never be spaced so closely that water issued from one jet after striking the runner interferes with another jet. As such the maximum number of jets so far used with a single runner of some large units is six. A Pelton wheel having more than one jet spaced around its runner is called *multiple jet Pelton wheel.* If P is the power developed by a Pelton wheel when working under head H and having one jet only, then the power developed by the same Pelton wheel will be (nP), if n jets are used for its working under the same head.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Sometimes even if by using more number of jets for a single runner, the required power is not developed then a number of runners mounted on a common shaft may be used. In some cases a combination of the above two systems may be used, i.e., a number of multiple jet wheels may be mounted on the same shaft.

21.10 RADIAL FLOW IMPULSE TURBINE

For a radial flow impulse turbine the inlet velocity triangle is not a straight line and hence

$$\text{Work done} = \frac{W}{g}(V_w u \pm V_{w_1} u_1)$$

Further if there is no loss of energy in the runner vanes then the work done may also be expressed by Eq. 21.14. Thus equating the two, we get

$$\frac{W}{g}(V_w u \pm V_{w_1} u_1) = \frac{W}{2g}(V^2 - V_1^2)$$

From the velocity triangles at the inlet and the outlet tips of a radial vane it can be shown that

$$[V_w u \pm V w_1 u_1] = \left[\frac{V^2 - V_1^2}{2} + \frac{u^2 - u_1^2}{2} + \frac{V_{r1}^2 - V_{r^2}}{2}\right]$$

Thus by substituting this value in the above expression it becomes

$$\frac{u^2 - u_1^2}{2g} + \frac{V_{r1}^2 - V_r^2}{2g} = 0$$

or
$$\frac{V_{r_1}^2}{2g} = \frac{V_r^2}{2g} - \left(\frac{u^2 - u_1^2}{2g}\right) \quad \ldots(21.23)$$

The second term on the right hand side of Eq. 21.23 represents the centrifugal head impressed on the water as it flows through the runner of a radial flow impulse turbine. For an outward flow turbine $u_1 > u$, then from Eq. 21.23, $V_{r1} > V_r$; and for an inward flow turbine $u_1 < u$ and hence $V_{r1} < V_r$. That is the centrifugal head increases the relative velocity of water in an outward flow turbine and decreases it in an inward flow turbine. As such a better control of speed can be enforced in the case of an inward flow turbine.

21.11 REACTION TURBINES

As stated earlier the principal distinguishing features of a reaction turbine are that only part of the total head of water is coverted into velocity head before it reaches the runner, and that the water completely fills all the passage in the runner. The pressure of water changes gradually as it passes through the runner. The two reaction type of turbines which are predominantly used these days are Francis turbine and Kaplan turbine, which are described below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

21.12 FRANCIS TURBINE

Figure 21.5 illustrates a Francis turbine which is a mixed flow type of reaction turbine. It is named in honour of James B. Francis (1815–92), an American Engineer, who was the first to develop an inward radial flow type of reaction turbine in 1849. Later on it was modified and the modern Francis turbine is a mixed flow type, in which water enters the runner radially at its outer periphery and leaves axially at its centre.

The water from the penstock enters a *scroll casing* (also called *spiral casing*) which completely surrounds the runner. The purpose of the casing is to provide an even distribution of water around the circumference of the turbine runner, maintaining an approximately constant velocity for the water so distributed. In order to keep the velocity of water constant throughout its path around the runner, the cross-sectional area of the casing is gradually decreased. The casing is made of a cast steel, plate steel, concrete or concrete and steel depending upon the pressure to which it is subjected. Out of these a plate steel scroll casing is commonly provided for turbines operating under 30 m or higher heads.

From the scroll casing the water passes through a *speed ring* or *stay ring* (see Fig. 21.5). The speed ring consists of an upper and a lower ring held together by series of fixed vanes called *stay vanes.* The number of stay vanes is usually taken as half the number of guide vanes. The speed ring has two functions to perform. It directs the water from the scroll casing to the guide vanes or wicket gates. Further it resists the load imposed upon it by the internal pressure of water and the weight of the turbine and the electrical generator and transmits the same to the foundation. The speed ring may be either of cast iron or cast steel or fabricated steel.

From the speed ring the water passes through a series of *guide vanes* or *wicket gates* provided all around the periphery of the turbine runner. The function of guide vanes is to regulate the quantity of water supplied to the runner and to direct water on to the runner at an angle appropriate to the design. The guide vanes are airfoil shaped and they may be made of cast steel, stainless steel or plate steel. Each guide vane is provided with two stems, the upper stem passes through the head cover and the lower stem seats in a bottom ring. By a system of levers and links, all the guide vanes may be turned about their stems, so as to alter the width of the passage between the adjacent guide vanes, thereby allowing a variable quantity of water to strike the runner. The guide vanes are operated either by means of a wheel (for very small units) or automatically by a governor.

The main purpose of the various components so far described is to lead the water to the *runner* with a minimum loss of energy. The runner of a Francis turbine consists of a series of a curved vanes (about 16 to 24 in number) evenly arranged around the circumference in the annular space between two plates. The vanes are so shaped that water enters the runner radially at the outer periphery and leaves it axially at the inner periphery. The change in the direction of flow of water, from radial to axial, as it passes through the runner, produces a circumferential force on the runner which makes the runner to rotate and thus contributes to the useful output of the runner. The runners are usually made up of cast iron, cast steel, mild steel or stainless steel. Often instead of making the complete runner of stainless steel, only those portions of the runner blades, which may be subjected to cavitation erosion, are made of stainless steel. This reduces the cost of the runner and at the same time ensures the operation of the runner with a minimum amount of maintenance. The runner is keyed to a shaft which is usually of forged steel. The torque produced by the runner is transmitted to the generator through the shaft which is usually connected to the generator shaft by a bolted flange connection.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

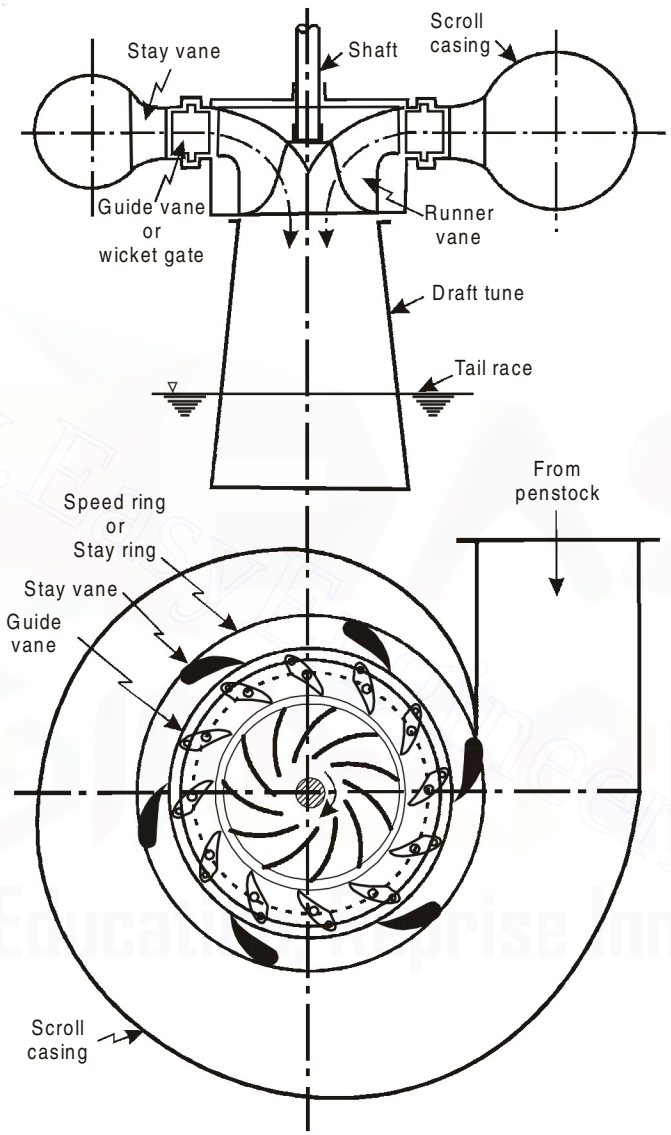


Figure 21.5 *Sectional arrangement of Francis turbine*

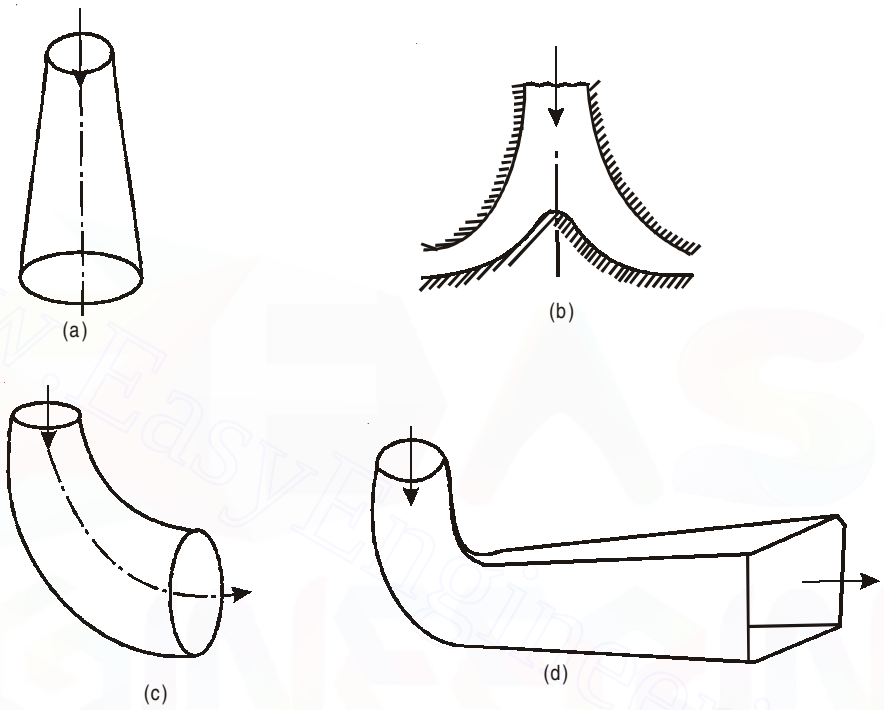
The water after passing through the runner flows to the tail race through a draft tube. A *draft tube* is a pipe or passage of gradually increasing cross-sectional area which connects the runner exit to the tail race. It may be made of cast or plate steel or concrete. It must be airtight and under all conditions of operation its lower end must be submerged below the level of water in the tail race. The draft tube has two purposes as follows:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) It permits a negative or suction head to be established at the runner exit, thus making it possible to instal the turbine above the tail race level without loss of head.

(ii) It converts a large proportion of velocity energy rejected from the runner into useful pressure energy i.e., it acts as a recuperator of pressure energy.



(a) Straight divergent tube
(b) Moody spreading tube (or hydraucone tube)
(c) Simple elbow tube
(d) Elbow tube Having circular cross section at inlet and rectangular at outlet

Figure 21.6 *Different types of draft tubes*

Figure 21.6 shows the different types of draft tubes which are employed in the field to suit particular conditions of installation. Of these the types (a) and (b) are the most efficient, but the types (c) and (d) have an advantage that they require lesser excavation for their installation. It has been observed that for straight divergent type draft tube the central cone angle should not be more than 8°. This is so because if this angle is more than 8° the water flowing through the draft tube will not remain in contact with its inner surface, with the result that eddies are formed and the efficiency of the draft tube is reduced.

21.13 WORK DONE AND EFFICIENCIES OF FRANCIS TURBINE

If W is the weight of water per second which strikes the runner then as derived in Chapter 20 the work done per second on the runner may be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\text{Work done} = \frac{W}{g}[V_w u - V w_1 u_1]$$

Evidently the maximum output under specified conditions is obtained by making the velocity of whirl at exit V_{w1}equal to zero.

Then

$$\text{Work done} = \frac{W}{g}(V_w u)$$

or Work done per unit weight of water

$$= \left(\frac{V_w u}{g}\right)$$

Now if H is the net head then the input energy per second for the runner = (WH).

Therefore hydraulic efficiency of the Francis turbine is given by

$$\eta_h = \frac{V_w u}{gH} \qquad ...(21.24)$$

However, if V_{w1} is not equal to zero, then

$$\eta_h = \frac{V_w u - V_{w_1} u_1}{gH} \qquad ...(21.24\ a)$$

The value of η_h ranges from 85 to 95%.

Again if P is the power available at the runner shaft then the mechanical efficiency is given by

$$\eta_m = \frac{P}{W\left(\frac{V_w u}{g}\right)} \qquad ...(21.25)$$

However, if V_{w1} is not equal to zero, then

$$\eta_m = \frac{P}{W\left(\frac{V_w u - V_{w1} u_1}{g}\right)} \qquad ...(21.25\ a)$$

Further the overall efficiency is given by Eq. 21.9 as

$$\eta_0 = \eta_h \times \eta_m = \frac{P}{(WH)}$$

The overall efficiency of a Francis turbine ranges from 80 to 90%.

Degree of reaction

Degree of reaction ρ, is defined as the ratio of pressure drop in the runner to the hydraulic work done on the runner. Thus is p and p_1 are the pressures at the inlet and the outlet of the runner, then

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\rho = \frac{\left(\frac{p}{w} - \frac{p_1}{w}\right)}{\frac{V_w u - V_{w_1} u_1}{g}}$$

and if $V_{w_1} = 0$, then

$$\rho = \frac{\left(\frac{p}{w} - \frac{p_1}{w}\right)}{\frac{V_w u}{g}}$$

As indicated in Example 21.17 for no loss of head in the runner the degree of reaction can be expressed in terms of guide vane and runner vane angles. Thus for a given runner and set of guide vanes the degree of reaction is more or less constant if there is no loss of head in the runner. However, in actual practice because of the head loss in the runner the degree of reaction is not constant.

21.14 WORKING PROPORTIONS OF FRANCIS TURBINE

(i) The ratio of the width B of the runner to the diameter D of the runner (see Fig. 21.8) is represented by n, that is

$$n = (B/D) \qquad ...(21.26)$$

The value of n ranges from 0.10 to 0.45.

(ii) The ratio of the velocity of flow V_f at the inlet tip of the vane to the spouting velocity $\sqrt{2gH}$ is known as *flow ratio* ψ, that is

$$\psi = \frac{V_f}{\sqrt{2gH}}; \text{ or } V_f = \psi \sqrt{2gH} \qquad ...(21.27)$$

The value of ψ ranges from 0.15 to 0.3.

(iii) In this case also the speed ratio K_u is defined as

$$K_u = \frac{u}{\sqrt{2gH}}; \text{ or } u = K_u \sqrt{2gH} \qquad ...(21.27a)$$

The value of K_u ranges from 0.60 to 0.90.

21.15 DESIGN OF FRANCIS TURBINE RUNNER

A Francis turbine runner is required to be designed to develop a known power P, when running at a known speed N r.p.m. under a known head H. The probable values of η_h, η_0, n and ψ are assumed. The design of the runner which involves the determination of the size and the vane angles is carried out as follows:

(i) Determine the required discharge Q from the relation

$$P = \eta_0 (WH) = \eta_0 (wQH)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(ii) If Z is the number of vanes in the runner, t is the thickness of the vane at inlet and B is the width of the wheel at inlet, then the area of flow section at the wheel inlet = $(\pi D - Zt)\, B = k\pi BD$, where k is a factor which allows for the thickness of the vanes. Then

$$Q = k\pi BDV_f = k\pi nD^2V_f \quad ...(21.28)$$

Since $B = nD$.

Thus assuming a suitable value of k, the diameter D and the width B of the runner can be determined. For the first approximation the vane thickness may be neglected in which case k may be assumed to be equal to unity.

(iii) The tangential velocity of the runner at inlet may be determined from the equation $u = (\pi DN/60)$.

(vi) The velocity of whirl V_w at inlet of the runner can be determined from the expression $\eta_h = (V_w u/gH)$.

(v) From the inlet velocity triangle, the guide vane angle α and the runner vane angle θ at inlet can be calculated from the expression $\tan\alpha = (V_f/V_w)$ and $\tan\theta = V_f/(V_w - u)$.

(vi) The runner diameter D_1 at the outlet end varies from (1/3) D to (2/3) D and usually it is taken equal to (1/2) D. Thus the tangential velocity of the runner at outlet may be determined from the expression $u_1 = (\pi D_1 N/60)$.

(vii) If t_1 and B_1 are respectively the thickness of the vane and the width of the runner at outlet, then

$$Q = (\pi D_1 - Zt_1)\, B_1 \times V_{f1} = k_1\pi D_1 B_1 V_{f1} \quad ...(21.29)$$

From Eq. 21.28 and 21.29

$$\frac{V_f}{V_{f_1}} = \frac{k_1\pi B_1 D_1}{k\pi BD} \quad ...(21.30)$$

Normally it is assumed that $V_f = V_{f1}$ and $k = k_1$, then $B_1 = 2B$ since $D_1 = (1/2)D$.

(viii) Generally the runner is designed to have the velocity of whirl V_{w1} at outlet equal to zero, i.e., $V_{w1} = 0$ and $\beta = 90°$. Then from outlet velocity triangle the runner vane angle ϕ at outlet may be determined from the expression $\tan\phi = (V_{f1}/u_1)$.

(ix) The number of runner vanes should be either one more or one less than the number of guide vanes, in order to avoid setting up of periodic impulse.

21.16 DRAFT TUBE THEORY

Refer Fig. 21.7 in which points 1, 2 and 3 have been considered at the runner entrance, runner exit and at the outlet end of the draft tube respectively. By applying Bernoulli's equation between points 1 and 2 the pressure and velocity heads at the inlet and the outlet ends of the runner may be obtained.

Applying Bernoulli's equation between points 2 and 3, we get

$$\frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 = \frac{p_3}{w} + \frac{V_3^2}{2g} + z_3 + h_f \quad ...(21.31)$$

where p_2, p_3 and V_2, V_3 are the pressures and velocities at points 2 and 3 respectively, and h_f is the loss of head in the draft tube. From Eq. 21.31, we have

$$\frac{p_2}{w} = \frac{p_3}{w} - (z_2 - z_3) - \left(\frac{V_2^2 - V_3^2}{2g}\right) + h_f$$

But

$$\frac{p_3}{w} = \frac{p_a}{w} + h$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad \frac{p_2}{w} = \frac{p_a}{w} - (z_2 - z_3 - h) - \left(\frac{V_2^2 - V_3^2}{2g}\right) + h_f$$

Since $(z_2 - z_3 - h) = H_s$, the height of runner exit above tail race level, thus

$$\frac{p_2}{w} = \frac{p_a}{w} - \left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + h_f \qquad \ldots(21.32)$$

It may be seen from Eq. 21.32 that the pressure at the runner exit is suction pressure, that is below atmospheric. H_s is known as static suction head and $[(V_2^2 - V_3^2)/2g]$ is known as dynamic suction head. Generally h_f is expressed as $h_f = k\,(V_2^2 - V_3^2)/2g$, then

$$\frac{p_2}{w} = \frac{p_a}{w} - \left[H_s + (1-k)\frac{V_2^2 - V_3^2}{2g}\right] \qquad \ldots(21.33)$$

Now if the draft tube is made cylindrical then $V_2 = V_3$; and if the frictional loss is neglected, then Eq. 21.32 indicates that the pressure head at the runner exit would be less than atmospheric pressure by an amount H_s, the height of the runner exit above the tail race. Hence with such a draft tube the turbine would not loose head H_s because of equal reduction in pressure head at the runner exit. On the other hand if a draft tube of gradually increasing cross-section is used, then the value of pressure at runner

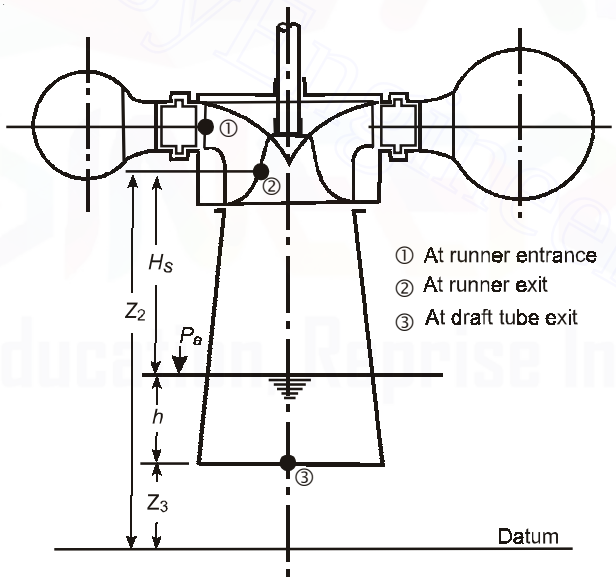


Figure 21.7 *Draft tube theory*

exit is further reduced by an amount equal to $(1-k)\,[(V_2^2 - V_3^2)/2g]$. In other words such a draft tube causes a large portion of kinetic head to be converted into pressure head.

The efficiency of a draft tube is defined as

$$\eta_d = \frac{\text{Actual regain of pressure head}}{\text{Velocity head at entrance to draft tube}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The actual regain of pressure head

$$= \left[\frac{V_2^2 - V_3^2}{2g}\right] - h_f = (1-k)\left(\frac{V_2^2 - V_3^2}{2g}\right)$$

$$\therefore \quad \eta_d = \frac{[(V_2^2 - V_3^2)/2g] - h_f}{(V_2^2/2g)} = \frac{(1-k)[(V_2^2 - V_3^2)/2g]}{(V_2^2/2g)} \qquad \ldots(21.34)$$

21.17 SHAPE OF FRANCIS TURBINE RUNNER AND DEVELOPMENT OF KAPLAN TURBINE RUNNER

A Francis turbine runner of given diameter, when required to develop certain power P, under a low head, should be so designed that it admits comparatively large quantity of water. This can be done either by increasing the flow ratio ψ or the ratio n ($= B/D$). An increase in the value of ψ means a higher

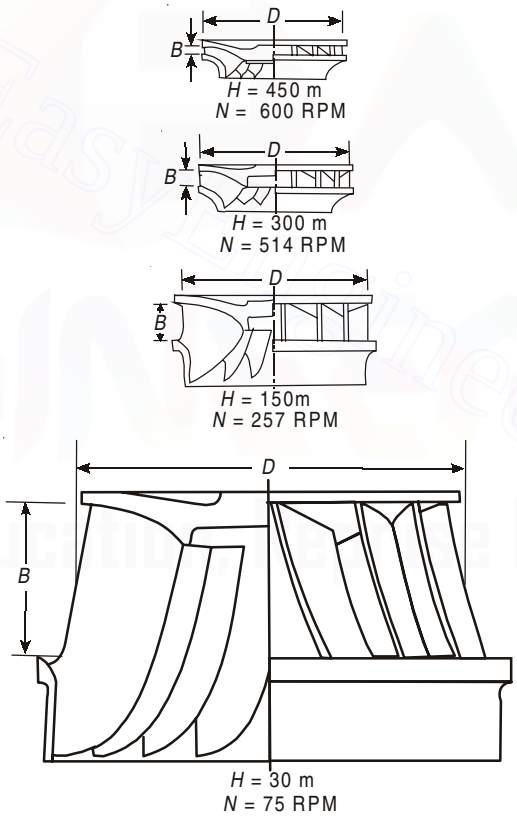


Figure 21.8 *Relative sizes of Francis turbine to produce 73550 W (10^5 h.p.)*

value of V_f which will results in excessive losses in the form of velocity energy rejected from the runner. As such ψ cannot be increased to any amount. On the other hand an increase in the value of n results in a higher value of B (for a given value of D), which will yield a larger inlet area. But a larger inlet area must be accompanied by a larger outlet area for all the water to escape freely. A larger outlet area is, however, obtained by making the discharge at the runner exit to be axial and hence the modern Francis

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

turbine runners are of 'mixed flow type'. Figure 21.8 shows the relative sizes and shapes of the Francis turbine runners required to develop 73550 W (10^5 h.p.) under varying heads. It will be observed that as the head for the turbine is reduced and hence the discharge is increased (in order to develop the required power) the runner size considerably increases and also the shape of the runner has to be such that the flow is more and more axial right from the entrance to the runner. Thus as the quantity of water flowing through the runner increases it has to be so designed that it has more and more axial flow. Since in the case of a Kaplan turbine a large quantity of water is required to be passed, it is so designed that the flow is purely axial right from the inlet section to the outlet section of the runner.

21.18 KAPLAN TURBINE

A Kaplan turbine is a type of propeller turbine which was developed by the Austrian engineer V. Kaplan (1876–1934). It is an axial flow turbine, which is suitable for relatively low heads, and hence requires a large quantity of water to develop large amount of power. It is also a reaction type of turbine and hence it operates in an entirely closed conduit from the head race to the tail race.

From Fig. 21.9 it will be seen that the main components of a Kaplan turbine such as scroll casing, stay ring, arrangement of guide vanes, and the draft tube are similar to those of a Francis turbine. Between the guide vanes and the runner the water in a Kaplan (or propeller) turbine turns through a right-angle into the axial direction and then passes through the runner. The runner of a Kaplan (or propeller) turbine has four or six (or eight in some exceptional cases) blades and it closely resembles a ship's propeller. The blades (or vanes) attached to a hub or boss are so shaped that water flows axially through the runner. Ordinarily the runner blades of a propeller turbine are fixed, but the Kaplan turbine runner blades can be turned about their own axis, so that their angle of inclination may be adjusted while the turbine is in motion. This adjustment of the runner blades is usually carried out automatically by means of a servomotor operating inside the hollow coupling of turbine and generator shaft. When both guide-vane angle and runner-blade angle may thus be varied, a high efficiency can be maintained over a wide range of operating conditions. In other words even at part load, when a lower discharge is flowing through the runner, a high efficiency can be attained in the case of a Kaplan turbine. It may be explained with the help of Fig. 21.10, in which inlet and outlet velocity triangles for a Kaplan turbine runner working at constant

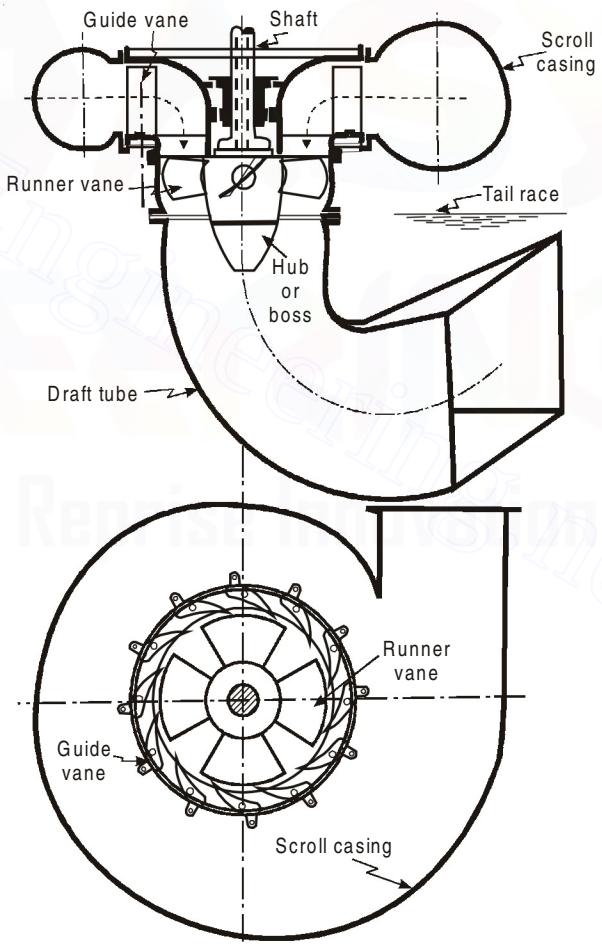


Figure 21.9 *Sectional arrangement of Kaplan turbine*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

speed under constant head at *full load* and at *part load* are shown. It will be observed that although the corresponding changes in the flow through the turbine runner does affect the shape of the velocity triangles, yet as the blade angles are simultaneously adjusted, the water under all the working conditions flows through the runner blades without shock. As such the eddy losses which are inevitable in Francis and propeller turbines are almost completely eliminated in a Kaplan turbine.

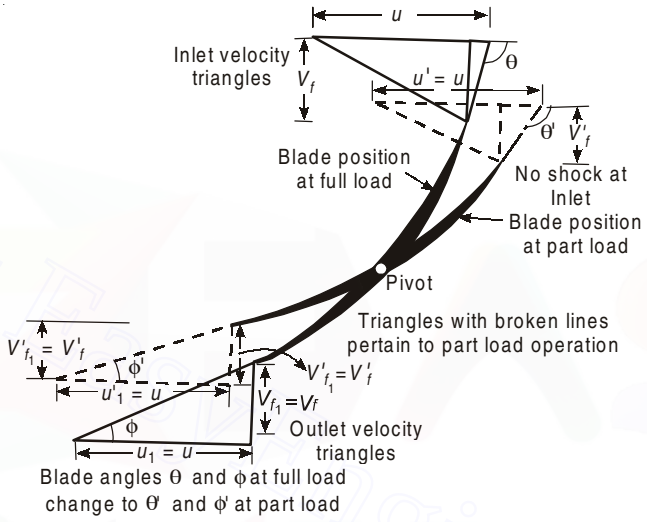


Figure 21.10 *Velocity triangles for a Kaplan turbine runner blade*

21.19 WORKING PROPORTIONS OF KAPLAN TURBINE

In general the main dimensions of Kaplan turbine runners are established by a procedure similar to that for a Francis turbine runner. However, the following are the main deviations:

(i) Choose an appropriate value of the ratio $n = (d/D)$, where d is hub or boss diameter and D is runner outside diameter. The value of n usually varies from 0.35 to 0.60.

(ii) The discharge Q flowing through the runner is given by

$$Q = \frac{\pi}{4}(D^2 - d^2)\, V_f = \frac{\pi}{4}(D^2 - d^2)\, \psi \sqrt{2gH} \qquad \ldots(21.35)$$

The value of flow ratio ψ for a Kaplan turbine is around 0.70.

(iii) The runner blades of Kaplan turbine runner are warped or twisted, the blade angle being greater at the outer tip than at the hub. This is because the peripheral velocity of the blades being directly proportional to radius, it will vary from section to section along the blade, and hence in order to have shock free entry and exit of water the blades with angles varying from section to section will have to be designed.

The expressions for the work done and the efficiencies of Kaplan turbine are same as those for Francis turbine.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

21.20 NEW TYPES OF TURBINES

A brief description of some of the new types of turbines is given below.

1. Deriaz or Diagonal Turbine. Figure 21.11 illustrates a Deriaz (or Diagonal) Turbine which is a reaction turbine. It is named in honour of its inventor P. Deriaz. This type of turbine is intermediate between the mixed-flow and the axial-flow turbines, because the flow of water as it passes through the

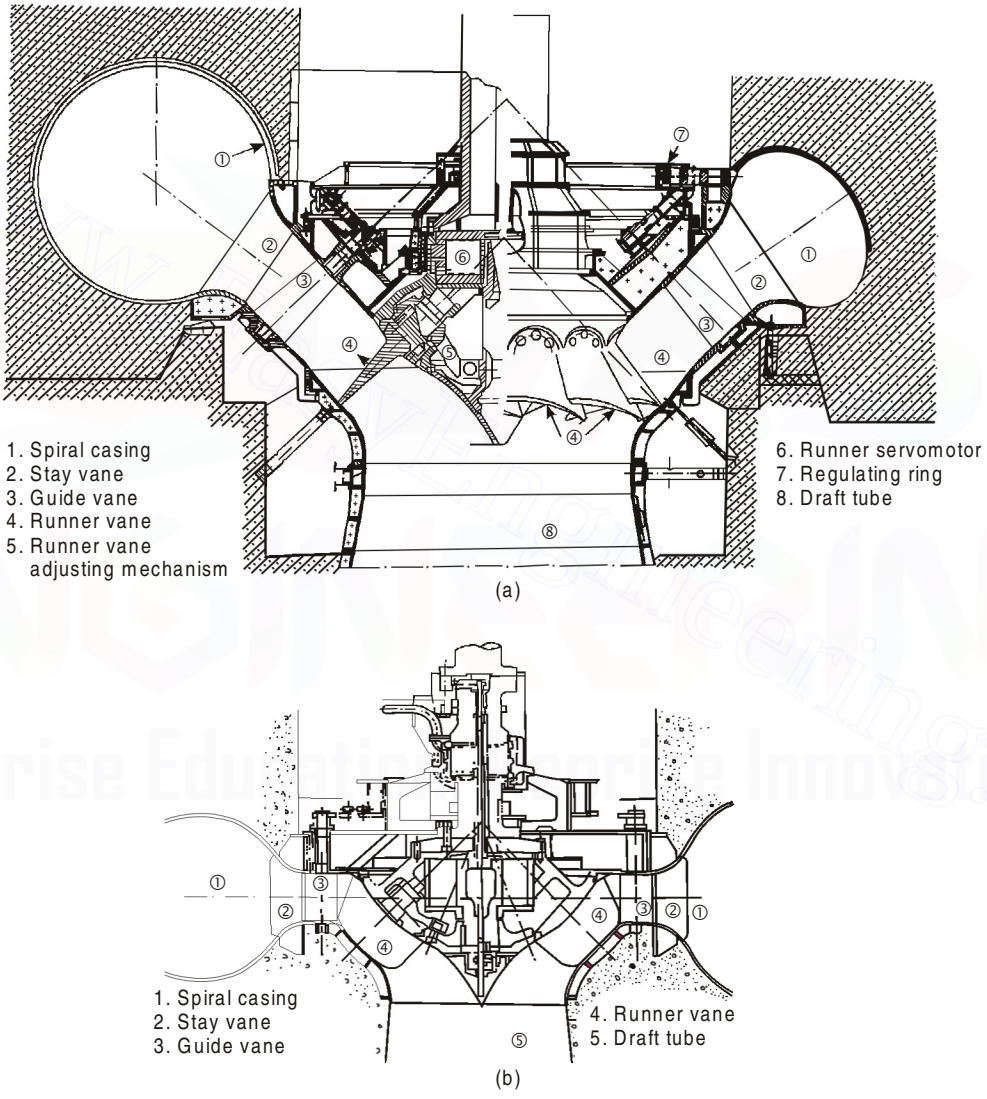


Figure 21.11 *Deriaz (or Diagonal) turbine*

runner is at an angle of 45° to the axis and hence it is also known as Diagonal turbine. It can be used under heads ranging from 30 m to 150 m. Alike Kaplan turbines the runner blades of these turbines are also adjustable. The other components such as scroll (or spiral) casing, stay vanes, guide vanes and

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

draft tube are same as those in the case of other reaction turbines. As shown in Fig. 21.11 (a) generally in the case of a Deriaz turbine the stay vanes as well as the guide vanes are also inclined as a result of which passages, unlike in conventional Kaplan turbines, are straight. However, some Deriaz turbines are also provided with stay vanes and guide vanes similar to those provided for conventional Kaplan turbines as shown in Fig. 21.11 (b). The Deriaz runner is so shaped that it can be used both as a turbine as well as a pump and hence it may be classified as a reversible type turbine. As such Deriaz turbines are quite suitable for pumped storage hydropower plants (See Chapter 27). In some of the Western countries these turbines have been installed but in India no Deriaz turbine has been installed so far.

2. Tubular Turbine. A Tubular turbine is an axial flow turbine with either adjustable or non-adjustable runner vanes and hence it is similar to Kaplan or propeller turbines. However, in a tubular turbine the scroll casing is not provided but the runner is placed in a tube extending from the head water to the tail water as shown in Fig. 21.12, and hence it is called tubular turbine. It is a low head turbine which can be used under heads ranging from 3 m to 15 m. As shown in Fig. 21.12 the tubular turbines may have either vertical or inclined or horizontal disposition of shaft. The tubular turbines are quite suitable for tidal power plants (See Chapter 27).

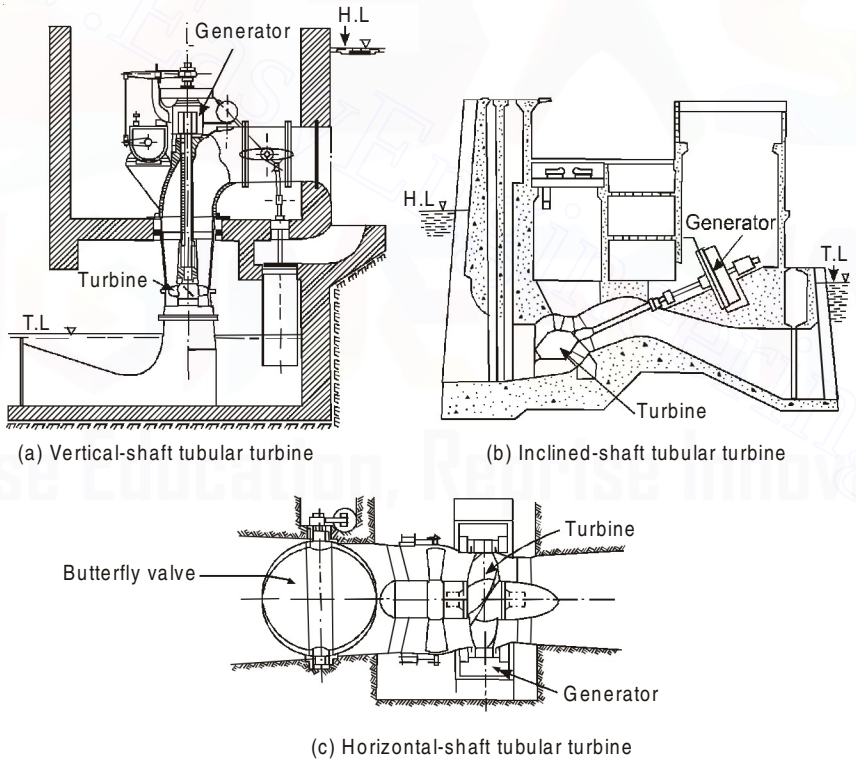


Figure 21.12 *Tubular turbine*

3. Bulb Turbine. It is same as tubular turbine with horizontal disposition of shaft in which the generator is housed in a bulb-shaped watertight casing as shown in Fig. 21.13 and hence it is called bulb turbine. The bulb together with the turbine runner form an integral unit. The outer surface of the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

bulb is streamlined so as to form a passage for the flow of water from the head pond to the turbine with minimum loss of energy. The bulb could be either upstream or downstream of the runner but in most of the cases it is on the upstream of the runner. The bulb turbines are also quite suitable for tidal power

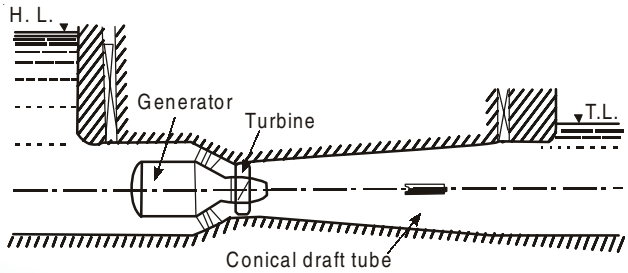


Figure 21.13 *Bulb turbine*

plants. A number of bulb turbines are operating successfully all over the world. In India at present there are only two power stations where bulb turbines have been installed. At Gandak Western Canal Power House three bulb turbine units each of 5.5 MW are operating under a head of 6.1 m. The runner diameter is 4.1 m and it runs at 107.2 r.p.m. The total discharge rating is about 103.8 m^3/sec per unit. The Kosi East Canal Power House has four bulb turbine units each of 5.6 MW operating under a head of 6.1 m and has discharge rating about 104.5 m^3/sec per unit. The runner diameter in this case is 4.5 m.

21.21 GOVERNING OF TURBINES

All the modern hydraulic turbines are directly coupled to the electric generators. The generators are always required to run at constant speed irrespective of the variations in the load. This constant speed N r.p.m. of the generator is given by the expression

$$f = \frac{pN}{60} \qquad ...(21.36)$$

where f is the frequency for power generated in cycles per second and p is the number of pair of poles for the generator. Usually $f = 50$ and hence

$$N = \frac{60f}{p} = \frac{3000}{p} \qquad ...(21.37)$$

The speed of the generator can be maintained constant only if the speed of the turbine runner is constant equal to the one given by Eq. 21.37. It is then known as the *synchronous speed* of the turbine runner for which it is designed.

The load on the generator will, however, go on varying and now if the input for the turbine remains same then the speed of the turbine runner will tend to either increase or decrease depending on the load on the generator is reduced or increased. This in turn will cause the speed of the generator also to vary accordingly, which is, however, not desirable because it may result in varying the frequency of power generation. As such the speed of the turbine runner is required to be maintained constant so that the generator always runs at constant speed under all conditions of working. It is usually done by regulating the quantity of water flowing through the runner in accordance with the variations in the load. Such an operation of regulation of speed of turbine runner is known as governing of turbine and is usually done automatically by means of a governor. One of the common types of governor which is predominantly used with modern turbines is *oil pressure governor.* As shown in Fig. 21.14 its component parts are as follows:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) Servomotor also known as relay cylinder.
(ii) Relay valve also known as control valve or distribution valve.
(iii) Actuator or pendulum which is belt or gear driven from the turbine main shaft.
(iv) Oil sump.
(v) Oil pump which is driven by belt connected to turbine main shaft.
(vi) A system of oil supply pipes connecting the oil sump with the relay valve and the relay valve with the servomotor.

The working of the governor is explained below:

When the load on the generator drops, the speed of turbine runner increases. Now since the actuator or pendulum is driven by the turbine main shaft, due to increase in the speed the balls move outward, resulting in an upward movement of the sleeve shown in Fig. 21.14 (a). As the sleeve moves up, the left

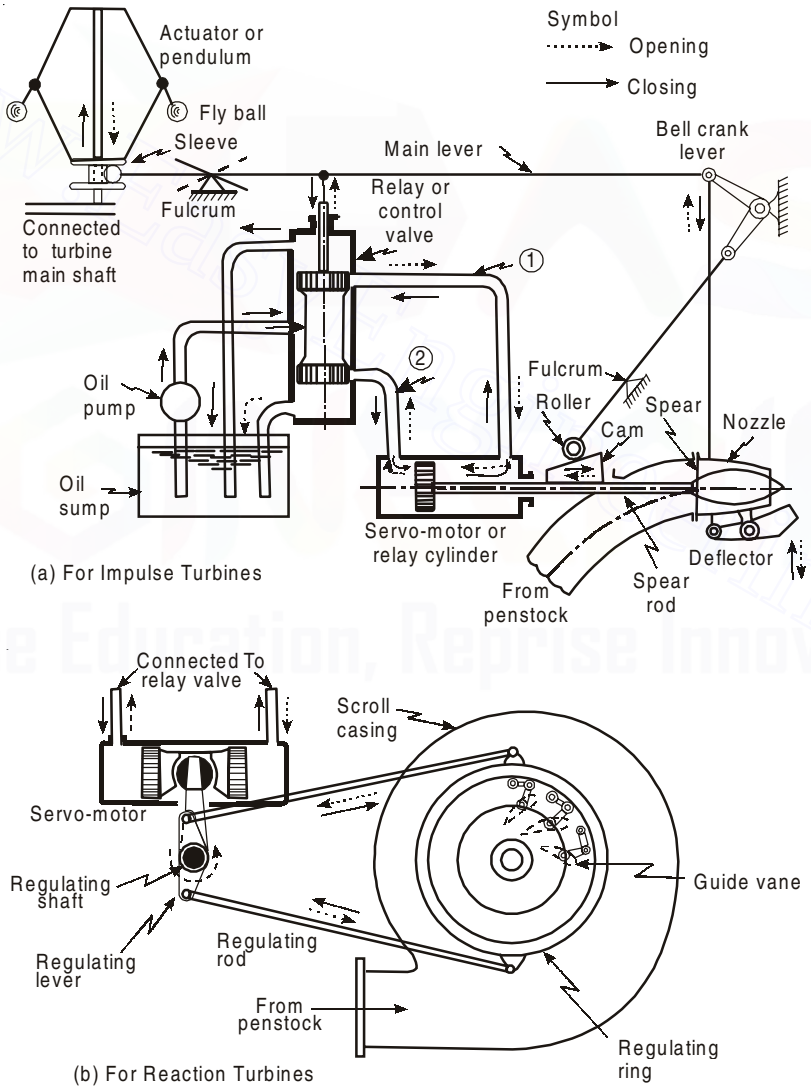


Figure 21.14 *Governing mechanism of turbines*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

hand end of the main lever is raised, which causes the bell crank lever to move downward and simultaneously pushes the piston of the control valve down in its cylinder. In the case of a Pelton turbine the downward motion of the bell crank lever brings the *deflector* in front of the jet thereby diverting a portion of the jet away from the buckets. On the other hand in the case of reaction turbine such as Francis or Kaplan turbine a *relief valve* shown in Fig. 21.15 is provided. A relief valve consists of a spear, which under normal condition of operation of turbine, is held by oil or water under pressure in such a position that it keeps the opening of the bye-pass from the spiral casing to the tail race closed. The downward motion of the bell crank lever (or other suitable arrangement) opens the pilot valve of the pressure chamber so that the pressure on the spear is reduced, thereby permitting the spear to be lifted up and allowing a portion of water to flow directly from the spiral casing to the tail race through the bye-pass without striking the turbine runner. Thus both these devices, viz., deflector and relief valve have similar function to perform. These eliminate the necessity of rapid closure of the nozzle opening or the guide vanes, at the same time the quantity of water striking the runner is reduced. The rapid closure of the nozzle opening or the guide vanes is not desirable because a sudden reduction of the rate of flow in the penstock may result in serious water hammer problems.

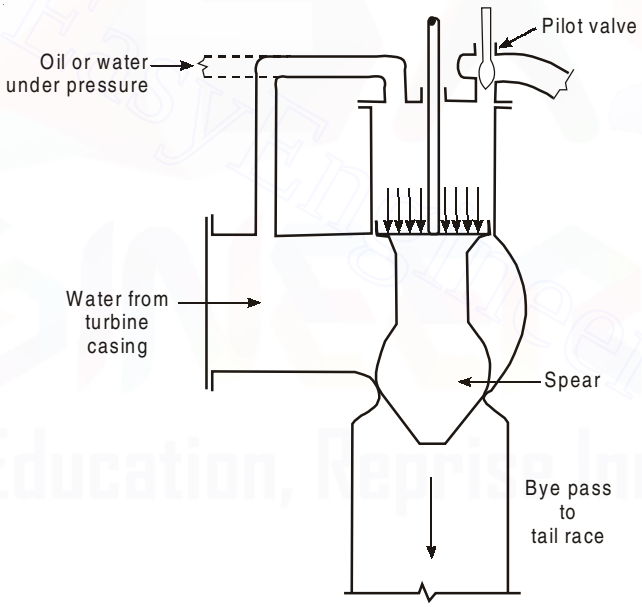


Figure 21.15 *Relief valve*

With the downward motion of the piston of the control valve the passage for pipeline 2 opens and oil under pressure is admitted from the control valve cylinder to the servomotor on the left side of the piston. The servomotor piston, therefore, moves to the right. In the case of Pelton wheel the servomotor piston being connected to the spear rod, thus causes the spear to move forward. The forward motion of the spear reduces the area of the nozzle outlet and thus decreases the rate of flow (as required for the decreased load) whereby the normal turbine speed is restored. However, in the case of a reaction turbine the forward motion of the servomotor piston is transmitted to the regulating ring as shown in Fig. 21.14 (b) which causes all the guide vanes to move simultaneously in one direction, and thus the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

area of flow passage between the adjacent guide vanes is reduced and the rate of flow of water striking the runner is also reduced. When once the normal turbine speed is restored the main lever attains its initial normal position and through a suitable cam arrangement the deflector is brought back to its original position or the relief valve stops byepassing of water to the tail race.

When the load on the generator increases, the speed of the turbine runner decreases. Due to this the balls move inward resulting in the downward movement of the sleeve. The left hand end of the main lever is lowered which pulls the piston of the control valve up in the cylinder. With the upward motion of the piston of the control valve the passage for pipeline 1 opens and oil under pressure rushes from the control valve cylinder to the servomotor on the right side of the piston. The servomotor piston then moves to the left. This increases the area of the nozzle outlet or the passage between the adjacent guide vanes, thereby allowing a large quantity of water to strike the runner (as required for the increased load) and the normal speed for the turbine runner is thus restored.

In the case of Kaplan turbine since in addition to guide vanes the runner vanes are also adjustable, the governor is required to operate both sets of vanes simultaneously. The runner vanes are also operated by a separate servomotor and a control valve, and the servomotors and the control valves for both, the runner and the guide vanes, are interconnected to ensure that for a given guide vane opening there shall be a definite runner vane inclination.

However, the large hydro-electric units are nowadays provided with *electrohydraulic governers.* In general, an electrohydraulic governor consists of a hydromechanical actuator and an electrical equipment cabinet, both mounted in the machine room. The electrical equipment cabinet contains the electrical part of the governor only, while the actuator cabinet contains the hydromechanical devices. A special techogenerator connected to the main shaft transmits currents at a frequency of 50 cycles to the electric measuring circuits of the governor. When the speed of the unit changes, the frequency of the techogenerator supplies also changes, whereupon a discriminator responding to frequency variations emits an electrical signal. After being amplified, this signal is transmitted to the electromechanical final control element, where it is transformed into a mechanical displacement transmitted through the lever system of the amplifier valve to the main governor valve. Being moved from its mid (or normal) position in direct proportion to the strength of the electric signal, the main governor feeds oil under pressure to the servomotor of the turbine distributor. The spear in the nozzle or the guide vanes thus turn in the direction corresponding to the sign of electric signal.

21.22 RUNAWAY SPEED

For a turbine working under maximum head and full gate opening, if the external load suddenly drops to almost zero value and at the same time the governing mechanism of the turbine also fails, then the turbine runner will tend to race up and it will attain the maximum possible speed. This maximum or limiting speed of the turbine runner is known as *runaway speed*. Obviously for safe design the various rotating components are designed for the runaway speed. For a Pelton wheel the runaway speed normally ranges from 1.8 to 1.9 times its normal speed, for a Francis turbine it normally ranges from 2 to 2.2 times its normal speed and for a Kaplan turbine it normally ranges from 2.5 to 3 times its normal speed.

21.23 SURGE TANKS

As indicated earlier when the load on the generator decreases the governor reduces the rate of flow of water striking the runner in order to maintain the constant speed for the runner. But the sudden reduction of the rate of flow in the penstock may lead to setting up of water hammer in the pipe, which

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

may cause excessive inertia pressure in the pipeline due to which the pipe may burst. Two devices viz., deflector and relief valve as described earlier are thus provided to avoid the sudden reduction of the rate of flow in the penstock. But neither of these devices are of any assistance when the load on the generator increases and the turbine is in need of more water. Thus in order to fulfil both the above noted requirements, in addition to the above noted devices certain other devices such as *surge tank* and *forebay* are usually employed. Surge tanks are employed in the case of high and medium head hydro-power plants where the penstock is very long, and forebays are suitable for medium and low head hydro-power plants where the length of the penstock is short.

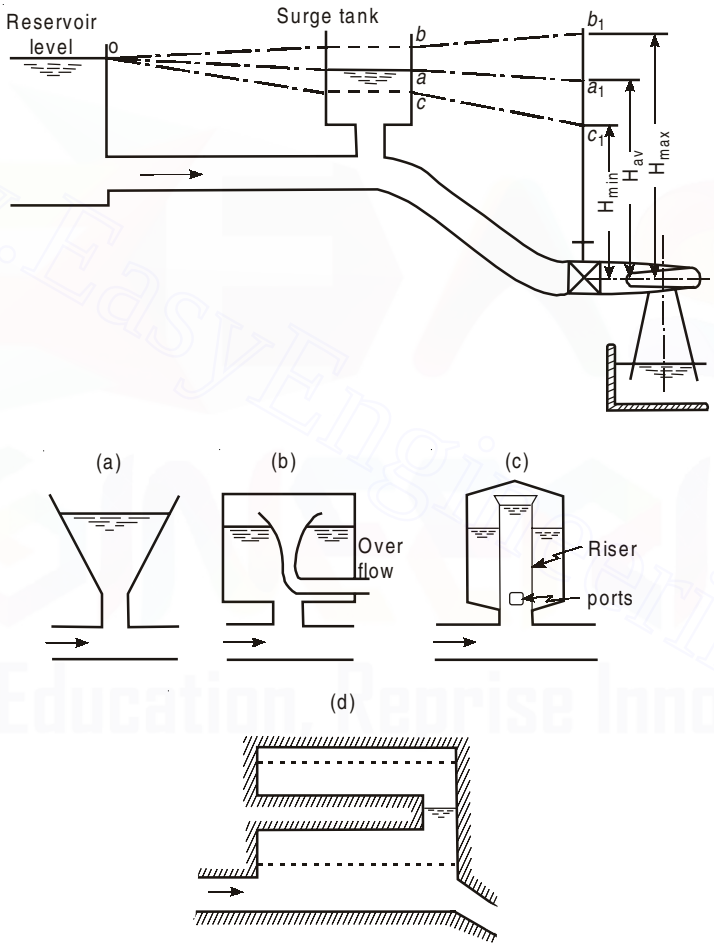


Figure 21.16 *Different types of surge tanks*

An ordinary surge tank is a cylindrical open-topped storage reservoir, as shown in Fig. 21.16, which is connected to the penstock at a point as close as possible to the turbine. The upper lip of tank is kept well above the maximum water level in the supply reservoir. When the load on the turbine is steady and normal and there are no velocity variations in the pipeline there will be a normal pressure gradient oaa_1. The water surface in the surge tank will be lower than the reservoir surface by an

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

amount equal to the friction head loss in the pipe connecting the reservoir and the surge tank. When the load on the generator is reduced, turbine gates are closed and the water moving towards the turbine has to move backward. The rejected water is then stored in the surge tank in the space between levels *a* and *b* and a rising pressure gradient obb_1 is developed. The retarding head so built up in the surge tank reduces the velocity of flow in the pipeline corresponding to the reduced discharge required by the turbine.

When the load on the generator increases, the governor opens the turbine gates to increase the rate of flow entering the runner. The increased demand of water by the turbine is partly met by the water stored between levels *a* and *c* in the surge tank. As such the water level in the surge tank falls and a falling pressure gradient occ_1 is developed. In other words, the surge tank develops an accelerating head which increases the velocity of flow in the pipeline to a value corresponding to the increased discharge required by the turbine.

The various other types of surge tanks are also shown in Fig. 21.16. Type (a) is a conical type surge tank, type (b) has an internal bell mouth spillway which permits the overflow to be easily disposed of. Type (c) is known as the *differential surge tank*, which is provided with a central riser pipe having small ports or holes at its lower end. The main advantage of this type is that for the same stabilising effects its capacity may be less than that of a simple cylindrical surge tank. This is so because in a differential tank retarding and accelerating heads are developed more promptly than in a simple surge tank in which the heads only built-up gradually as the tank fills. Moreover no water is spilled to waste from the differential tank. Type (d) is also similar in performance to the differential tank, but it is suitable when appropriate earth or rock excavation can be carried out.

ILLUSTRATIVE EXAMPLES

Example 21.1. *Prove that the maximum efficiency of Pelton wheel occurs when the ratio of bucket velocity u to the jet velocity V is given by the expression*

$$\rho = \frac{u}{V} = \frac{1-\cos\theta + k_1}{2(1-\cos\theta) + k_1 + k_2}$$

where the loss due to bucket friction and shock is expressed as $\frac{k_1(V-u)^2}{2g}$ *and that due to bearing friction and windage losses as* $\left(k_2 \frac{u^2}{2g}\right)$; *$k_1$ and k_2 are constants and θ being the bucket angle at outlet. Neglect volumetric losses. [Note: $\angle\theta = (180° - \phi)$ of Fig. 21.4].*

Solution

Considering the losses given in the problem the net amount of work done per unit weight of water

$$= \frac{1}{g}\left[(V-u)(1-\cos\theta)\right]u - k_1\frac{(V-u)^2}{2g} - k_2\frac{u^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus

$$\eta = \frac{\frac{1}{g}[(V-u)(1-\cos\theta)]u - k_1\frac{(V-u)^2}{2g} - k_2\frac{u^2}{2g}}{\frac{V^2}{2g}}$$

For η to be maximum $\left(\frac{d\eta}{du}\right) = 0$, thus by differentiating with respect to u, we get

$$\frac{d\eta}{du} = \frac{2(V-2u)(1-\cos\theta) + 2k_1(V-u) - 2k_2u}{V^2} = 0$$

or $$(V-2u)\,(1-\cos\theta) + k_1\,(V-u) - k_2u = 0$$

or $$V\,(1-\cos\theta + k_1) = u[2(1-\cos\theta) + k_1 + k_2]$$

$$\therefore \quad \frac{u}{V} = \frac{1-\cos\theta + k_1}{2(1-\cos\theta) + k_1 + k_2}$$

Example 21.2. *In a Francis turbine, prove that hydraulic efficiency η_h of the turbine can be expressed as*

$$\eta_h = \frac{2}{2 + \frac{k_1 + k_2 + k_3 + k_4}{(\cot\alpha - \cot\theta)\{\cot\alpha(1+n^2) - n(\cot\phi + n\cot\theta)\}}}$$

where k_1, k_2, k_3 and k_4 represent the fraction of the losses in the guide vanes, runner vanes, draft tube and at exit respectively expressed in terms of the velocity of flow head; α, θ and ϕ are the guide vane angle, the runner vane angle at inlet and the runner vane angle at outlet respectively and n is the ratio of the inner to outer diameter of the runner. Assume the velocity of flow to remain constant in the runner.

Solution

If H is the net head acting on the turbine then the work done by the runner per unit weight of water may be expressed as

$$\frac{V_w u - V_{w_1} u_1}{g} = H - \text{losses}$$

The losses may be combinedly expressed as

$$(k_1 + k_2 + k_3 + k_4)\,\frac{V_f^2}{2g}$$

also $V_f = V_{f1}$ being constant.

Thus

$$\frac{V_w u - V_{w_1} u_1}{g} = H - (k_1 + k_2 + k_3 + k_4)\,\frac{V_f^2}{2g} \qquad \text{...(i)}$$

From inlet and outlet velocity triangles

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_w = V_f \cot\alpha\,;\ \frac{V_f}{V_w - u} = \tan\theta$$

or $$\frac{V_f}{V_f \cot\alpha - u} = \tan\theta\,;$$

$$\therefore \quad u = V_f(\cot\alpha - \cot\theta)$$

$$\frac{V_{f_1}}{v_{w_1} + u_1} = \tan\phi\,;$$

or $$\frac{V_f}{V_{w_1} + u_1} = \tan\phi$$

Also $$u = \frac{\pi DN}{60}\,;\ u_1 = \frac{\pi D_1 N}{60}$$

$$\therefore \quad u_1 = \left(\frac{D_1}{D}\right)u$$

$$= nu = n\,V_f\,(\cot\alpha - \cot\theta)$$

Introducing this value of u_1, we get

$$\frac{V_f}{V_{w_1} + nV_f(\cot\alpha - \cot\theta)} = \tan\phi = \frac{1}{\cot\phi}$$

or $$V_{w1} = V_f[\cot\phi - n(\cot\alpha - \cot\theta)]$$

The hydraulic efficiency is given by

$$\eta_h = \frac{V_w u - V_{w_1} u_1}{gH}$$

From Eq. (i), we have

$$gH = (v_w u - v_{w1}\,u_1) + (k_1 + k_2 + k_3 + k_4)\,\frac{V_f^2}{2}$$

Thus

$$\eta_h = \frac{(V_w u - V_{w_1} u_1)}{(V_w u - V_{w_1} u_1) + (k_1 + k_2 + k_3 + k_4)\dfrac{V_f^2}{2}}$$

$$= \frac{2}{2 + \dfrac{(k_1 + k_2 + k_3 + k_4)V_f^2}{(V_w u - V_{w_1} u_1)}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the values of V_w, u, V_{w1} and u_1, we get

$$\eta_h = \frac{2}{2+\dfrac{(k_1+k_2+k_3+k_4)}{(\cot\alpha-\cot\theta)\{\cot\alpha(1+n^2)-n(\cot\phi+n\cot\theta)\}}}$$

Example 21.3. *A Pelton wheel has to be designed for the following data. Power to be developed = 6 000 kW. Net head available = 300 m; Speed = 550 r.p.m.; Ratio of jet diameter to wheel diameter =1/10; and overall efficiency = 85%.*

Find the number of jets; diameter of the jet; diameter of the wheel; and the quantity of water required.

Solution

Assume $K_v = 0.98$; speed ratio $K_u = 0.46$

Thus $V = K_v\sqrt{2gH}$

$= 0.98\sqrt{2\times9.81\times300} = 75.19$ m/s

$u = K_u\sqrt{2gH}$

$= 0.46\sqrt{2\times9.81\times300} = 35.29$ m/s

$$\eta_0 = \frac{P}{wQH}$$

or $$0.85 = \frac{6000\times10^3}{9810\times Q\times300}$$

$\therefore$ $Q = 2.399$ m^3/s

$$u = \frac{\pi DN}{60} = \frac{\pi\times D\times550}{60} = 35.29 \text{ m/s}$$

$\therefore$ $D = 1.225$ m

Since $$\frac{d}{D} = \frac{1}{10}$$

$\therefore$ $$d = \frac{1.225}{10} = 0.1225 \text{ m} = 122.5 \text{ mm}$$

Area of jet $$a = \frac{\pi}{4}\times(0.1225)^2 = 0.0118 \text{ m}^2$$

Total jet area required $$= \frac{Q}{V} = \frac{2.399}{75.19} = 0.0.319 \text{ m}^2$$

$\therefore$ No. of jets required $$= \frac{0.0319}{0.0118} = 2.7 \approx 3$$

Thus three jets are required each having a diameter

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$d = \left(\frac{0.0319}{3(\pi/4)}\right)^{1/2} = 0.1164 \text{ m} = 116.4 \text{ mm}$$

$$\frac{d}{D} = \frac{0.1164}{1.225} = \frac{1}{10.52}$$

which is close to the given value.

Example 21.4. *A Pelton wheel has a mean bucket speed of 12 m/s and is supplied with water at a rate of 750 litres per second under a head of 35 m. If the bucket deflects the jet through an angle of 160°, find the power developed by the turbine and its hydraulic efficiency. Take the coefficient of velocity as 0.98. Neglect friction in the bucket. Also determine the overall efficiency of the turbine if its mechanical efficiency is 80%.*

Solution

From Eq. 21.10 the power developed by the turbine is given as

$$P = \frac{W}{g}\left[(V-u)(1+k\cos\phi)\right]u$$

$$W = wQ$$

$$w = 9810 \text{ N/m}^3;$$

and $$Q = 750 \text{ l/s} = \frac{750}{1000} = 0.75 \text{ m}^3/\text{s}$$

$\therefore$ $$W = 9810 \times 0.75 = 7357.5 \text{ N/s}$$

$$V = 0.98\sqrt{2 \times 9.81 \times 35} = 25.68 \text{ m/s}$$

$$u = 12 \text{ m/s}$$

$$k = 1 \text{(for neglecting the friction in the buckets)}$$

$$\phi = (180° - \theta)$$

$$= (180° - 160°) = 20°$$

$\therefore$ $$\cos\phi = \cos 20° = 0.9397$$

Thus by substitution, we get

$$P = \frac{7357.5}{9.81}\left[(25.68-12)(1+0.9397)\right] \times 12 \text{ W}$$

$$= 238816 \text{ W}$$

$$= \frac{238816}{1000} \text{ kW}$$

$$= 238.816 \text{ kW}$$

Since 1 metric h.p. = 735.5 W, the power developed by the turbine in metric h.p. is

$$P = \frac{238816}{735.5} = 324.699 \text{ metric h.p.}$$

From Eq. 21.11 the hydraulic efficiency of the turbine is given as

$$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{2\times 12(25.68-12)(1+0.9397)}{(25.68)^2}$$

$$= 0.966 \text{ or } 96.6\%$$

The overall efficiency of the turbine is given by Eq. 21.9 as

$$\eta_0 = \eta_h \times \eta_m$$

$$\eta_m = 80\% \text{ or } 0.80$$

$$\therefore \quad \eta_0 = 0.966 \times 0.80$$

$$= 0.773 \text{ or } 77.3\%$$

Example 21.5. *The following are the design particulars of a large Pelton turbine:*

(i) Head at the distributor = 630 m
(ii) Discharge = 12.5 m^3/s
(iii) Power developed = 65 MW
(iv) Speed of rotation = 500 r.p.m.
(v) Runner diameter = 1.96 m
(vi) Number of jets = 4
(vii) Jet diameter = 0.192 m
(viii) Angle through which the jet is deflected by the bucket = 165°
(ix) Mechanical efficiency of the turbine = 96%
(a) Determine the hydraulic power losses in the distributor-nozzle assembly and the buckets.

(b) If the loss in the buckets is given to be proportional to V_r^2, where V_r is the relative velocity at inlet, determine the best speed of rotation for this head and discharge.

Solution

(a) From Eq. 21.17, mechanical efficiency of Pelton turbine is given as

$$\eta_m = \frac{P}{W\left[\frac{(V-u)u(1+k\cos\phi)}{g}\right]}$$

η_m = 96% or 0.96; and P = 65 MW = 65 × 10^6 W

Thus by substitution, we get

$$0.96 = \frac{65\times 10^6}{W\left[\frac{(V-u)u(1+k\cos\phi)}{g}\right]}$$

or $$W\left[\frac{(V-u)u(1+k\cos\phi)}{g}\right] = \frac{65\times 10^6}{0.96} = 67.71\times 10^6 \text{ W}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

which is the power developed by the turbine runner.

Power supplied to the turbine runner at the distributor is

$$P_1 = wQH$$

$w = 9810\ \text{N/m}^3$; $Q = 12.5\ \text{m}^3/\text{s}$; and $H = 630\ \text{m}$

Thus by substitution, we get

$$P_1 = 9810 \times 12.5 \times 630\ \text{W} = 77.25 \times 10^6\ \text{W}$$

$\therefore$ Power losses in the distributor-nozzle assembly and the buckets

$$= (P_1 - P)$$
$$= (77.25 \times 10^6 - 67.71 \times 10^6)\ \text{W}$$
$$= 9.54 \times 10^6\ \text{W}$$

(b) The head loss in the buckets is given to be proportional to V_r^2, i.e.,

$$h_L = CV_r^2 = C(V-u)^2$$

where C is constant of proportionality

Total disharge $= 12.5\ \text{m}^3/\text{s}$

No. of jets $= 4$

$\therefore$ Discharge through each jet $= \dfrac{12.5}{4}\ \text{m}^3/\text{s}$

Diameter of jet $= 0.192\ \text{m}$

$$\therefore \quad V = \frac{12.5}{4 \times \frac{\pi}{4} \times (0.192)^2} = 107.93\ \text{m/s}$$

$$u = \frac{\pi DN}{60}$$

$D = 1.96\ \text{m}$; and $N = 500$ r.p.m

$$\therefore \quad u = \frac{\pi \times 1.96 \times 500}{60} = 51.31\ \text{m/s}$$

Thus by substitution, we get

$$h_L = C(107.93\text{-}51.31)^2 = 3206\ C \qquad \text{...(i)}$$

The best speed of rotation is the one which would provide maximum hydraulic efficieny, for which the condition is given by Eq. 21.12 as

$$u = \frac{V}{2}; \quad \text{or } V = 2u$$

$$\therefore \quad h_L = C(2u-u)^2 = Cu^2 \qquad \text{...(ii)}$$

From Eqs (i) and (ii), we have

$$Cu^2 = 3206\ C$$

or $$u = 56.62\ \text{m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus if N is the best speed of rotation then

$$u = \frac{\pi DN}{60} = 56.62$$

or $$\frac{\pi \times 1.96 \times N}{60} = 56.62$$

$\therefore$ $N = 552$ r.p.m.

Example 21.6. *A Pelton wheel produces 1000 hp under a gross head of 200 m. Its nozzle has a diameter of 10 cm and the losses in pipe line due to friction amount to 90 Q^2 where Q is the disharge in m^3/s. Assuming the gross head and efficiency of the wheel to be constant and C_v for the nozzle as 0.98, find the disharge and overall efficiency.*

If the power produced is reduced to 800 hp (i) by operating the needle in the nozzle and (ii) by closing the valve provided in the main, determine the discharge in either case. What is the additional head loss in case (ii)? Comment on the results.

Solution

Net head $H = (200 - 90\ Q^2)$

$\therefore$ $V = 0.98\sqrt{2 \times 9.81 \times (200 - 90\ Q^2)}$

Assuming jet diameter to be same as nozzle diameter, we have

$$Q = \frac{\pi}{4} \times (0.1)^2 \times 0.98\sqrt{2 \times 9.81 \times (200 - 90\ Q^2)}$$

Solving for Q, we get

$Q = 0.465\ m^3/s$

Power $P = 1000 \times 746$ W (since 1 hp = 746 W)

Net head $H = [200 - 90 \times (0.465)^2] = 180.54$ m

$$\eta_0 = \frac{p}{wQH}$$

$$= \frac{1000 \times 746}{9810 \times 0.465 \times 180.54}$$

$= 0.906$ or 90.6%

(i) When the power produced is reduced to 800 hp by operating the needle in the nozzle, then since power developed $P \propto Q \propto d^2$, where d is the diameter of the jet, the required jet diameter

$$d = 10 \times \left(\frac{800}{1200}\right)^{1/2}$$

$= 8.16$ cm $= 0.0816$ m

It is assumed that there is no loss of head in the nozzle, and if Q is the required discharge in this case then, we have

$$Q = \frac{\pi}{4} \times (0.0816)^2 \times 0.98\sqrt{2 \times 9.81 \times (200 - 90\ Q^2)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solving for Q, we get

$$Q = 0.314 \text{ m}^3/\text{s}$$

(ii) When the power produced is reduced to 800 hp by closing the valve provided in the main then the jet diameter remains the same as 10 cm, but there will be additional head loss at the valve. Thus if h_L is the loss of head at the valve then

Net head $H = (200 - 90\ Q^2 - h_L)$

$$\therefore \quad V = 0.98\sqrt{2\times 9.81\times(200-90Q^2-h_L)}$$

and
$$Q = \frac{\pi}{4}\times(0.1)^2\times 0.98\sqrt{2\times 9.81\times(200-90\ Q^2-h_L)} \quad ...(i)$$

Assuming the efficiency to remain constant, we have

$$0.906 = \frac{800\times 746}{9810\times Q\times(200-90\ Q^2-h_L)}$$

or
$$\left(200-90Q^2-h_L\right) = \frac{800\times 746}{0.906\times 9810\times Q} = \frac{67.148}{Q} \quad ...(ii)$$

From Eqs (i) and (ii), we get

$$Q = \frac{\pi}{4}\times(0.1)^2\times 0.98\sqrt{\frac{2\times 9.81\times 67.148}{Q}}$$

Solving for Q, we get

$$Q = 0.427 \text{ m}^3/\text{s}$$

Introducing this value of Q in Eq. (ii), we get

$$200-90(0.427)^2-h_L = \frac{67.148}{0.427}$$

or
$$h_L = 26.34 \text{ m}$$

i.e., the additional head loss in case (ii) is 26.34 m

The provision of a valve in the main to reduce the discharge when the power produced is reduced, results in additional head loss and also requires more discharge, and hene it is not a satisfactory arrangement. On the other hand the provision of a needle in the nozzle and its operation to reduce the discharge when the power produced is reduced, results in negligible head loss and requires less discharge, and hence it is a satisfactory arrangement which is commonly adopted.

Example 21.7 *A pipeline 1200 m long supplies water to 3 single jet Pelton wheels. The head above the nozzle is 360 m. The velocity coefficient for the nozzle is 0.98 and the coefficient of friction for the pipeline is 0.02. The turbine efficiency based on the head at the nozzle is 0.85. The specific speed of each turbine is 15.3 (in m, kW, r.p.m., units) and the head lost due to friction in the pipeline is 12 m of water. If the operating speed of each turbine is 560 r.p.m., determine:*

(i) the total power developed.

(ii) the diameter of each nozzle.

(iii) the diameter of the pipeline.

(iv) volume of water used per second.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Net head available at the base of the nozzle

$$= (360 - 12) = 348 \text{ m}$$

As indicated in Chapter 22

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

By substitution, we have

$$15.3 = \frac{560\sqrt{P}}{(348)^{5/4}}$$

∴ Power developed by each turbine is

$$P = \left[\frac{15.3 \times (348)^{5/4}}{560}\right]^2 = 1686.37 \text{ kW}$$

∴ Total power developed by 3 turbines

$$= (3 \times 1686.37) = 5\,059.11 \text{ kW}$$

Also $$\eta_0 = \frac{P}{wQH}$$

By substitution, we have

$$0.85 = \frac{1686.37 \times 10^3}{9810 \times Q \times 348}$$

∴ Discharge required by each turbine is

$$Q = 0.581 \text{ m}^3/\text{s}$$

∴ Total volume of water used per second by all the three turbines = $(3 \times 0.581) = 1.743 \text{ m}^3$, i.e., total

$$Q = 1.743 \text{ m}^3/\text{s}$$

$$V = K_v\sqrt{2gH}$$

$$= 0.98\sqrt{2 \times 9.81 \times 348} = 80.98 \text{ m/s}$$

Thus if diameter of the nozzle is d then

$$0.581 = \frac{\pi}{4} d^2 \times 80.98$$

$$d = 0.0956 \text{ m} = 95.6 \text{ mm}$$

Further $$h_f = \frac{flv^2}{2gD} = \frac{flQ^2}{2g(\pi/4)^2 D^5}$$

or $$12 = \frac{0.02 \times 1200 \times (1.743)^2}{2 \times 9.81 \times (\pi/4)^2 D^5}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore$ Diameter of pipe $D = 0.871$ m = 871 mm

Example 21.8 *The following data were obtained from a test on a Pelton wheel:*

(a) Head at the base of the nozzle = 32 m

(b) Discharge of the nozzle = 0.18 m^3/s

(c) Area of the jet = 7500 sq. mm

(d) Power available at the shaft = 44 kW

(e) Mechanical efficiency = 94 %

Calculate the power lost (i) in the nozzle; (ii) in the runner; (iii) in mechanical friction.

Solution

Power at the base of the nozzle

$= wQH$

$= (9\,810 \times 0.18 \times 32)$

$= 56\,510$ W = 56.51 kW

Velocity of flow through the nozzle

$$V = \frac{0.18}{7500 \times 10^{-6}} = 24 \text{ m/s}$$

Power at the nozzle exit (i.e., Kinetic energy of the jet)

$$= \frac{waV^3}{2g}$$

$$= \left[\frac{9810 \times 7500 \times 10^{-6} \times (24)^3}{2 \times 9.81}\right]$$

$= 51\,840$ W = 51.84 kW

$\therefore$ Power lost in the nozzle

$= (56.51 - 51.84) = 4.67$ kW

Power supplied to the runner is equal to the kinetic energy of the jet

$= 51.84$ kW

Power developed by the runner

$$= \frac{44}{0.94} = 46.81 \text{ kW}$$

$\therefore$ Power lost in the runner

$= (51.84 - 46.81) = 5.03$ kW

Power lost in mechanical friction

$= (46.81 - 44) = 2.81$ kW

As a check on computation, the difference of power at the base of the nozzle and the power available at the shaft must be equal to the sum of the power lost in the nozzle, in the runner and in mechanical friction.

Thus, we have

$(56.51 - 44) = 12.51$ kW

and $(4.67 + 5.03 + 2.81) = 12.51$ kW

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 21.9 *An inward flow turbine (reaction type with radial discharge) with an overall efficiency of 80% is required to develop 150 kW. The head is 8 m; peripheral velocity of the wheel is* $0.96\sqrt{2gH}$ *; the radial velocity of the flow is* $0.36\sqrt{2gH}$ *. The wheel is to make 150 r.p.m., and the hydraulic losses in the turbine are 22% of the available energy. Determine:*

(i) the angle of the guide blade at inlet
(ii) the wheel vane angle at inlet
(iii) the diameter of the wheel
(iv) the width of the wheel at inlet

Solution

$$\eta_o = 80\%,\ H = 8\text{ m},\ P = 150\text{ kW},\ N = 150\text{ r.p.m.}$$

Peripheral velocity $u = 0.96\sqrt{2gH}$

$$= 0.96\sqrt{2\times 9.81\times 8} = 12.03\text{ m/s}$$

Velocity of flow at inlet

$$V_f = 0.36\sqrt{2gH}$$

$$= 0.36\sqrt{2\times 9.81\times 8} = 4.51\text{ m/s}$$

Hydraulic losses = 22%

$\therefore \quad \eta_h = (100 - 22) = 78\ \%$

For radial discharge

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.78 = \frac{V_w \times 12.03}{9.81\times 8}$$

$\therefore \quad V_w = 5.09$ m/s

From inlet velocity triangle, we have

$$\tan\alpha = \frac{V_f}{V_w} = \frac{4.51}{5.09} = 0.8861$$

$\therefore \quad \alpha = 41^\circ\ 33'$

Also $$\tan(180 - \theta) = \frac{V_f}{u - V_w} = \frac{4.51}{12.03 - 5.09} = 0.6499$$

$\therefore \quad \theta = 147^\circ$

$$u = \frac{\pi DN}{60}$$

or $$12.03 = \frac{\pi \times D \times 150}{60}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad D = 1.532 \text{ m}$$

Also $$\eta_0 = \frac{P}{wQH}$$

or $$0.80 = \frac{150 \times 10^3}{9810 \times Q \times 8}$$

$$\therefore \quad Q = 2.389 \text{ m}^3/\text{s}$$

Neglecting the vane thickness, we have

$$Q = \pi DBV_f$$

or $$2.389 = \pi \times 1.532 \times B \times 4.51$$

$$\therefore \quad B = \frac{2.389}{\pi \times 1.532 \times 4.51}$$

$$= 0.110 \text{ m} = 110 \text{ mm}$$

Example 21.10. *An inward flow reaction turbine discharges radially and the velocity of flow is constant and equal to the velocity of discharge from the turbine. Show that the hydraulic efficiency can be expressed by*

$$\eta_h = \frac{1}{1 + \dfrac{\frac{1}{2}(\tan^2\alpha)}{\left(1 - \dfrac{\tan\alpha}{\tan\theta}\right)}}$$

where α and θ are respectively the guide vane angle and wheel vane angle at inlet.

Solution

From the inlet velocity triangle

$$V_f = V_w \tan\alpha$$

$$u = V_w - \frac{V_f}{\tan\theta}$$

$$= V_w - \frac{V_w \tan\alpha}{\tan\theta} = V_w\left(1 - \frac{\tan\alpha}{\tan\theta}\right)$$

For radial discharge at outlet $V_{w1} = 0$ and $V_1 = V_{f1}$

Thus $$\frac{V_w u}{g} = H - \frac{V_1^2}{2g}$$

or $$H = \frac{V_w u}{g} + \frac{V_1^2}{2g}$$

$$= \frac{V_w u}{g} + \frac{V_{f1}^2}{2g} = \frac{V_w u}{g} + \frac{V_f^2}{2g}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the value of V_f, we get

$$H = \frac{V_w u}{g} + \frac{V_w^2 \tan^2 \alpha}{2g}$$

Also substituting the value of u from above, we get

$$H = \frac{V_w^2}{g}\left(1 - \frac{\tan\alpha}{\tan\theta}\right) + \frac{V_w^2 \tan^2 \alpha}{2g}$$

$$= \frac{V_w^2}{g}\left(1 - \frac{\tan\alpha}{\tan\theta} + \frac{\tan^2 \alpha}{2}\right)$$

Now $$\eta_h = \frac{V_w u}{gH} = \frac{V_w^2\left(1 - \frac{\tan\alpha}{\tan\theta}\right)}{gH}$$

or $$\eta_h = \frac{\left(1 - \frac{\tan\alpha}{\tan\theta}\right)}{\left(1 - \frac{\tan\alpha}{\tan\theta} + \frac{\tan^2\alpha}{2}\right)}$$

or $$\eta_h = \frac{1}{1 + \frac{\frac{1}{2}(\tan^2\alpha)}{1 - \frac{\tan\alpha}{\tan\theta}}}$$

Example 21.11. *Design a Francis turbine runner with the following data: Net head H = 68 m; speed N = 750 r.p.m; output power P = 330 kW; η_h = 94%; η_o = 85 %; flow ratio ψ = 0.15; breadth ratio n = 0.1; inner diameter of runner is $\left(\frac{1}{2}\right)$ outer diameter. Also assume 6% of circumferential area of the runner to be occupied by the thickness of the vanes. Velocity of flow remains constant throughout and flow is radial at exit.*

Solution

$$\eta_0 = \frac{P}{wQH}$$

or $$0.85 = \frac{330 \times 10^3}{9810 \times Q \times 68}$$

$\therefore$ $$Q = 0.582 \text{ m}^3/\text{s}$$

$$V_f = \psi\sqrt{2gH}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 0.15 \times \sqrt{2 \times 9.81 \times 68} = 5.48 \text{ m/s}$$

Also $Q = (k\pi n D^2) V_f$

or $0.582 = 0.94 \times \pi \times 0.10 \times D^2 \times 5.48$

$\therefore D = 0.60 \text{ m} = 600 \text{ mm}$

Since $n = (B/D)$; $B = (0.10 \times 600) = 60 \text{ mm}$

$$u = \frac{\pi DN}{60} = \frac{\pi \times 0.60 \times 750}{60} = 23.56 \text{ m/s}$$

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.94 = \frac{V_w \times 23.56}{9.81 \times 68}$$

$\therefore V_w = 26.62 \text{ m/s}$

From inlet velocity triangle, we have

$$\tan \alpha = \frac{V_f}{V_w} = \frac{5.48}{26.62} = 0.2059$$

$\therefore \alpha = 11°38'$

$$\tan \theta = \frac{V_f}{V_w - u} = \frac{5.48}{(26.62 - 23.56)} = 1.7908$$

$\therefore \theta = 60° 49'$

Now $D_1 = \frac{D}{2} = 0.30 \text{ m} = 300 \text{ mm}$

$$\frac{u}{D} = \frac{u_1}{D_1} \; ; u_1 = 11.78 \text{ m/s}$$

Since the flow is radial at exit and $V_f = V_{f1}$, from outlet velocity triangle

$$\tan \phi = \frac{V_{f_1}}{u_1} = \frac{5.48}{11.78} = 0.4652$$

$\therefore \phi = 24° 57'$

Example 21.12. *A Kaplan turbine produces 60 000 kW under a net head of 25 m with an overall efficiency of 90%. Taking the value of speed ratio Ku as 1.6, flow ratio ψ as 0.5 and the hub diameter as 0.35 times the outer diameter, find the diameter and speed of the turbine.*

Solution

$$\eta_0 = \frac{P}{wQH}$$

or $$0.90 = \frac{60000 \times 10^3}{9810 \times Q \times 25}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad Q = 271.83 \text{ m}^3/\text{s}$$

$$Q = \frac{\pi}{4}(D^2 - d^2)\psi\sqrt{2gH}$$

or $$271.83 = \frac{\pi}{4} \times D^2 [1-(0.35)^2] \times 0.5 \times \sqrt{2\times 9.81\times 25}$$

$$\therefore \quad D = 5.97 \text{ m}$$

$$u = K_u\sqrt{2gH}$$

$$= 1.6 \times \sqrt{2\times 9.81\times 25} = 35.44 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$

or $$35.44 = \frac{\pi\times 5.97\times N}{60}$$

$$\therefore \quad N = 113.38 \text{ r.p.m.}$$

Example 21.13. *Determine the efficiency of a Kaplan turbine developing 3000 kW under a net head of 5 m. It is provided with a draft tube with its inlet (diameter 3 m) set 1.6 m above the tail race level. A vacuum gage connected to the draft tube indicates a reading of 5 m of water. Assume draft tube efficiency as 78%.*

Solution

Neglecting the head loss in the draft tube, from Eq. 21.32, we have

$$\frac{P_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right]$$

or $$-5 = -\left[1.6 + \frac{V_2^2 - V_3^2}{2g}\right]$$

or $$\frac{V_2^2 - V_3^2}{2g} = 3.4$$

From Eq. 21.34, we have

$$\eta_d = \frac{(V_2^2 - V_3^2)/2g}{V_2^2/2g}$$

or $$0.78 = \frac{3.4}{V_2^2/2g}$$

$$\therefore \quad V_2 = 9.25 \text{ m/s}$$

$$Q = \left[\frac{\pi}{4}\times(3)^2\times 9.25\right] = 65.38 \text{ m}^3/\text{s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\eta_0 = \frac{P}{wQH}$$

$$= \frac{3000\times10^3}{9810\times65.38\times5}$$

$$= 0.9355 = 93.55\%$$

Example 21.11. *A propeller turbine runner has outer diameter of 4.5 m, and the diameter of the hub 2 m. It is required to develop 20 600 kW when running at 150 r.p.m., under a head of 21 m. Assuming hydraulic efficiency of 94% and overall efficiency of 88%, determine the runner vane angles at inlet and exit at the mean diameter of the vanes. Also determine the runner vane angles at inlet and exit at two sections-near the hub and the outer periphery.*

Solution

$$\eta_0 = \frac{P}{wQH}$$

or $$0.88 = \frac{20600\times10^3}{9810\times Q\times21}$$

$\therefore$ $$Q = 113.63 \text{ m}^3/\text{s}$$

From Eq. 21.35, we have

$$Q = \frac{\pi}{4}(D^2 - d^2)\, V_f$$

or $$113.63 = \frac{\pi}{4}\left[(4.5)^2 - (2)^2\right] V_f$$

or $$V_f = 8.90 \text{ m/s}$$

(i) Computations for mean diameter.

Mean diameter $$D_m = \left(\frac{4.5+2}{2}\right) = 3.25 \text{ m}$$

$$u_m = \frac{\pi D_m N}{60}$$

$$= \frac{\pi\times3.25\times150}{60} = 25.53 \text{ m/s}$$

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.94 = \frac{V_w\times25.53}{9.81\times21}$$

or $$V_w = 7.59 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From inlet velocity triangle, we have

$$\tan(180° - \theta) = \frac{V_f}{u_m - V_w}$$

$$= \frac{8.90}{(25.53 - 7.59)} = 0.4961$$

$$\therefore \quad \theta = 153°37'$$

From outlet velocity triangle, since

$$V_{f1} = V_f \text{ and } u_1 = u,$$

$$\tan \phi = \frac{V_{f_1}}{u_1}$$

$$= \frac{8.90}{25.53} = 0.348\ 6$$

$$\therefore \quad \phi = 19°\ 13'$$

(ii) Computations for section near the hub.

$$u_i = \frac{\pi D_i N}{60}$$

$$= \frac{\pi \times 2 \times 150}{60} = 15.71 \text{ m/s}$$

$$\eta_h = 0.94 = \frac{V_w \times 15.71}{9.81 \times 21}$$

or $$V_w = 12.33 \text{ m/s}$$

From inlet velocity triangle, we have

$$\tan(180° - \theta) = \frac{V_f}{u_i - V_w}$$

$$= \frac{8.90}{(15.71 - 12.33)} = 2.6331$$

$$\therefore \quad \theta = 110°\ 48'$$

From outlet velocity triangle, we have

$$\tan \phi = \frac{V_{f1}}{u_i}$$

$$= \frac{8.90}{15.71} = 0.5665$$

$$\therefore \quad \phi = 29°\ 32'$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) Computations for section at the outer periphery.

$$u_0 = \frac{\pi D_0 N}{60}$$

$$= \frac{\pi \times 4.5 \times 150}{60} = 35.34 \text{ m/s}$$

$$\eta_h = 0.94 = \frac{V_w \times 35.34}{9.81 \times 21}$$

or $$V_w = 5.48 \text{ m/s}$$

From inlet velocity triangle, we have

$$\tan (180° - \theta) = \frac{V_f}{u_0 - V_w}$$

$$= \frac{8.90}{(35.34 - 5.48)} = 0.2981$$

$$\therefore \quad \theta = 163° 24'$$

From outlet velocity triangle, we have

$$\tan \phi = \frac{V_{f_1}}{u_0}$$

$$= \frac{8.90}{35.34} = 0.2518$$

$$\therefore \quad \phi = 14° 8'$$

Example 21.15. *The velocity of whirl at inlet to the runner of an inward flow reaction turbine is* $\left(3.15\sqrt{H}\right)$ *m/s and the velocity of flow at inlet is* $\left(1.05\sqrt{H}\right)$ *m/s. The velocity of whirl at exit is* $\left(0.22\sqrt{H}\right)$ *m/s in the same direction as at inlet and the velocity of flow at exit is* $\left(0.83\sqrt{H}\right)$ *m/s, where H is the head in metres. The inner diameter of the runner is 0.6 times the outer diameter. Assuming hydraulic efficiency of 80%, compute the angles of the runner vanes at inlet and exit.*

Solution

$$\eta_h = \frac{(V_w u - V_{w1} u_1)}{gH} \; ; \left(\frac{u}{D}\right) = \left(\frac{u_1}{D_1}\right)$$

or $$0.8 = \frac{[(3.15\sqrt{H})u - (0.22\sqrt{H})0.6u]}{9.81H}$$

$$\therefore \quad u = \left(2.60\sqrt{H}\right) \text{ and } u_1 = \left(1.56\sqrt{H}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From inlet velocity triangle, we have

$$\tan\theta = \frac{V_f}{(V_w - u)}$$

$$= \frac{1.05\sqrt{H}}{(3.15\sqrt{H}) - (2.60\sqrt{H})} = 1.9091$$

$$\therefore \quad \theta = 62° 21'$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{(u_1 - V_{w1})}$$

$$= \frac{0.83\sqrt{H}}{(1.56\sqrt{H}) - (0.22\sqrt{H})} = 0.619\ 4$$

$$\therefore \quad \phi = 31° 46'$$

Example 21.16. *A straight conical draft tube attached to a Francis turbine has an inlet of diameter 3 m and its outlet area is 20 m^2. The velocity of water at inlet is 5 m/s. The inlet is set 5 m above the tail race level. Assuming the loss of head in the draft tube equals half the velocity head at its outlet, determine (i) the pressure head at the top of the draft tube, (ii) total head at the top taking tail race level as datum, (iii) power of water at outlet of runner, (iv) power of water at the end of the draft tube, (v) the power lost in the draft tube.*

Solution

From Eq. 21.32, we have

$$\frac{p_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + h_f$$

$$h_f = \frac{1}{2}\frac{V_3^2}{2g}$$

Thus

$$\frac{p_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + \frac{1}{2}\frac{V_3^2}{2g}$$

By continuity, we have

$$\frac{\pi \times (3)^2}{4} \times V_2 = 20 \times V_3$$

or

$$V_3 = 0.353\ 4\ V_2$$

$$= (0.353\ 4 \times 5) = 1.767 \text{ m/s}$$

$$\therefore \quad \frac{p_2}{w} = -\left[5 + \frac{(5)^2 - (1.767)^2}{2 \times 9.81}\right] + \frac{1}{2}\frac{(1.767)^2}{2 \times 9.81}$$

$$= -6.035\ 5 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Total head at the top of the draft tube is equal to

$$\frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 = \left[-6.0355 + \frac{(5)^2}{2\times 9.81} + 5\right]$$

$$= 0.2387 \text{ m}$$

$$Q = \left[\frac{\pi\times(3)^2}{4}\times 5\right] = 35.34 \text{ m}^3/\text{s}$$

∴ Power at the outlet of the runner

$$= (9\,810 \times 35.34 \times 0.2387)$$

$$= 82\,754 \text{ W} = 82.754 \text{ kW}$$

Total head at the end of the draft tube is equal to

$$\frac{V_3^2}{2g} = \frac{(1.767)^2}{2\times 9.81} = 0.1591 \text{ m}$$

∴ Power at the end of the draft tube

$$= (9810 \times 35.34 \times 0.1591)$$

$$= 55\,158 \text{ W} = 55.158 \text{ kW}$$

Head lost in the draft tube is equal to

$$\frac{1}{2}\frac{V_3^2}{2g} = \frac{1}{2}\frac{(1.767)^2}{2\times 9.81} = 0.079\,6 \text{ m}$$

∴ Power lost in the draft tube

$$= (9810 \times 35.34 \times 0.0796)$$

$$= 27\,596 \text{ W} = 27.596 \text{ kW}$$

Note. Power lost in the draft tube can also be determined by subtracting the power at the end of the draft tube from the power at the outlet of the runner.

Example 21.17. *In a Francis turbine of very low specific speed, the velocity of flow from inlet to exit of the runner remains constant. If the turbine discharges radially, show that the degree of reaction ρ can be expressed as*

$$\rho = \frac{1}{2} - \frac{1}{2}\left[\frac{\cot\theta}{(\cot\alpha - \cot\theta)}\right]$$

where α and θ are the guide and runner vane angles respectively and the degree of reaction ρ is equal to the ratio of pressure drop to the hydraulic work done in the runner, assuming that the losses in the runner are negligible.

Solution

Applying Bernoulli's equation between the inlet and exit of the runner and neglecting the potential difference, we get

$$\frac{p}{w} + \frac{V^2}{2g} = \frac{p_1}{w} + \frac{V_1^2}{2g} + \frac{V_w u}{g} \qquad \text{(for radial discharge)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where $\frac{p}{w}$ and $\frac{p_1}{w}$ are the pressure heads at the inlet and the exit of the runner respectively.

Thus pressure head drop due to hydraulic work done in the runner is given by

$$\frac{p}{w} - \frac{p_1}{w} = \frac{V_1^2}{2g} - \frac{V^2}{2g} + \frac{V_w u}{g}$$

Now
$$\rho = \frac{\left(\frac{p}{w} - \frac{p_1}{w}\right)}{\frac{V_w u}{g}}$$

or
$$\rho = \frac{\frac{V_1^2}{2g} - \frac{V^2}{2g} + \frac{V_w u}{g}}{\frac{V_w u}{g}}$$

or
$$\rho = 1 + \frac{1}{2}\left[\frac{V_1^2 - V^2}{V_w u}\right] \quad \ldots(i)$$

For radial discharge

$$V_1 = V_{f1} = V_f$$

also
$$V_w = V \cos \alpha$$
$$V_f = V \sin \alpha$$
$$u = V_w - V_f \cot \theta$$

or
$$u = V [\cos \alpha - \sin \alpha \cot \theta]$$

and
$$V_1 = V_f = V \sin \alpha$$

Thus, introducing these values in Eq. (i) above and simplifying it, we get

$$\rho = \frac{1}{2} - \frac{1}{2}\left[\frac{\cot \theta}{\cot \alpha - \cot \theta}\right]$$

Example 21.18. *The inlet area of a draft tube provided for a Francis turbine is X times the exit area of the tube. Head loss in the tube = (K V^2), where V is equal to square root of the product of velocities at inlet and exit of the draft tube. Show that draft tube efficiency is given by the expression* $\eta_d = [1 - X^2 - 2g\,KX]$.

Solution

From Eq. 21.34, we have

$$\eta_d = \frac{\left[(V_2^2 - V_3^2)/2g\right] - h_f}{\left[V_2^2/2g\right]}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_3 = XV_2; h_f = K(V_2 \times V_3) = KXV_2^2$$

$\therefore$ By substitution, we get

$$\eta_d = [1 - X^2 - 2gKX]$$

Example 21.19. *Full load is supplied by the turbine shaft when the diameter of jet issuing from the nozzle is 150 mm. If the load suddenly drops to 36% of the full load what diameter of the jet should be attained by regulating the spear rod?*

Solution

Power developed, $P \propto Q \propto d^2$; where d is the diameter of the jet.

$\therefore$ Required jet diameter

$$= 150\left(\frac{36}{100}\right)^{1/2} = 90 \text{ mm}$$

Example 21.20. *A reaction turbine works at 450 r.p.m under a head of 120 m. Its diameter at inlet is 1.2 m and the flow area is 0.4 m^2. The angles made by the absolute and relative velocities at inlet are 20° and 60° respectively with the tangential velocity. Determine the volume flow rate, the power developed and the hydraulic efficiency. Assume whirl at outlet to be zero.*

Solution

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 1.2 \times 450}{60} = 28.27 \text{ m/s}$$

From inlet velocity triangle, we have

$$V_f \cot \alpha = u + V_f \cot \theta$$

Thus by substitution, we get

$$V_f \cot 20° = 28.27 + V_f \cot 60°$$

$\therefore$ $V_f = 13.03$ m/s

$\therefore$ $Q = (0.4 \times 13.03) = 5.212 \text{ m}^3/\text{s}$

Work done by the runner per second is equal to the power developed P, which is thus given by

$$P = wQ\left(\frac{V_w u}{g}\right)$$

$$V_w = V_f \cot 20°$$

$$= (13.03 \times 2.7475) = 35.8 \text{ m/s}$$

$\therefore$ $$P = (9810 \times 5.212) \times \left(\frac{35.8 \times 28.27}{9.81}\right)$$

$$= 5\,275 \times 10^3 \text{ W}$$

$$= 5\,275 \text{ kW}$$

Hydraulic efficiency is given by

$$\eta_h = \frac{V_w u}{gH}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{35.8 \times 28.27}{9.81 \times 120}$$

$$= 0.86 \text{ or } 86\%$$

Example 21.21. *The casing diameter of a propeller turbine is 5 m and its hub is of 1.7 m diameter. When the turbine is operating at a speed of 140 r.p.m., under a head of 20.5 m and discharging 160 m^3/s, the power developed is 25800 kW. At this regime the axial flow velocity is constant everywhere. At the inlet of the runner blade cascade the tangential velocity varies with radius and is equal to $\frac{12.4}{R}$ m/s where R is radius in m. At the exit of the blade cascade the tangential velocity is (0.3 R – 0.25) m/s.*

Determine, based on one-dimensional theory, the blade angles at inlet and exit at the hub and the tip. Determine the head utilized by the runner, the hydraulic and mechanical efficiencies.

Solution

The discharge Q through the runner is given by Eq. 21.35 as

$$Q = \frac{\pi}{4}\left(D^2 - d^2\right) V_f$$

$$Q = 160 \text{ m}^3/\text{s}; D = 5 \text{ m and } d = 1.7 \text{ m}$$

Thus by substitution, we get

$$160 = \frac{\pi}{4}\left[(5)^2 - (1.7)^2\right] V_f$$

$$\therefore \quad V_f = 9.21 \text{ m/s}$$

(i) **Computations for section near the hub**

$$u_i = \frac{\pi D_i N}{60}$$

$$= \frac{\pi \times 1.7 \times 140}{60} = 12.46 \text{ m/s}$$

$$\therefore \quad \theta = 76°59'$$

Tangential velocity V_w at inlet is given by

$$V_w = \frac{12.4}{R}$$

$$R = R_i = \frac{1.7}{2} = 0.85 \text{ m}$$

$$\therefore \quad V_w = \frac{12.4}{0.85} = 14.59 \text{ m/s}$$

From inlet velocity triangle, we have

$$\therefore \quad \tan\theta = \frac{V_f}{(V_w - u_i)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{9.21}{(14.59-12.46)} = 4.3239$$

$\therefore \quad \theta = 76^\circ\ 59'$

Tangential velocity V_{w_1} at exit is given by

$$V_{w_1} = (0.3\,R - 0.25)$$
$$= [(0.3 \times 0.85) - 0.25] = 0.005 \text{ m/s}$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{u_1 + V_{w_1}}$$

$V_{f_1} = 9.21$ m/s; $u_1 = u_i = 12.46$ m/s

Thus

$$\therefore \quad \tan\phi = \frac{9.21}{(12.46+0.005)} = 0.7389$$

$\therefore \quad \theta = 36^\circ\ 27'$

(ii) Computations for section at the outer periphery

$$u_0 = \frac{\pi D_0 N}{60}$$
$$= \frac{\pi \times 5 \times 140}{60} = 36.65 \text{ m/s}$$

Tangential velocity at inlet is given by

$$V_w = \frac{12.4}{R}$$

$$R = R_0 = \frac{5}{2} = 2.5\,\text{m}$$

$$\therefore \quad V_w = \frac{12.4}{2.5} = 4.96 \text{ m/s}$$

From inlet velocity triangle, we have

$$\tan(180-\theta) = \frac{V_f}{u_0 - V_w}$$
$$= \frac{9.21}{(36.65-4.96)} = 0.2906$$

$\therefore \quad \theta = 163^\circ\ 48'$

Tangential velocity V_{w_1} at exit is given by

$$V_{w_1} = (0.3R - 0.25)$$
$$= [(0.3 \times 2.5) - 0.25] = 0.5 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From outlet velocity triangle, we have

$$\tan \phi = \frac{V_{f_1}}{u_1 - V_{w_1}}$$

$$V_{f_1} = V_f = 9.21 \text{ m/s}; \; u_1 = u_0 = 36.65 \text{ m/s}$$

Thus $$\tan \phi = \frac{9.21}{(36.65 - 0/5)} = 0.2548$$

$$\therefore \quad \phi = 14°18'$$

(iii) Computations for mean diameter

Mean diameter $$D_m = \left(\frac{5 + 1.7}{2}\right) = 3.35 \text{ m}$$

$$u_m = \frac{\pi D_m N}{60}$$

$$= \frac{\pi \times 3.35 \times 140}{60} = 24.56 \text{ m/s}$$

Hydraulic efficiency $$\eta_h = \frac{V_w u - V_{w_1} u_1}{gH}$$

Tangential velocity V_w at inlet is given by

$$V_w = \frac{12.4}{R}$$

$$R = R_m = \frac{3.35}{2} = 1.675 \text{ m}$$

$$\therefore \quad V_w = \frac{12.4}{1.675} = 7.40 \text{ m/s}$$

Tangential velocity V_{w_1} at exit is given by

$$V_{w_1} = (0.3R - 0.25)$$

$$= [(0.3 \times 1.675) - 0.25] = 0.2525 \text{ m/s}$$

$$u = u_1 = u_m = 24.56 \text{ m/s}; \; H = 20.5 \text{ m}$$

Thus by substitution, we get

$$\eta_h = \frac{(7.40 - 0.2525) \times 24.56}{9.81 \times 20.5}$$

$$= 0.873 \text{ or } 87.3\%$$

$\therefore$ Head utilized by the runner = $(0.873 \times 20.5) = 17.9$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Mechanical efficiency

$$\eta_m = \frac{P}{wQ\left(\frac{V_w u - V_{w_1} u_1}{g}\right)}$$

P =25800 kW = 25800 × 10^3 W; w = 9810 N/m^3; Q = 160 m^3/s

$$\left(\frac{V_w u - V_{w_1} u_1}{g}\right) = \eta_h \times H = (0.873 \times 20.5)$$

Thus by substitution, we get

$$\eta_m = \frac{25800 \times 10^3}{9810 \times 160 \times 0.873 \times 20.5}$$

$$= 0.918 \text{ or } 91.8\%$$

Example 21.22. *The draft tube of a Kaplan turbine has inlet diameter 2.6 m and inlet is set at 2.9 m above the tail race. When the turbine develops 2100 metric horse power under a net head of 6.5 m, it is found that the vacuum gage fitted at inlet to draft tube indicates a negative head of 4 m. If the turbine efficiency is 86%, determine the draft tube efficiency. If the turbine output is reduced to half with the same head, speed and draft tube efficiency, what would be the reading of the vacuum gage? Atmospheric pressure is 10.3 m of water and specific weight of water is 1000 kg/m^3.*

Solution

$$\eta_0 = \frac{p}{\frac{wQH}{75}}$$

or $$0.86 = \frac{2100 \times 75}{1000 \times Q \times 6.5}$$

$$\therefore \quad Q = 28.175 \text{ m}^3/\text{s}$$

The velocity at inlet to draft tube is

$$V_2 = \frac{28.175}{(\pi/4) \times (2.6)^2} = 5.307 \text{ m/s}$$

From Eq. 21.33, we have

$$\frac{p_2}{w} = \frac{p_a}{w} - \left[H_s + (1-k)\frac{V_2^2 - V_3^2}{2g}\right]$$

$\frac{p_2}{w}$ = (10.3 – 4) = 6.3 m; $\frac{p_a}{w}$ = 10.3 m; and H_s = 2.9 m

Thus by substitution, we get

$$6.3 = 10.3 - \left[2.9 + (1-k)\frac{V_2^2 - V_3^2}{2g}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$(1-k)\frac{V_2^2 - V_3^2}{2g} = 1.1$$

From Eq. 21.34, efficiency of draft tube is given as

$$\eta_d = \frac{(1-k)\left[\left(V_2^2 - V_3^2\right)/2g\right]}{\left(V_2^2/2g\right)}$$

$$\frac{V_2^2}{2g} = \frac{(5.307)^2}{2\times 9.81} = 1.435 \text{ m}$$

$$\therefore \quad \eta_d = \frac{1.1}{1.435} = 0.767 \text{ or } 76.7\%$$

When the turbine output is reduced to half,

$$P = \frac{2100}{2} = 1050 \text{ metric horse power}$$

Thus with the same head and the efficiency of the turbine, we have

$$0.86 = \frac{1050\times 75}{1000\times Q\times 6.5}$$

$$\therefore \quad Q = 14.088 \text{ m}^3/\text{s}$$

and $$V_2 = \frac{14.088}{(\pi/4)\times(2.6)^2} = 2.653 \text{ m/s}$$

$$\therefore \quad \frac{V_2^2}{2g} = \frac{(2.653)^2}{2\times 9.81} = 0.359 \text{ m}$$

For the same draft tube efficiency, we have

$$0.767 = \frac{(1-k)\left[\left(V_2^2 - V_3^2\right)/2g\right]}{0.359}$$

or $$(1-k)\left[\left(V_2^2 - V_3^2\right)/2g\right] = 0.275$$

Thus from Eq. 21.33, we have

$$\frac{p_2}{w} = 10.3 - [2.9 + 0.275] = 7.125 \text{ m (abs.)}$$

$\therefore$ reading of vacuum gage would be

$$(7.125 - 10.3) = -3.175 \text{ m}$$

i.e., negative head of 3.175 m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 21.23. *A Francis turbine supplied through a 6 m diameter penstock has the following particulars:*

Output of installation	*63500 kW*
Flow 117 m^3/s	
Speed 150 r.p.m	
Hydraulic efficiency	*92%*
Mean diameter of turbine at entry	*4 m*
Mean blade height at entry	*1 m*
Entry diameter of draft tube	*4.2 m*
Velocity in tail race	*2.4 m/s*

The static pressure head in the penstock measured just before entry to the runner is 57.4 m. The point of measurement is 3 m above the level of the tail race. The loss in the draft tube is equivalent to 30% of the velocity head at entry to it. The exit plane of the runner is 2 m above the tail race and the flow leaves the runner without swirl. Determine:

(a) The overall efficiency,

(b)The direction of flow relative to the runner at inlet,

(c)The pressure head at entry to the draft tube.

Solution

(a) The net head H for the turbine is given by Eq. 21.2 as

$$H = \left(\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1\right) - \left(Z_2 + \frac{V_2^2}{2g}\right)$$

$$\frac{p_1}{w} = 57.4 \text{ m}; V_1 = \frac{117}{(\pi/4)\times(6)^2} = 4.14 \text{ m/s}; (Z_1 - Z_2) = 3 \text{ m};$$

and $V_2 = 2.4$ m/s.

Thus by substitution, we get

$$H = \left[57.4 + \frac{(4.14)^2}{2\times 9.81} + 3 - \frac{(2.4)^2}{2\times 9.81}\right] = 60.98 \text{ m}$$

The overall efficiency is given by

$$\eta_0 = \frac{P}{wQH}$$

$$= \frac{63500\times 10^3}{9810\times 117\times 60.98}$$

$$= 0.907 \text{ or } 90.7\%$$

(b) Neglecting the vane thickness, the velocity of flow at inlet is given by Eq. 21.28 as

$$V_f = \frac{Q}{\pi BD}$$

$Q = 117 \text{ m}^3/\text{s}$; $B = 1$ m; and $D = 4$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by substitution, we get

$$V_f = \frac{117}{\pi \times 1 \times 4} = 9.31 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 4 \times 150}{60} = 31.42 \text{ m/s}$$

Hydraulic efficiency

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.92 = \frac{V_w \times 31.42}{9.81 \times 60.98}$$

$$\therefore \quad V_w = 17.52 \text{ m/s}$$

The direction of flow relative to the runner at inlet is given by

$$\tan \theta = \frac{V_f}{u - V_w}$$

$$= \frac{9.31}{(31.42 - 17.52)} = 0.6698$$

$$\therefore \quad \theta = 146°11'$$

(c) The pressure head at entry to the draft tube is given by Eq. 21.32 as

$$\frac{p_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + h_f$$

$$H_s = 2 \text{ m}; V_2 = \frac{117}{(\pi/4) \times (4.2)^2} = 8.44 \text{ m/s}; \; V_3 = 2.4 \text{ m/s}$$

and $$h_f = 0.30\left(\frac{V_2^2}{2g}\right) = 0.30 \times \frac{(8.44)^2}{2 \times 9.81} = 1.09 \text{ m}$$

Thus by substitution, we get

$$\frac{p_2}{w} = -\left[2.0 + \frac{(8.44)^2 - (2.4)^2}{2 \times 9.81}\right] + 1.09$$

$$= -4.25 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. Hydraulic turbines are the machines which use the energy of water (hydro-power) and convert it into electrical power which is known as hydroelectric power.
2. Hydraulic turbine consists of a wheel called runner having a number of specially designed vanes or blades or buckets. The water possessing a large amount of hydraulic energy when strikes the runner, it does work on the runner and causes it to rotate. The mechanical energy so developed is supplied to the generator coupled to the runner, which then generates electrical energy.
3. The water surface in the storage reservoir is known as *head race,* and that in the tail race channel is known as *tail race.*
4. The difference between the head race level and the tail race level is known as gross head H_1.

   The head available at the entrance to the turbine is known as net head H which is given by

   $$H = H_1 - h_f$$

   where

   h_f = total loss of head between the head race and the entrance of the turbine.
5. The various efficiencies of the turbines may be expressed as (i) Hydraulic efficiency, (ii) Machanical efficiency, (iii) Volumetric efficiency and (iv) Overall efficiency.

   Hydraulic efficiency η_h is the ratio of the power developed by the runner to the net power supplied by the water at the entrance to the turbine.

   Mechanical efficiency η_m is the ratio of the power available at the turbine shaft to the power developed by the runner.

   Volumetric efficiency η_v is the ratio of the quantity of water actually striking the runner and the quantity of water supplied to the turbine.

   Overall efficiency η_o is the ratio of the power available at the turbine shaft to the power supplied by the water at the entrance to the turbine.

   The relationship between the various efficiencies is given as

   $$\eta_o = \eta_h \times \eta_v \times \eta_m$$
6. The hydraulic turbines may be classified as (i) Impulse turbines and (ii) Reaction turbines.

   In an *impulse turbine* all the available energy of water is converted into kinetic energy by passing it through a contracting nozzle provided at the end of the penstock which carry water from the storage reservoir to the turbine. The water coming out of the nozzle is formed into a free jet which impingess on a series of buckets of the runner thus causing it to revolve.

   Pelton wheel is the most commonly used impulse turbine.

   In a *reaction turbine* st the entrance to the runner, only a part of the available energy of water is converted into kinetic energy and a substantial part remains in the form of pressure energy.

   Francis turbine and *Kaplan turbine* are the two most commonly used reaction turbines.

   The impulse turbines may be classified as high head turbines, while the reaction turbines are the medium to low head turbines.
7. A Pelton wheel is tangential flow turbine in which the runner consists of circular disc with a number of bucket evenly spaced round its periphery. One or more nozzles are mounted so that each directs a jet along a tangent to the circle through the centres of the buckets called the pitch circle.

   For a Pelton wheel turbine

   (i) The velocity of the jet V is given by

   $$V = K_v \sqrt{2gH}$$

   where

   K_v = coefficient of velocity for the nozzle which ranges from 0.97 to 0.99; and

   H = net head

   (ii) The velocity of the wheel u at pitch circle is given by

   $$u = K_u \sqrt{2gH}$$

   where

   K_u = speed ratio which ranges from 0.43 to 0.47

   (iii) Least diameter d of the jet is given by

   $$d = \left[\frac{4Q}{\pi K_v \sqrt{2gH}}\right]^{1/2}$$

   where

   Q = discharge through the jet.

   (iv) Pitch diameter D is given by

   $$D = \frac{60 K_u \sqrt{2gH}}{\pi N}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where

N = speed of rotation of wheel in r.p.m

(v) The number of buckets Z is given by

$$Z = \left(\frac{D}{2d} + 15\right) = (0.5\text{ m} + 15)$$

where

$m = (D/d)$

8. The hydraulic efficiency of a Pelton wheel is given by

$$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2}$$

where

ϕ = angle of the bucket at the outlet tip; and

k = fraction by which the relative velocity at exit tip of the bucket is reduced.

For the maximum hydraulic efficiency the condition is $u = \frac{V}{2}$; and

$$(\eta_h)_{max} = \frac{1}{2}(1+k\cos\phi)$$

9. Francis turbine is a mixed flow turbine in which water enters the runner at the outer pariphery in the radial direction and leaves it at the centre in the direction parallel to the axis of rotation of the runner.

For a Francis turbine

(i) The ratio of the width B and the diameter D of the runner is represented by n, i.e., $n = (B/D)$ the value of which ranges from 0.10 to 0.45.

(ii) The ratio of the velocity of flow V_f at the inlet tip of vane to the spouting velocity $\left(\sqrt{2gH}\right)$ is known as *flow ratio* ψ, i. e., $\psi = \left(V_f / \sqrt{2gH}\right)$ the value of which ranges from 0.15 to 0.3.

(iii) In this case also the speed ratio K_u is defined as

$$K_u = \frac{u}{\sqrt{2gH}}$$

The value of K_u range from 0.60 to 0.90.

(iv) The discharge through a Francis turbine is given by

$$Q = k\pi BD\,V_f = k\pi\, n\, D^2\, V_f$$

where

k = a factor which allows for the thickness of the vanes.

10. The hydraulic effeciency of a Francis turbine is given by

$$\eta_h = \frac{V_w u}{gH}$$

where

V_w = velocity of whirl at inlet; and

u = tangential velocity at inlet

11. A Kaplan turbine is an axial flow turbine which is suitable for low heads, and hence requires a large quantity of water to develop large amount of power.

For a Kaplan turbine

(i) The ratio of the hub diameter d and runner outside diameter D is represented by n, i..e., $n = (d/D)$, the value of which varies from 0.35 to 0.60.

(ii) The discharge through a Kaplan turbine runner is given by

$$Q = \frac{\pi}{4}\left(D^2 - d^2\right)V_f = \frac{\pi}{4}\left(D^2 - d^2\right)\psi\sqrt{2gH}$$

The value of flow ratio for a Kaplan turbine is around 0.70.

(iii) The runner of a Kaplan turbine has four or six blades which are warped or twisted. The runner blades can be turned about their own axis so that even at part load when a lower discharge is flowing through the runner, a high efficiency can be attained.

12. A draft tube is a pipe of gradually increasing cross-sectional area used for discharging water from the exit of the runner of a reaction turbine to the tail race.

The efficiency of draft tube is given by

$$\eta_d = \frac{\left[\left(V_2^2 - V_3^2\right)/2g\right] - h_f}{V_2^2/2g}$$

where

V_2 = velocity at the inlet of the draft tube;

V_3 = velocity at the outlet of the draft tube; and

h_f = loss of head in the draft tube.

13. A governor is a device used to regulate the discharge through the runner of a turbine so that under varying loads the speed of rotation of the runner remains constant.

Oil pressure governor is commonly used with modern turbines.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

14. The runaway speed of a turbine runner is the maximum speed that may be attained by it if the external load suddenly drops to almost zero value and at the same time the governing mechanism of the turbine fails.

15. A surge tank is provided to store water when the discharge through the runner is reduced by the governing operation due to reduction in the load and supply the same when due to increase in load more discharge is required through the runner.

PROBLEMS

21.1 Show from first principles that the peripheral coefficient of a Pelton wheel is 0.5.

21.2 For the kinetic energy at the exit from buckets of Pelton turbine to be minimum, prove that

$$\rho = \frac{u}{V} = \frac{(k - \cos\phi)k}{(k-1)^2 + 2k(1-\cos\phi)}$$

where u = bucket velocity, V = jet velocity, k = ratio of relative velocity at exit to that at inlet and ϕ stands for the bucket vane angle at exit [which is same as angle $(180° - \phi)$ in Fig. 21.4]. Hence, show that the hydraulic efficiency is given by

$$\eta_h = \frac{2C_v^2 k}{(1+k)} \text{ where } \phi = 180°$$

21.3 Design a Pelton wheel which is required to develop 1 500 kW, when working under a head of 160 m at a speed of 420 r.p.m. The overall efficiency may be taken as 85% and assume other data required.

[*Ans.* D = 1.17 m; No. of jets = 3; d = 93.21 mm; No. of buckets = 21]

21.4 A double jet Pelton wheel is required to generate 7500 kilowatts when the available head at the base of the nozzle is 400 m. The jet is deflected through 165° and the relative velocity of the jet is reduced by 15% in passing over the buckets. Determine (i) the diameter of each jet, (ii) total flow, (iii) force exerted by the jet on buckets in tangential direction. Assume generator efficiency of 95%, overall efficiency of 80%, K_v = 0.97, and K_u = 0.46.

[*Ans.* (i) 136.5 mm; (ii) 2.515 m^3/s; (iii) 103.42 kN]

21.5 The Pykara Power House is equipped with inpulse turbine of Pelton type. Each turbine delivers a maximum power of 14 000 kW when working under a head of 840 m and running at 600 r.p.m. Find the least diameter of the jet and the mean diameter of the wheel. What would be the approximate diameter of orifice at the nozzle tip? Determine the value of the jet ratio and state if it is within the limits. Specify the number of buckets for the wheel. Take the overall efficiency of the turbine as 89.2 per cent. Assume C_v = 0.988 and K_u = 0.45.

[*Ans.* 138.3 mm; 1839 mm; 173 mm (= 1.25 × 138.3); 13.3 hence it is within safe limits; 22]

21.6 A reaction turbine works under a head of 6 m. The guide blades are inclined at 30° to the tangent at periphery and the runner vanes make 110° to the forward tangent at the periphery at inlet. If the discharge is radial and if all the exit velocity is wasted, find the hydraulic efficiency of the turbine. Assume velocity of flow to be constant. Find also the velocity of flow.

[**Hint**. For η_h use the expression derived in Illustrative Example 21.7]

[*Ans.* 87.89%; 3.77 m/s]

21.7 An inward flow reaction turbine works under a head of 22.5 m. The external and internal diameters of the runner are 1.35 m and 1 m respectively. The angle of guide vanes is 15° and the moving vanes are radial at inlet. Radial velocity of flow through runner is constant and there is no velocity of whirl at outlet. Determine the speed of the runner in r.p.m., and the angle of vane at outlet. If the turbine develops 275 kW, find the specific speed. Neglect friction losses.

[*Ans.* 206.5 r.p.m; 19° 53°; 69.88 r.p.m.]

21.8 An inward flow reaction turbine is required to develop 300 kW at 200 r.p.m. The active head at the turbine is 18 m. Determine the outside and inside diameters, the inlet and exit angles for the vanes and the exit angle for the guide vanes. Assume the inlet diameter equal to twice the outlet diameter, the hydraulic efficiency as 80%, the constant radial velocity of flow of 3.6 m/s through the runner, the mechanical efficiency

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

as 95% and the width ratio as 0.10. Water leaves the runner radially.

[*Ans.* 1.406 m; 0.703 m; 144° 56′; 26° 4′; 20° 34′]

21.9 Show that in a turbine with radial vanes at inlet and outlet and operating under head h, if the velocity of flow at outlet is C times that at inlet, the peripheral velocity u and the hydraulic efficiency η_h are given by

$$u = \sqrt{\frac{2gh}{2 + C^2 \tan^2 \alpha}} \; ; \eta_h = \frac{2}{2 + C^2 \tan^2 \alpha}$$

where α is guide blade angle.

21.10 Two inward flow reaction turbines have same runner diameter viz., 0.60 m and the same efficiency, they work under the same head and they have the same velocity of flow viz., 6 m/s. One of the runners A revolves at 520 r.p.m., and has an inlet vane angle of 65°. If the other runner B has an inlet vane angle of 110°, at what speed should it run? [*Ans.* 600 r.p.m.]

21.11 Find the leading dimensions of the runner of an inward flow Francis turbine producing 625 kW, at 1000 r.p.m., under a head of 100 m. Assume the guide vane angle of 16°, the axial length of the blade at inlet 0.1 times the outer diameter, inner radius 0.6 times the outer radius, radial velocity of flow constant, final discharge radial, hydraulic efficiency 87%, mechanical efficiency 97%, allowance for the blade thickness 5%.

[*Ans.* 541.3 mm; 324.8 mm; 54.13 mm; 90.22 mm; 78°26′; 26° 55′]

21.12 A reaction turbine runner has inlet diameter 3.5 m, exit diameter 2.1 m, width 0.5 m both at inlet and exit, and vane angles with the direction of motion 60° and 150°at inlet and exit respectively. If the guide blade angle is 15° and the rate of flow of water is 15 m^3/s, calculate the required speed of the runner for smooth flow at the inlet. For this condition, also compute the torque exerted on the runner, the power developed, the energy extracted per unit weight of water, and the pressure drop through the runner.

[*Ans.* 47 r.p.m.; 309.903 kN.m; 1 525.3 kW; 10.366 N.m/N; 60.15 kN/m^2]

21.13 Explain the purpose of providing (i) scroll casing; (ii) stay vanes; (iii) guide vanes, for a reaction turbine.

21.14 What are the uses of a draft tube? Describe with neat sketches different types of draft tubes.

21.15 (a) Explain briefly the principles on which a Kaplan turbine works.

(b) A Kaplan turbine runner develops 9 300 kW under a net head of 7.4 m. Mechanical efficiency of the wheel is 86%, speed ratio based on outer diameter is 2.2 and the flow ratio is 0.66. Diameter of the boss is 0.35 times the external diameter of the wheel. Determine (i) diameter of the runner; (ii) its synchronous speed; (iii) the specific speed of the runner. Assume mechanical efficiency equal to overall efficiency.

[*Ans.* (i) 5.21 m; (ii) 97.12 r.p.m.; (iii) 767.4 r.p.m.]

21.16 A Kaplan turbine develops 2 100 kW under a net head of 7.2 m with an overall efficiency of 87%. It is to be fitted with an elbow-type draft tube (draft tube efficiency 88%) having its inlet 1.8 m diameter. Determine how much above or below the tail race level should the draft tube inlet be set so that vacuum pressure there does not exceed 460 mm of mercury.

[*Ans.* 1.83 m below the tail race]

21.17 A Kaplan turbine develops 2575 kW under a net head of 7.5 m. It is provided with an elbow type draft tube with a circular inlet of 2.5 m diameter. The inlet is set at a height of 1.5 m above the tail race level. A vacuum gage connected to draft tube inlet records a reading of 36.3 kN/m^2. If the efficiency of the draft tube is 78%, calculate the efficiency of the Kaplan turbine. If the ratio of area of circular inlet and rectangular exit of the draft tube is 1 : 5, determine the power lost due to friction in the tube. If the turbine output were reduced to 1 250 kW, head and speed remaining unchanged, determine the vacuum gage reading.

[*Ans.* 96%; 181.9 kW; 19.78 kN/m^2]

21.18 What is a surge tank and a forebay and what are their functions? Describe with neat sketches different types of surge tanks.

21.19 What do you understand by governing of hydraulic turbines? Explain with sketches the working of an oil pressure governor.

21.20 Write short notes on:

Scroll casing, draft tube, wicket gates, speed

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ratio, flow ratio, turbine efficiencies, relief valve, oil pressure governor, electro-hydraulic governor.

21.21 A Kaplan turbine produces 44 000 kW under a head of 25 m with an overall efficiency of 90%. Taking the valve of speed ratio K_u as 1.6, flow ratio ψ as 0.5 and the hub diameter as 0.35 times the outer diameter, find the diameter and speed of the turbine. [*Ans.* 5.11 m; 132.4 r.p.m.]

21.22 A reaction turbine 0.5 m diameter when running at 600 r.p.m., develops 265 kW, the flow through the turbine being 0.75 m^3/s. Under the above operating condition the pressure head at the entrance to the turbine is 28 m, the elevation of the turbine above the tail water level is 1.3 m and the velocity of flow at entrance to the turbine runner is 3.7 m/s. Assuming the runner vane angle at inlet as 90°, calculate the effective head on the turbine and the efficiency. [*Ans.* 42.58 m; 84.6%]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Performance of Turbines

Chapter 22

22.1 INTRODUCTION

Turbines are often required to work under varying conditions of head, speed, output and gate opening. As such, in order to predict their behaviour, it is essential to study the performance of the turbines under the varying conditions. The variations in the conditions for working may be as follows:

(i) The head and hence the output of the turbine may change, the speed being correspondingly adjusted so that no appreciable change in efficiency occurs, the gate opening remaining constant.

(ii) The output may be varied by the movements of the gates or the spear (or needle); the head and speed remaining constant. These are normal operating conditions for most of the turbines.

(iii) The head and speed may vary. Such variations are common particularly in low head units. It may, however, be stated that although the speed is permitted to vary within very narrow limits, the head may vary by even 50% or more.

(iv) The speed may be allowed to vary by adjusting the load on the turbine; the head and gate opening remaining constant. These conditions can be developed only for laboratory turbines or those in the test plant and are otherwise uncommon.

In order to predict the behaviour of a turbine working under varying conditions and to facilitate comparison between the performances of the turbines of the same type but having different outputs and speeds and working under different heads, it is often convenient to express the test results in terms of certain *unit quantities*. Similarly to facilitate the comparison between the turbines of different types certain *specific quantities* are also required to be considered. Both these quantities are explained in the following sections.

22.2 PERFORMANCE UNDER UNIT HEAD—UNIT QUANTITIES

From the output of a turbine corresponding to different working heads it is possible to compute the output which would be developed if the head was reduced to unity (say 1 metre); the speed being adjusted so that the efficiency remains unaffected. The efficiency of a given turbine will remain unchanged if the velocity triangle under working head H and under unit head are geometrically similar so that shock at entry is avoided.

Figure 22.1 shows two inlet velocity triangles for Francis turbine under its working head H and under unit head, the subscript u being used to denote unit conditions. Both these triangles are similar if their corresponding sides are parallel. It then follows that

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{u}{V_w} = \frac{u_u}{(V_{wu})} \text{ and } \frac{u}{V_f} = \frac{u_u}{(V_{fu})} \quad \text{... (22.1)}$$

Further it may be shown that

$$\frac{(V_w u/g)}{(V_f^2/2g)} = \frac{(V_{wu})(u_u)/g}{(V_{fu}^2/2g)} \quad ; \quad \frac{(V_w u/g)}{(V_{f1}^2/2g)} = \frac{(V_{wu})(u_u)/g}{(V_{f1u}^2/2g)}$$

The above expression indicates that the ratio of the useful energy to wasted velocity energy is same under the working head and the unit head conditions. In other words the required condition of unchanged efficiency is fulfilled. Thus

$$\eta_h = \eta_{hu}; \text{ or } \frac{V_w u}{gH} = \frac{V_{wu} u_u}{g \times 1} \quad \text{... (22.2)}$$

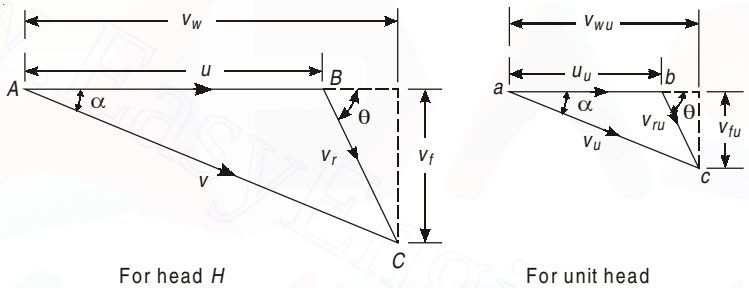


Figure 22.1 *Inlet velocity triangles of head H and unit head*

Eliminating V_w and V_{wu} from Eqs. 22.1 and 22.2 we obtain

$$u_u^2 = \frac{u^2}{H} \quad ; \quad \text{or } u_u = \frac{u}{\sqrt{H}} \quad \text{... (22.3)}$$

and

$$V_{fu}^2 = \frac{V_f^2}{H} \quad ; \quad \text{or } V_{fu} = \frac{V_f}{\sqrt{H}} \quad \text{... (22.4)}$$

Since $u = \frac{\pi DN}{60}$ and $u_u = \frac{\pi DN_u}{60}$ it follows that

$$N_u = \frac{N}{\sqrt{H}} \quad \text{... (22.5)}$$

where N_u represents the speed of the turbines under unit head and it is known as *unit speed.*

Further it can be shown that both the speed ratio K_u and the flow ratio ψ remain constant under the head H and the unit head.

The rate of flow $Q = (k\pi DB)V_f$ and $Q_u = (k\pi DB)V_{fu}$, hence

$$\frac{Q_u}{Q} = \frac{V_{fu}}{V_f} = \frac{1}{\sqrt{H}}; \quad \therefore Q_u = \frac{Q}{\sqrt{H}} \quad \text{... (22.6)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where Q_u represents the discharge flowing through the turbine under a unit head and is known as *unit discharge*.

The power developed

$$P = (wQH)\eta_0$$

and

$$P_u = (wQ_u \times 1)\eta_c, \text{ hence}$$

$$\frac{P_u}{P} = \frac{Q}{Q_u \times H} = \frac{1}{H^{3/2}} \; ; \; \therefore P_u = \frac{P}{H^{3/2}} \qquad \ldots (22.7)$$

where P_u represents the output of the turbine under unit head and is known as *unit power*.

Although the above expressions have been deduced from the velocity triangles of a Francis turbine, the same are applicable to all types of turbines. The advantage of obtaining the performance at unit head is that from the known values of discharge Q, speed N and power P of a turbine working under head H, the value Q_1, N_1 and P_1 corresponding to some other head H_1, may be computed as indicated below:

Since Q_u, N_u and P_u will be same for both the heads, we have

$$Q_u = \left(Q/\sqrt{H}\right) = \left(Q_1/\sqrt{H_1}\right); \qquad \therefore Q_1 = \left(Q\sqrt{H_1}\right)/\sqrt{H}$$

$$N_u = \left(N/\sqrt{H}\right) = \left(N_1/\sqrt{H_1}\right); \qquad \therefore N_1 = \left(N\sqrt{H_1}\right)/\sqrt{H}$$

$$P_u = (P/H^{3/2}) = (P_1/H_1{}^{3/2}); \qquad \therefore P_1 = (PH_1{}^{3/2})/H^{3/2}$$

The above derived expressions are based on the assumption that the efficiencies remain constant at all the heads, which is however not correct. In practice it has been found that the efficiency varies with the head which will cause the scale effect.

The expressions for the various quantities as derived above can be used for comparing the performance of any one turbine only under different conditions of operation. However, according to another definition for the unit quantities as indicated below more general expressions for the unit quantities may be obtained which would facilitate a comparison between the performance of different turbines of the same type. According to this definition the unit speed, unit discharge and unit power are defined respectively as the speed, discharge and power of a turbine having a runner diameter of 1 m and operating under a head of 1 m (or unit head).

Thus from Eq. 22.3

$$\frac{\pi \times 1 \times N_u}{60} = \frac{\pi ND}{60} \times \frac{1}{\sqrt{H}} \; ; \text{or } N_u = \frac{ND}{\sqrt{H}} \qquad \ldots (22.5\text{ a})$$

Similarly from Eqs. 22.6 and 22.7 the following expressions for Q_u and P_u may be obtained

$$Q_u = \frac{Q}{D^2\sqrt{H}} \qquad \ldots (22.6\text{ a})$$

and

$$P_u = \frac{P}{D^2 H^{3/2}} \qquad \ldots (22.7\text{ a})$$

By using these expressions the values of the unit quantities may be determined for any one turbine and the same applies to all the similar turbines of the same type. It may however be stated that the values of the unit quantities for similar turbines will be equal only if their efficiencies are equal. Since the efficiencies of even the similar turbines are not equal the values of the unit quantities will also differ.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

22.3 PERFORMANCE UNDER SPECIFIC CONDITIONS

The performance of turbines under unit head facilitate the comparison of turbines of the same type. However, the turbines of different types may be compared by considering an imaginary turbine called *specific turbine*. The specific turbine is an imaginary turbine which is identical in shape, geometrical proportions, blade angles, gate setting etc., with the actual turbine but reduced to such a size that it will develop one kilowatt power (or one metric horse power) under unit head. The suffix *s* is used to denote the symbols expressing the performance of the specific turbine.

Figure 22.2 shows the actual and the specific runners for two types of turbines. Since the actual turbine under unit conditions and the specific turbine both work under a head of 1 metre it is evident that the velocity triangle *abc* in Fig. 22.1 will hold good for the specific turbine as well. In other words $u_s = u_u$, $V_{fs} = V_{fu}$ etc. Now $Q_u = \pi D\,(nD) V_{fu}$ and $Q_s = \pi D_s\,(nD_s) V_{fs}$

$$\therefore \quad \frac{Q_u}{Q_s} = \left(\frac{D}{D_s}\right)^2 \qquad \text{... (22.8)}$$

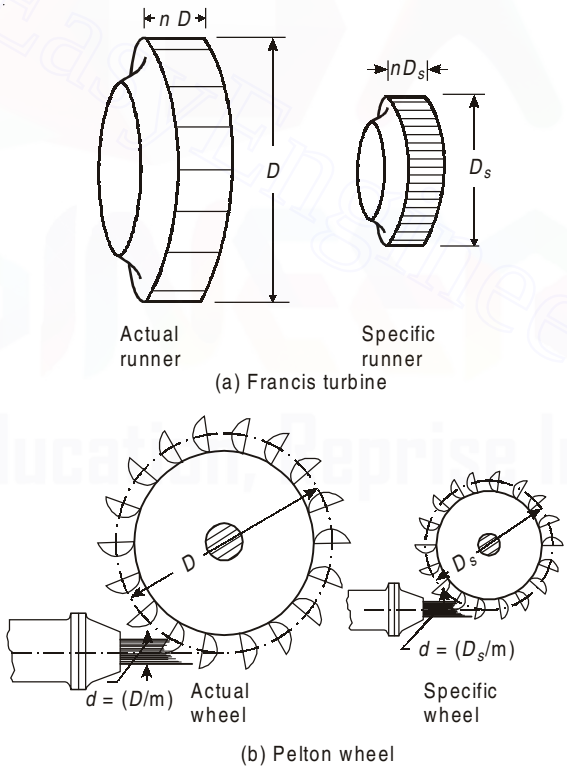


Figure 22.2 *Actual and specific runners*

Again $P_u = wQ_u \times 1 \times \eta_0$; and $P_s = wQ_s \times 1 \times \eta_0$
Assuming η_o to be same for both

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{P_u}{P_s} = \frac{Q_u}{Q_s} = \left(\frac{D}{D_s}\right)^2 \quad \text{... (22.9)}$$

Equation 22.9 shows that the power of geometrically similar turbines working under the same head vary as the square of the runner diameters.

Since P_s = 1 kW (or 1 metric h.p.) (by definition) and $P_u = \frac{P}{H^{3/2}}$

$$D_s = \frac{D}{\sqrt{P_u}} = \frac{DH^{3/4}}{\sqrt{P}}$$

Further $u_u = \frac{\pi D N_u}{60}$

and $u_s = \frac{\pi D_s N_s}{60}$

but $u_u = u_s$

hence $\frac{\pi D N_u}{60} = \frac{\pi D_s N_s}{60}$

$$\therefore \quad N_s = N_u \frac{D}{D_s} = N_u \sqrt{P_u} \quad \text{... (22.10)}$$

but $N_u = \frac{N}{\sqrt{H}}$

and $P_u = \frac{P}{H^{3/2}}$

$$\therefore \quad N_s = \frac{N\sqrt{P}}{H^{5/4}} \quad \text{... (22.11)}$$

This value of N_s, the speed of specific runner is known as *specific speed*. For any other turbine also exactly same relationship for N_s may be derived.

A significant point about the above noted expression for the specific speed is that it is independent of the dimensions or size, both of the actual turbine and of the specific turbine. It therefore means that all turbines of the same geometrical shape, working under the same value of K_u and ψ, and thus having the same efficiency, will have the same specific speed, no matter what their sizes be and what powers they develop under what heads. As such it may be stated that N_s represents the specific speed of the actual turbine, as well as of the specific turbine. Therefore, the following general definition for the specific speed may be given.

The specific speed of any turbine is the speed in r.p.m. of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 kilowatt power when working under unit head (i.e., 1 metre).

In metric units the power is expressed in metric horse power and hence in metric units the following definition for the specific speed may be given.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The specific speed of any turbine is the speed in r.p.m. of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 metric horse power when working under unit head (i.e., 1 metre).

The specific speed is usually computed for the operating conditions corresponding to the maximum efficiency.

It may be seen from Eq. 22.11, which is dimensionally non-homogeneous, that the specific speed is not a pure number. By considering the dimensions of N, P and H it can be shown that the value of N_s in SI units is equal to about 0.858 times its value in metric units and the value of N_s in metric units is equal to about 4.44 times its value in F.P.S. units.

There exists a general trend to select a turbine of higher specific speed because higher specific speed of a turbine results in the reduction of the runner diameter as well as the overall size of the runner, due to which the weight and the cost of the runner are reduced. This may be explained from Eq. 22.11 which shows that for a given power output and head, an increase in the specific speed permits a higher speed of rotation for the turbine runner, corresponding to which even for a runner of smaller diameter a higher peripheral velocity may be developed. As such by increasing the specific speed the runner diameter as well as its overall size may be reduced. Further as shown in Illustrative Example 22.7, the specific speed of a turbine may be expressed in terms of unit speed N_u, unit discharge Q_u and overall efficiency η_o as

$$N_s = 3.13\, N_u \sqrt{Q_u \eta_o}$$

From the above expression it is seen that the specific speed may be increased by increasing the unit speed N_u or unit discharge Q_u. It is, however, more convenient to increase the specific speed by increasing unit discharge Q_u since by this as seen from Eq. 22.6 (a) for a given discharge Q and head H, the runner diameter is reduced proportionally to the square root of the unit discharge Q_u and the rotational speed N is increased proportionally to the specific speed. The unit discharge may be increased if the head H acting on the turbine is reduced and the discharge Q flowing through the runner is increased. As indicated in Chapter 21 a turbine operating under a low head and consequently required to allow larger discharge to flow through the runner must have a larger area of flow passage which may be obtained by designing the runner with axial flow right from its inlet end. Thus, axial flow turbines are high specific speed turbines. Further corresponding to high specific speed of an axial flow turbine the speed of rotation of its runner is also high, and hence for the same operating conditions, viz., output power (or discharge) and head, the diameter and the overall size of a runner of an axial flow turbine is much smaller than that for the runner of other types of turbines having lower specific speeds. This fact has been illustrated in Fig. 22.3 which shows relative sizes of different types of runners designed to develop same power and operating

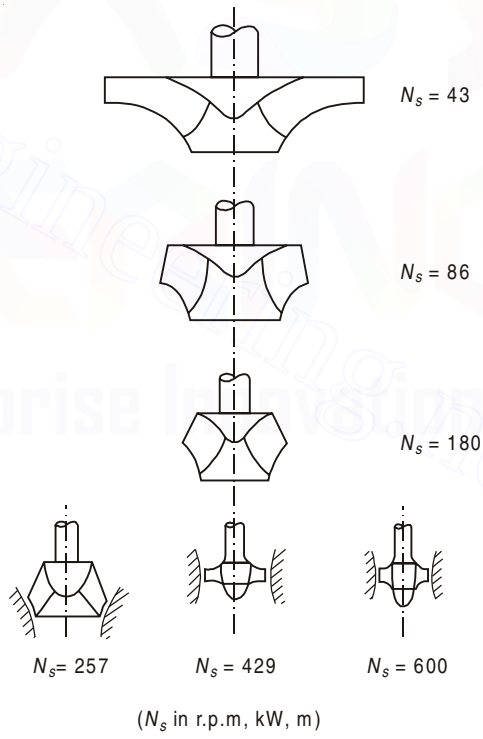


Figure 22.3 *Relative sizes of turbine runners developing same power under same head for different specific speeds*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

under the same head, for different specific speeds. It may be seen from Fig. 22.3 that for low specific speed radial flow runner a large diameter and relatively narrow fluid passage is required to be provided, and for increased specific speeds the runners of smaller diameters and wider flow passages are required to be provided.

It may thus be stated that an increase in the specific speed of a turbine results in the reduction of the diameter as well as the overall size of the turbine runner. Since this reduction in size of a turbine runner will lead to a corresponding reduction in the cost, it is evident that for economic reasons a turbine runner with highest specific speed possible should be selected.

22.4 EXPRESSIONS FOR SPECIFIC SPEEDS IN TERMS OF KNOWN COEFFICIENTS FOR DIFFERENT TURBINES

From the general expression for the specific speed represented by Eq. 22.11, it is possible to obtain the expressions for specific speed in terms of known coefficients for different turbines as discussed below.

1. Specific Speed of Pelton Wheel. An expression for the specific speed of Pelton wheel may be obtained in terms of K_v (or C_v), K_u, η_o and m as below:

$$u = K_u\sqrt{2gH} = \frac{\pi DN}{60}$$

$$\therefore \quad N = \left[\frac{60K_u\sqrt{2gH}}{\pi D}\right] = \left[\frac{84.6K_u\sqrt{H}}{D}\right]$$

$$P = wQH \times \eta_o$$

$$= \frac{9810}{1000}\left(\frac{\pi d^2}{4}\right) \times K_v\sqrt{2gH} \times H \times \eta_o$$

$$= 34.13\,(K_v d^2\,\eta_o\,H^{3/2})\ \text{kW}$$

Since $w = 9810\ \text{N/m}^3$ and $g = 9.81\ \text{m/s}^2$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$\therefore \quad N_s = \frac{(84.6)K_u(\sqrt{H})(34.13)^{1/2}(K_v d^2\eta_o H^{3/2})^{1/2}}{D(H^{5/4})}$$

or $$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{(D/d)}$$

or $$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{m} \qquad ...(22.12)$$

Taking $K_u = 0.46$, $K_v = 0.98$ and $\eta_o = 85\%$, which are the usual values, Eq. 22.12 gives

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$N_s = \frac{207.5}{m} \qquad \ldots(22.12a)$$

In metric units since power P is expressed in metric horse power, we have

$$P = \frac{wQH}{75} \times \eta_o$$

$$= \frac{w(\pi/4)d^2 \times K_v\sqrt{2gH} \times H \times \eta_o}{75}$$

$$= 46.39\ (K_v\, d^2\, H^{3/2}\eta_o)\ \text{h.p.}$$

Since $w = 1000$ kg (f)/m^3 and $g = 9.81$ m/s^2

Thus $$N_s = \frac{(84.6)K_u(\sqrt{H})(46.39)^{1/2}(K_v d^2 \eta_o H^{3/2})^{1/2}}{D(H^{5/4})}$$

or $$N_s = \frac{(576.2)K_u(K_v\eta_o)^{1/2}}{m} \qquad \ldots(22.13)$$

For the above noted values of K_u, K_v and η_o Eq. 22.13 gives

$$N_s = \frac{242}{m} \qquad \ldots(22.13\ a)$$

Equation 22.12 (a) or 22.13(a) gives a relationship between the specific speed N_s and jet ratio m for a single jet Pelton wheel. Since the value of m is more or less constant and in most of the cases it is equal to 12, almost all the Pelton wheels have the same specific speed. However, in some exceptional cases an abnormally low value of m equal to 7 has also been used. As such from Eq. 22.12 (a) it is found that for m varying from 7 to 12 the value of N_s in SI units varies from about 30 to 17. Similarly from Eq. 22.13 (a) for m varying from 7 to 12 the value of N_s in metric units varies from about 35 to 20. It is thus observed that for Pelton wheel the specific speed has quite a narrow range of variation.

For a multiple jet Pelton wheel having n number of jets, the expression for specific speed may be obtained in the same manner. In this case the discharge will be (nQ) and hence the power will be (nP). Therefore in this case the specific speed will be $\sqrt{n}$ times the specific speed for a single jet Pelton Wheel.

2. Specific Speed of Francis Turbine. For Francis turbines an expression for the specific speed may be obtained in terms of K_u, ψ and n as indicated below:

$$u = K_u\sqrt{2gH} = \frac{\pi DN}{60}$$

$$\therefore \quad N = \frac{84.6K_u\sqrt{H}}{D}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$P = wQH \times \eta_o$$

$$= [w\,(k\pi n\,D^2)\,\psi\sqrt{2gH}\ \times (H \times \eta_o)]$$

$$= 136.51\,(k\,nD^2\,\psi\,H^{3/2}\,\eta_o)\ \text{kW}$$

Since $w = 9810\ \text{N/m}^3$ and $g = 9.81\ \text{m/s}^2$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$\therefore \quad N_s = \frac{(84.6)K_u\sqrt{H}\times(136.51)^{1/2}(knD^2\psi H^{3/2}\eta_o)^{1/2}}{DH^{5/4}}$$

or $$N_s = 988.45\left[K_u(\sqrt{k\ n\ \psi\ \eta_o})\right] \qquad ...(22.14)$$

In metric units since power is expressed in metric horse power, we have

$$P = \frac{wQH}{75} \times \eta_o$$

$$= \frac{[w(k\ \pi\ n\ D^2)\psi\sqrt{2gH}\times(H\times\eta_o)]}{75}$$

$$= 185.54\,(k\,n\,D^2\,\psi\,H^{3/2}\,\eta_0)$$

since $w = 1000\ \text{kg(f)/m}^3$ and $g = 9.81\ \text{m/s}^2$

Thus $$N_s = \frac{(84.6)K_u\sqrt{H}(185.54)^{1/2}(knD^2\psi H^{3/2}\eta_o)^{1/2}}{D(H^{5/4})}$$

or $$N_s = 1152.4\left[K_u(\sqrt{k\ n\ \psi\ \eta_o})\right] \qquad ...(22.14\ a)$$

Assuming k and η_o to be constant, Eq. 22.14 indicate that specific speed N_s for Francis turbines depends on speed ratio K_u, flow ratio ψ and breadth ratio n. Generally K_u ranges from 0.6 to 0.9, ψ ranges from 0.15 to 0.30 and n ranges from 0.10 to 0.45. The variation of an ψ or all of these will alter N_s, and hence a much greater range of N_s is available for Francis turbines as compared to that for Pelton wheel.

Francis turbine runners are classified as *slow runners* (N_s = 51 to 103), *normal* or *medium runners* (N_s = 103 to 154) and *fast runners* (N_s = 154 to 257). The corresponding ranges of N_s in metric units are for *slow runners* (N_s = 60 to120), for *normal* or *medium runners* (N_s = 120 to 180) and for *fast runners* (N_s = 180 to 300). It has been observed that the increase in N_s from 51 to 154 (or 60 to 180) is obtained by increasing n and ψ, but to attain the higher value, it would be necessary to increase simultaneously K_u, ψ and n. It may, however, be pointed out that in order to increase N_s the values of K_u and ψ cannot be increased to any value, because unduly higher values will result in increased hydraulic losses and consequently lower efficiency. It has been explained in Chapter 21 that as the value of n increases the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

runner shapes are also altered. For slow runners the flow is predominantly radial and exit is axial. For fast runners the flow is more or less axial both at entry and exit.

3. Specific Speed of Kaplan and Propeller Turbines. Since these axial flow turbines work under low heads, it is evident from Eq. 22.11 for N_s that these turbines have very high specific speed ranging from 257 to 858 (or 300 to 1000 in metric units). As for the Francis turbine in this case also it can be shown that

$$N_s = C_1\left(\sqrt{C_2}\right)\left[K_u\sqrt{\psi(1-n^2)\eta_o}\right] \quad \ldots(22.15)$$

where $C_1 = \dfrac{60\sqrt{2g}}{\pi} = 84.6$; and

$$C_2 = \frac{w\pi\sqrt{2g}}{4} = 34.13$$

However, in metric units the value of C_2 is obtained as

$$C_2 = \frac{w\pi\sqrt{2g}}{4\times 75} = 46.39$$

Now if $\eta_o = 90\%$; $n = (d/D) = 0.35$ then

in SI units

$$N_s = 439.22\left(K_u\sqrt{\psi}\right) \quad \ldots(22.15\text{ a})$$

and in metric units

$$N_s = 512.07\left(K_u\sqrt{\psi}\right) \quad \ldots(22.15\text{ b})$$

22.5 PERFORMANCE CHARACTERISTIC CURVES

The turbines are generally designed to work at particular values of H, Q, P, N and η_o which are known as the designed conditions. But often the turbines are required to work at conditions different from those for which they have been designed. Therefore, it is essential to determine the exact behaviour of the turbines under the varying conditions by carrying out tests either on the actual turbines or on their small scale models. The results of these tests are usually graphically represented and the resulting curves are known as *characteristic curves*. For the sake of convenience the characteristic curves are plotted in terms of unit quantities. These characteristic curves are of the following three types:

(i) Constant head characteristic curves (or Main characteristic curves).

(ii) Constant speed characteristic curves (or Operating characteristic curves).

(iii) Constant efficiency curves

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

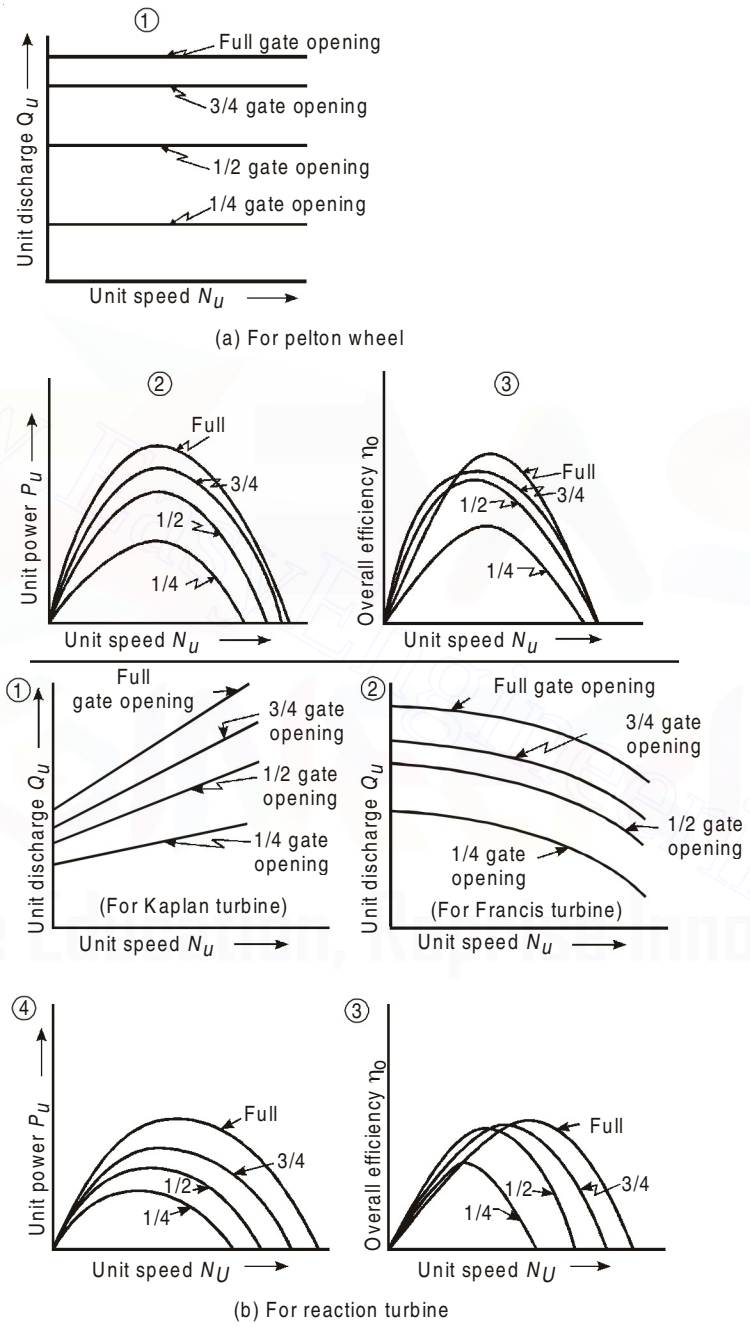


Figure 22.4 *Constant head characteristics for Pelton wheel and reaction turbines*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

1. Constant Head Characteristic Curves. In order to obtain these curves the tests are performed on the turbine by maintaining a constant head and a constant gate opening and the speed is varied by changing the load on the turbine. A series of values of N are thus obtained and corresponding to each value of N, discharge Q and the output power P are measured. A series of such tests are performed by varying the gate opening, the head being maintained constant at the previous value. From the data of the tests the values of Q_u, P_u, N_u and η_o are computed for each gate opening. Then with N_u as abscissa the values of Q_u, P_u and η_o for each gate opening are plotted. The curves thus obtained for Pelton wheel and the reaction turbines for four different gate openings are shown in Fig. 22.4.

For Pelton wheels since Q_u depends only on the gate opening and is independent of N_u, the $Q_u v/s\, N_u$ plots are horizontal straight lines. However, for low specific speed Francis turbines $Q_u v/s\, N_u$ are drooping curves, thereby indicating that as the speed increases the discharge through the turbine decreases. This is so because in these turbines a centrifugal head is developed which retards the flow. Since the centrifugal head increases with the speed, the flow through the turbine is reduced as the speed increases. On the other hand, for high specific speed Francis turbines as well as Kaplan turbines, since the flow is axial there is no such centrifugal head developed which may cause the retardation of the flow.

The curves of $P_u v/s\, N_u$ and $\eta_o v/s\, N_u$ are parabolic in shape for the different turbines as shown in Fig. 22.4. It will be observed that for a Pelton wheel, for each gate opening the maximum value of η_o is attained at almost the same value of N_u which corresponds to $K_u = 0.46$ [$K_u = (N_u \pi D)/60\sqrt{2g}$]. However, in case of reaction turbines for each gate opening the maximum value of η_o is attained at different values of N.

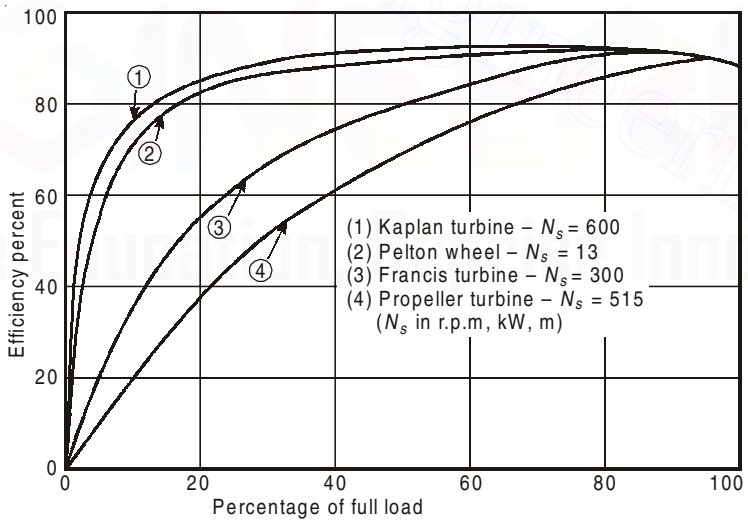


Figure 22.5 *Percentage of full load v/s η_o curves for different types of turbines*

2. Constant Speed Characteristic Curves. In order to obtain these curves the tests are performed on the turbines by operating them at constant speed. The constant speed is attained by regulating the gate opening thereby varying the discharge flowing through the turbine as the load varies. The head may

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or may not remain constant. The power developed corresponding to each setting of the gate opening is measured and the corresponding values of η_o are computed. Further knowing the total load capacity of the turbine the percentage of full load may be computed from the measured power, and a plot of η_o v/s percentage of full load is prepared. The curves thus obtained for the four different types of turbines working at constant speed under constant head at varying gate openings are shown in Fig. 22.5, from which it may be seen that as the % full load increases η_o also increases. In other words at reduced loads η_o is also less. At 100% full load η_o is near about the maximum efficiency in all cases. Further it may be observed that the Kaplan turbine and the Pelton wheel maintain a high efficiency over a longer range of the part load as compared with either the Francis or the fixed blade propeller turbine. Therefore this is an important factor in the selection of turbines.

In addition to the above graph, output power v/s discharge and $\eta_o v/s$ discharge graphs are also plotted which are shown in Fig. 22.6. Both these graphs are plotted from a certain minimum discharge Q_o which is required to initiate the motion of the turbine runner from its state of rest. Further since output power is directly proportional to discharge if head is constant, output power v/s discharge graph is a straight line. On the other hand $\eta_o v/s$ discharge graph is curvilinear and as shown in Fig. 22.6 the overall efficiency η_o increases with discharge and remains more or less constant beyond a particular value of discharge.

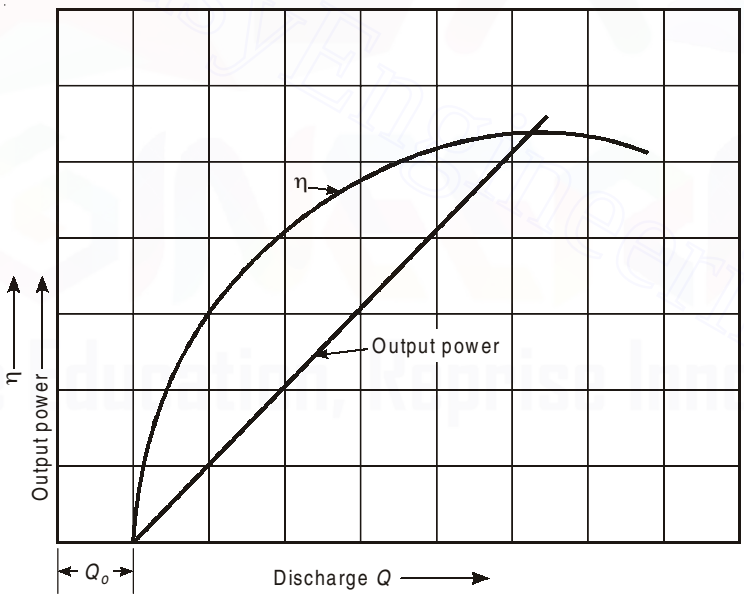


Figure 22.6 η_o *and output power* v/s *discharge curves*

3. Constant Efficiency (or Iso-efficiency) Curves. Figure 22.7 shows the constant efficiency or iso-efficiency curves. These curves show the efficiencies of the turbine for all conditions of running and hence these are also known as universal characteristic curves of the turbine. In order to draw these curves the following procedure is adopted. By operating the turbine at about 8 to 10 gate openings, the corresponding number of η_o v/s N_u and N_u v/s Q_u (or P_u) curves are plotted. On the η_o v/s N_u curves

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

set of horizontal lines (each line representing the same efficiency) are drawn which will cut the curves corresponding to each gate opening at different points. These points are projected on the corresponding N_u *v/s* Q_u (or P_u) curves for each gate opening and the points of the same efficiency are joined by smooth curves which are the iso-efficiency curves. It is, however, clear from the graph that the innermost iso-efficiency curve represents the highest efficiency of the turbine and the outer curves represent the lower efficiencies. Further it will be seen from Fig. 22.7 that for a given unit discharge (or unit power) if a vertical line is drawn it will cut some iso-efficiency curve at two points and it will also touch some other inner iso-efficiency curve of higher efficiency, which may not have been drawn, just at one point only. Thus for a given unit discharge (or unit power) the vertical line touches the curve of maximum efficiency at only one point. It, therefore, means that if the peak points of the various iso-efficiency curves are joined together by a smooth curve then we obtain the best performance curve for the turbine, because the peak points of each iso-efficiency curve are the points of maximum efficiency for some value of the unit discharge (or unit power). With the help of these characteristic curves it is possible to predict the best performance of the turbine. Thus if the gate opening is known then the point of intersection of the N_u *v/s* Q_u (or P_u) curve for this gate opening and the best performance curve will represent the conditions for the best performance for that particular gate opening. From this point unit discharge (or unit power) can be known by drawing a vertical line and unit speed by drawing a horizontal line. Now if the head on the turbine is known the discharge Q, the power output P and the speed N for maximum efficiency at this gate opening can be computed. On the other hand if the head H and the speed N are known, then unit speed N_u can be computed. Then by drawing a horizontal line for this value of N_u, to cut the best performance line, the point of maximum efficiency is known corresponding to which unit discharge or unit power can be obtained, and as before the discharge Q and power P can be calculated at which the turbine efficiency will be maximum for the given head H and the speed N.

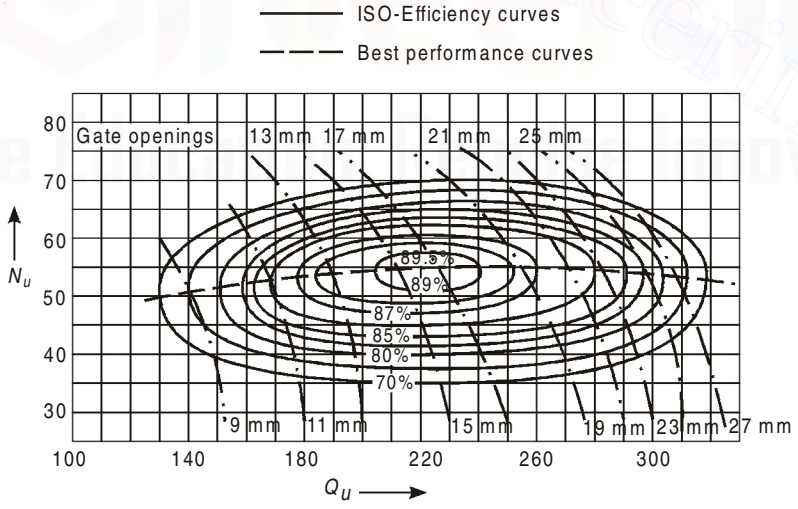


Figure 22.7 *Constant efficiency curve or Universal characteristics of a Francis turbine*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

22.6 MODEL TESTING OF TURBINES

In order to have an idea about the performance of the actual turbine in advance, a small scale model of the turbine which is geometrically similar to the actual turbine is first prepared. The various linear dimensions of the model turbine bear the same proportion to their corresponding dimensions of the actual turbine. The model turbine is then tested under a known head, speed and flow rate and its output as well as the efficiency are determined. From these test results it will be possible to predict the performance of the actual turbine. Moreover on the basis of the test results if the design is to be modified it may be so done in the model turbine without incurring much expenditure. Thus model testing of turbines assists in obtaining a perfect design for the actual turbines, as well as in the development of the new types of turbines with higher specific speed and better efficiency.

The various variables involved in this case are discharge Q, head H, speed of rotation of runner N, runner diameter D, output power P, mass destiny ρ and viscosity μ of the flowing fluid. It may however be stated that in the problems of turbomachines generally shaft work gH is used as one of the variables instead of head H. By adopting the usual methods of dimensional analysis these variables may be grouped into following dimensionless parameters:

$$\left(\frac{Q}{ND^3}\right), \left(\frac{gH}{N^2D^2}\right), \left(\frac{P}{\rho gHND^3}\right), \left(\frac{\mu}{\rho ND^2}\right)$$

The parameter $\left(\frac{Q}{ND^3}\right)$ is known as *discharge number* or *flow number*, $\left(\frac{gH}{N^2D^2}\right)$ is known as *head number*, and $\left(\frac{P}{\rho gHND^3}\right)$ is known as *power number*. By combining these parameters, their alternative expressions may be obtained. Thus the discharge number may also be expressed as $\left(\frac{Q}{D^2\sqrt{gH}}\right)$ and the power number as $\left(\frac{P}{\rho g^{3/2}H^{3/2}D^2}\right)$. Further by dividing the square root of the power number by the head number raised to power (3/4) the following dimensionless parameter n_s is obtained

$$n_s = \frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}$$

The parameter n_s is known as *dimensionless specific speed* or *shape number* of the turbine.

It may, however, be noted that in the above expressions the output power P has been considered in terms of (N.m/s) [or kg(f)m/s in metric units]. But if the output power P is considered in terms of kW then in the above expressions P will have to be replaced by $(P \times 10^3)$. Further for water since $\rho = 1000$ kg/m^3, the dimensionless specific speed n_s becomes

$$n_s = \frac{N\sqrt{P\times 10^3}}{(1000)^{1/2}(9.81)^{5/4}H^{5/4}} = \frac{N_s}{17.362}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $N_s = 17.362\, n_s$

In metric units if the output power is considered in terms of horse power (metric) then in the above expressions P will have to be replaced by $(P \times 75)$. Further for water since $\rho = 102$ msl/m^3, the dimensionless specific speed n_s becomes

$$n_s = \frac{N\sqrt{P \times 75}}{(102)^{1/2}(9.81)^{5/4}H^{5/4}} = \frac{N_s}{20.247}$$

or $N_s = 20.247 n_s$

These expressions give relationship between the specific speed (Eq. 22.11) and the dimensionless specific speed.

For complete similarity to exist between the model and the actual (or prototype) turbines the above noted parameters must have the same values for the model and the prototype turbines. The parameter $\left(\frac{\mu}{\rho N D^2}\right)$ represents Reynolds number which should also be equal in the model and the prototype turbines for complete similitude. But in actual practice, it is not possible to have the same value of the Reynolds number in the model and the prototype turbines, on account of considerable difference in their sizes. However, since the flow in the prototype turbine is turbulent, it has been observed that if the flow in the model turbine is also turbulent, then even if the Reynolds numbers are not equal for the model and the prototype turbines the similarity between them can be ensured.

Thus for the complete similarity to exist between the model and the prototype turbines, the following conditions may be required to be satisfied:

or
$$\left.\begin{aligned}\left(\frac{Q}{ND^3}\right)_m &= \left(\frac{Q}{ND^3}\right)_p \\ \left(\frac{Q}{D^2\sqrt{gH}}\right)_m &= \left(\frac{Q}{D^2\sqrt{gH}}\right)_p\end{aligned}\right\} \quad \text{... (22.16)}$$

$$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p \quad \text{... (22.17)}$$

or
$$\left.\begin{aligned}\left(\frac{P}{\rho gHND^3}\right)_m &= \left(\frac{P}{\rho gHND^3}\right)_p \\ \left(\frac{P}{\rho g^{3/2}H^{3/2}D^2}\right)_m &= \left(\frac{P}{\rho g^{3/2}H^{3/2}D^2}\right)_p\end{aligned}\right\} \quad \text{... (22.18)}$$

$$\left(\frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}\right)_p \quad \text{... (22.19)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since in most of the cases $g_m = g_p$ and also $\rho_m = \rho_p$ the above conditions may be simplified as follows:

$$\left.\begin{aligned}\left(\frac{Q}{ND^3}\right)_m &= \left(\frac{Q}{ND^3}\right)_p \\ \text{or} \quad \left(\frac{Q}{D^2\sqrt{H}}\right)_m &= \left(\frac{Q}{D^2\sqrt{H}}\right)_p\end{aligned}\right\} \quad \text{... (22.16 a)}$$

$$\left(\frac{H}{N^2D^2}\right)_m = \left(\frac{H}{N^2D^2}\right)_p \quad \text{... (22.17 a)}$$

$$\left.\begin{aligned}\left(\frac{P}{HND^3}\right)_m &= \left(\frac{P}{HND^3}\right)_p \\ \text{or} \quad \left(\frac{P}{H^{3/2}D^2}\right)_m &= \left(\frac{P}{H^{3/2}D^2}\right)_p\end{aligned}\right\} \quad \text{... (22.18 a)}$$

$$\left(\frac{N\sqrt{P}}{H^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{H^{5/4}}\right)_p \quad \text{... (22.19 a)}$$

where the subscripts m and p refer to model and prototype turbines respectively. By determining the values of P, H, N and D from the model test the values of the above noted parameters are determined, which will be same for both the model and the prototype turbines. For the prototype turbine since its diameter D and head H are known, its speed N, power P and discharge Q can thus be calculated with the help of the known values of these parameters.

A little consideration will show that the above noted conditions may be achieved if, K_u, ψ and N_s have the same values for the model and the prototype turbines. Further it may be seen that the various terms in the above expressions represent the unit quantities and hence for similarity between the model and the prototype turbines the values of the various unit quantities must be equal for the both.

The above noted conditions for establishing the similarity between the model and the prototype turbines are based on the assumption that the efficiency of the model is equal to that of the prototype. However, the efficiencies of the model and the prototype turbines are not equal. This is so because the energy losses are proportionately more in the model turbine than that in the prototype turbine. As such the efficiency of the model turbine is lower than that of the prototype turbine. On account of difference in the efficiencies of the model and the prototype turbines the scale effect would be developed and it would therefore be possible source of error in predicting the performance of the prototype turbine on the basis of the model test results. However, by modifying the above noted conditions for the similarity the error due to such a scale effect may be eliminated. The modified expressions for the similarity conditions may be obtained as indicated below.

If η_{om} and η_{op} are the overall efficiencies of the model and the prototype turbines respectively, then

$$\eta_{om} = \left(\frac{H - h_L}{H}\right)m \text{ ; and } \eta_{op} = \left(\frac{H - h_L}{H}\right)_p$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where H and h_L are respectively the head acting on the turbine and the loss of head in the turbine and the subscript m and p refer to model and prototype respectively. From the above expressions the net effective heads available for the model and the prototype turbines may be obtained as

$$(H - h_L)_m = \eta_{om} (H)_m$$

and $$(H - h_L)_m = \eta_{op} (H)_p$$

By considering these values of the head in place of H in Eq. 22.16 (a) to 22.19 (a) the following expressions are obtained:

$$\left(\frac{Q}{D^2\sqrt{\eta_o H}}\right)_m = \left(\frac{Q}{D^2\sqrt{\eta_o H}}\right)_p \quad \text{... (22.16 b)}$$

$$\left(\frac{\eta_o H}{N^2 D^2}\right)_m = \left(\frac{\eta_o H}{N^2 D^2}\right)_p \quad \text{... (22.17 b)}$$

$$\left(\frac{P}{\eta_o^{3/2} H^{3/2} D^2}\right)_m = \left(\frac{P}{\eta_o^{3/2} H^{3/2} D^2}\right)_p \quad \text{... (22.18 b)}$$

$$\left(\frac{N\sqrt{P}}{\eta_o^{5/4} H^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{\eta_o^{5/4} H^{5/4}}\right)_p \quad \text{... (22.19 b)}$$

These equations thus represent the conditions for similarity between the model and the prototype turbines with due allowance for the difference in their efficiencies.

From Eq. 22.19 (b) the ratio of the specific speeds for the model and prototype turbines may be expressed as

$$\frac{(N_s)_m}{(N_s)_p} = \left(\frac{\eta_{om}}{\eta_{op}}\right)^{5/4}$$

For determining the efficiency of a prototype turbine from the efficiency obtained for its model, the following general expression has been given:

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{\alpha} \left(\frac{H_m}{H_p}\right)^{\beta} \quad \text{... (22.20)}$$

where η_{om} and η_{op} are the overall efficiencies of the model and the prototype respectively, D_m and D_p are the respective diameters of their runners, H_m and H_p are the heads acting on the model and the prototype turbines respectively and α and β are the numerical exponents. Different values have been suggested for the exponents α and β by different investigators, but the most commonly adopted values are those recommended by L.F. Moody which are $\alpha = 0.20$ and $\beta = 0$. Thus Eq. 22.20 becomes

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.20} \quad \text{...(22.20 a)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Moody's Eq. 22.20 (a) is however applicable for head $H < 150$ m. For head $H > 150$ m the following relationship has been recommended:

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.25} \left(\frac{H_m}{H_p}\right)^{0.10} \quad \text{... (22.20 } b\text{)}$$

It may however be stated that different investigators have recommended different values for the exponent of (H_m/H_p) varying from 0.04 to 0.10 in Eq. 22.20 (b).

22.7 CAVITATION IN TURBINES

When the pressure in any part of the flow passage reaches the vapour pressure of the flowing liquid, it starts vaporizing and small bubbles of vapour form in large numbers. These bubbles (or vapour-filled pockets or cavities) are carried along by the flow, and on reaching the high pressure zones these bubbles suddenly collapse as the vapour condenses to liquid again. Due to sudden collapsing of the bubbles or cavities the surrounding liquid rushes in to fill them. The liquid moving from all directions collides at the centre of the cavity, thus giving rise to very high local pressure, which may be as high as 686.7 MN/m^2 [7000 kg(f)/cm^2]. Any solid surface in the vicinity is also subjected to these intense pressures. The alternate formation and collapse of vapour bubbles may cause severe damage to the surface which ultimately fails by fatigue and the surface becomes badly scored and pitted. This phenomenon is known as *cavitation* which is found to occur in turbines as well as in various hydraulic structures such as penstocks, gates, valves, spillways etc.

In reaction turbines the cavitation may occur at the runner exit or the inlet to the draft tube where the pressure is considerably reduced. Due to cavitation the metal of the runner vanes and the draft tube is gradually eaten away in these zones, which results in lowering the efficiency of the turbine. As such the turbine components should be so designed that as far as possible cavitation is eliminated. In order to determine whether cavitation will occur in any portion of the turbine, D. Thoma of Germany has developed a dimensionless parameter called *Thoma's cavitation factor* σ which is expressed as

$$\sigma = \frac{H_a - H_v - H_s}{H} \quad \text{... (22.21)}$$

where H_a is atmospheric pressure head; H_v is vapour pressure head; H_s is suction pressure head (or height of runner outlet above tail race); and H is working head of turbine. Complete similarity in respect of cavitation can be ensured if the value of σ is same in both the model and the prototype. Moreover it has been found that σ depends on N_s of the turbine and for a turbine of particular N_s the factor σ can be reduced upto a certain value upto which its efficiency η_0 remains constant. A further decrease in the value of σ results in a sharp fall in η_0.. The value of σ at this turning point is called *critical cavitation factor* σ_c. The value for σ_c for different turbines may be determined with the help of the following empirical relationships:

For Francis turbines

$$\sigma_c = 0.625 \left(\frac{N_s}{380.78}\right)^2 \quad \text{... (22.22)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{380.78}\right)^3\right] \quad \text{... (22.23)}$$

For Kaplan turbines, values of σ_c obtained by Eq. 22.23 should be increased by 10 per cent.

In the above expressions N_s is in (r.p.m, kW, m) units. However, if N_s is in (r.p.m, h.p, m) units, the expressions for σ_c would be as follows:

For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{444}\right)^2 \quad \text{... (22.22 a)}$$

For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{444}\right)^3\right] \quad \text{... (22.23 a)}$$

Again for Kaplan turbines, values of σ_c obtained by Eq. 22.23 (a) should be increased by 10 per cent.

(*a*) *Suction Specific Speed.* In addition to Thoma's criterion the consideration of suction specific speed provides another very useful criterion for establishing similarity in respect of cavitation in the turbines. The *suction specific speed S* may be defined as the speed of a geometrically similar turbine such that when it is developing a power equal to 1 kilowatt (or in metric units equal to 1 metric horse power) the total suction head H_{sv} is equal to 1m (in absolute units). According to this definition the expression for suction specific speed may be obtained by replacing the total head H in Eq. 22.11 for the specific speed by the total suction head H_{sv}. Thus

$$S = \frac{N\sqrt{P}}{H_{sv}^{5/4}} \quad \text{... (22.24)}$$

By having the same value of the suction specific speed for the model and the prototype turbines the similarity in respect of cavitation can be established.

The total suction head H_{sv} can be expressed as

$$H_{sv} = H_a - H_v - H_s$$

and hence from Eq. 22.21, we have

$$H_{sv} = \sigma H \quad \text{... (22.25)}$$

By substituting the value of H_{sv} in Eq. 22.24, we get

$$S = \frac{N\sqrt{P}}{(\sigma H)^{5/4}}$$

or

$$\sigma = \left(\frac{N_s}{S}\right)^{4/5} \quad \text{... (22.26)}$$

Equation 22.26 represents the relation between the two parameters σ and S, both of which are useful for establishing a similarity in respect of cavitation in the model and prototype turbines. However, the concept of suction specific speed is more commonly used in the case of pumps.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

22.8 SELECTION OF TURBINES

The selection of a suitable type of turbine is usually governed by the following factors:

(i) Head and Specific Speed. It has been found that there is a range of head and specific speed for which each type of turbine is most suitable which is given in Table 22.1.

TABLE 22.1

S. No.	*Head in metres*	*Type of turbine*	*Specific speed*
1.	300 or more	Pelton wheel Single or Multiple jet	8.5 to 47 (in SI units) 10 to 55 (in metric units)
2.	150 to 300	Pelton or Francis	30 to 85 (in SI units) 35 to 100 (in metric units)
3.	60 to 150	Francis or Deriaz (or Diagonal)	85 to 188 (in SI units) 100 to 220 (in metric units)
4.	Less than 60	Kaplan or Propeller or Deriaz or Tubular	188 to 860 (in SI units) 220 to 1000 (in metric units)

However, as a general rule, it may be stated that as far as possible a turbine with highest permissible specific speed should be chosen, which will not only be the cheapest in itself but its relatively small size and high rotational speed will reduce the size of the generator as well as the power house. But the specific speed cannot be increased indefinitely, because higher specific speed turbine is generally more liable to cavitation. The cavitation may, however, be avoided by installing the turbine at a lower level with respect to the tail race.

(ii) Part Load Operation. The turbines may be required to work with considerable load variations. As the load deviates from the normal working load, the efficiency would also vary. In Fig. 22.5 a plot between η_0 and % of full load has been shown. At part load the performance of Kaplan and Pelton turbines is better in comparison to that of Francis and Propeller turbines. The variability of load will influence the choice of type of turbine if the head lies between 150 m to 300 m or lies below 30 m. For higher range of heads Pelton wheel is preferable for part load operation in comparison to Francis turbine, though the former involves higher initial cost. For heads below 30 m, Kaplan turbine is preferable for part load operation in comparison to Propeller turbine.

In addition to above mentioned factors there are certain other factors to be considered for the selection of the suitable type of a turbine. The *overall cost* which includes the initial cost and the running cost should be considered. The *cavitation* characteristics of the turbine should also be considered since it affects the installation of a reaction turbine.

ILLUSTRATIVE EXAMPLES

Example 22.1. *A Francis turbine working under a head of 5 m at a speed of 210 r.p.m., develops 75 kW when the rate of flow of water is 1.8 m^3/s. The runner diameter is 1 m. If the head on this turbine is increased to 16 m, determine its new speed, discharge and power.*

Solution

$$N_1 = \left[\frac{N\sqrt{H_1}}{\sqrt{H}}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{210\times\sqrt{16}}{\sqrt{5}} = 375.7 \text{ r.p.m.}$$

$$Q_1 = \left[\frac{Q\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \frac{1.8\times\sqrt{16}}{\sqrt{5}} = 3.22 \text{ m}^3/\text{s}$$

$$P_1 = \left[\frac{PH_1^{3/2}}{H^{3/2}}\right]$$

$$= \left[\frac{75\times(16)^{3/2}}{(5)^{3/2}}\right] = 429.33 \text{ kW}$$

Example 22.2. *A hydraulic turbine has an output of 6 600 kW when it works under a head of 25 m and runs at 100 r.p.m. What is the type of turbine? What would be its speed and what power will it develop when working under a head of 16 m?*

Solution

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{100\sqrt{6600}}{(25)^{5/4}} = 145.33$$

In this case since $85 < N_s < 188$, it can be a Francis turbine.

$$N_1 = \left[\frac{N\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \left[\frac{100\sqrt{16}}{\sqrt{25}}\right] = 80 \text{ r.p.m.}$$

$$P_1 = \left[\frac{P(H_1)^{3/2}}{(H)^{3/2}}\right]$$

$$= \left[\frac{6600(16)^{3/2}}{(25)^{3/2}}\right] = 3379.2 \text{ kW}$$

Example 22.3. *A turbine develops 7 460 kW under a head of 24.7 m at 135 r.p.m. What is the specific speed? What would be its normal speed and output under a head of 19.5 m?*

Solution

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{135\times\sqrt{7460}}{(24.7)^{5/4}} = 211.75$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

If N_1 is the speed under a head of 19.5 m, then

$$N_1 = N\left(\frac{H_1}{H}\right)^{1/2}$$

$$= 135\left(\frac{19.5}{24.7}\right)^{1/2} = 119.95 \text{ r.p.m.}$$

Similarly, if P_1 is the output under a head of 19.5 m, then

$$P_1 = P\left(\frac{H_1}{H}\right)^{3/2}$$

$$= 7460\left(\frac{19.5}{24.7}\right)^{3/2} = 5\,232.94 \text{ kW}$$

Examples 22.4. *A Pelton wheel develops 5520 kW under a head of 240 m at an overall efficiency of 80% when revolving at a speed of 200 r.p.m. Find the unit discharge, unit power and unit speed. Assume peripheral coefficient = 0.46. If the head on the same turbine falls during the summer season to 150 m, find the discharge, power and speed for this head.*

Solution

$$\eta_o = \frac{P}{wQH}$$

or $$0.80 = \frac{5520\times10^3}{9810\times Q\times 240}$$

$$\therefore \quad Q = 2.931 \text{ m}^3/\text{s}$$

Unit discharge $$Q_u = \frac{Q}{\sqrt{H}} = \frac{2.931}{\sqrt{240}} = 0.189$$

Unit power $$P_u = \frac{P}{H^{3/2}} = \frac{5520}{(240)^{3/2}} = 1.485$$

Unit speed $$N_u = \frac{N}{\sqrt{H}} = \frac{200}{\sqrt{240}} = 12.91$$

Since peripheral coefficient is assumed to be constant equal to 0.46, the efficiency may be assumed to be constant under different heads and hence the unit quantities will be same under different heads. Thus

$$Q_u = 0.189 = \frac{Q_1}{\sqrt{150}}; \qquad \therefore \; Q_1 = 2.315 \text{ m}^3/\text{s}$$

$$P_u = 1.489 = \frac{P_1}{(150)^{3/2}}; \qquad \therefore \; P_1 = 2\,735.47 \text{ kW}$$

$$N_u = 12.91 = \frac{N_1}{\sqrt{150}}; \qquad \therefore \; N_1 = 158.1 \text{ r.p.m.}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 22.5 *A model of Francis turbine one-fifth of full size, develops 3 kW at 306 r.p.m. under a head of 1.77 m. Find the speed and power of full size turbine operating under a head of 5.7 m, if (a) the efficiency of the model and the full size turbine are same, (b) the efficiency of the model turbine is 76% and the scale effect is considered.*

Solution

(a) For the same efficiency of the model and the prototype, from Eq. 22.17 (a)

$$\frac{D_m N_m}{\sqrt{H_m}} = \frac{D_p N_p}{\sqrt{H_p}}$$

or $$\frac{306}{5\times\sqrt{1.77}} = \frac{N_p}{\sqrt{5.7}}; \qquad \therefore N_p = 109.8 \text{ r.p.m.}$$

Further from Eq. 22.18 (a)

$$\frac{P_m}{D_{m^2} H_m^{3/2}} = \frac{P_p}{D_p^2 H_p^{3/2}}$$

or $$\frac{3}{(1.77)^{3/2}} = \frac{P_p}{(5)^2\times(5.7)^{3/2}}; \qquad \therefore P_p = 433.43 \text{ kW}$$

(b) According to Moody's Eq. 22.20 (a)

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.2}$$

or $$\frac{1-\eta_{op}}{1-0.76} = \left(\frac{1}{5}\right)^{0.2}; \qquad \therefore \eta_{op} = 0.826$$

From Eq. 22.17 (b), we have

$$\frac{D_m N_m}{\sqrt{\eta_{om} H_m}} = \frac{D_p N_p}{\sqrt{\eta_{op} H_p}}$$

or $$\frac{306}{5\times\sqrt{0.76\times1.77}} = \frac{N_p}{\sqrt{0.826\times5.7}}$$

$\therefore \quad N_p = 114.5$ r.p.m.

From Eq. 22.18 (b), we have

$$\frac{P_m}{\eta_{om}^{3/2} H_m^{3/2} D_m^2} = \frac{P_p}{\eta_{op}^{3/2} H_p^{3/2} D_p^2}$$

or $$\frac{3}{(0.76)^{3/2}(1.77)^{3/2}} = \frac{P_p}{(0.826)^{3/2}(5.7)^{3/2}(5)^2}$$

$\therefore \quad P_p = 491.09$ kW

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 22.6. *Estimate the maximum height of straight conical draft tube of 13240 kW Francis turbine running at 150 r.p.m., under a net head of 27 m. The turbine is installed at a station where the effective atmospheric pressure is 10.6 m of water. The draft tube must sink at least 0.77 m below the tail race.*

Solution

Specific speed of the turbine

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{150(13240)^{1/2}}{(27)^{5/4}} = 280.43$$

From Eq. 22.22, we have

$$\sigma_c = 0.625\left(\frac{N_s}{380.78}\right)^2 = 0.625\left(\frac{280.43}{380.78}\right)^2 = 0.339$$

But $$\sigma_c = \frac{H_a - H_v - H_s}{H}$$

$$(H_a - H_v) = 10.6 \text{ m}; H = 27 \text{ m}$$

Thus by substitution, we get

$$0.339 = \frac{10.6 - H_s}{27}; \therefore H_s = 1.45 \text{ m}$$

Hence the maximum length of the draft tube

$$= (1.45 + 0.77) = 2.22 \text{ m}$$

Example 22.7 (*a*) *Define 'unit torque' and derive expression for it.*

(b) Prove that specific speed can be expressed as

$$N_s = 3.13\, N_u \sqrt{Q_u \eta_o}$$

where N_u and Q_u are the unit speed and unit discharge and η_0 is the overall efficiency.

Solution

(a) *Unit torque* may be defined as the torque per unit head acting on the turbine. Thus if T is torque and H is the head acting then unit torque $T_u = \frac{T}{H}$

Torque = Force × distance

i.e., Torque $T \propto$ (Force F)

and $$F = \frac{wQ}{g}(V_w - V_{w_1})$$

i.e., $$F \propto Q V_w$$

but $$Q \propto \sqrt{H} \qquad \text{and } V_w \propto \sqrt{H}$$

Thus $$F \propto H$$

$\therefore$ $$F \propto H \text{; or } T = \text{constant } (H)$$

'Unit torque' T_u is the torque developed due to head equal to 1 m (or unit head)

Thus $$T_u = \text{Constant} \times 1$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad T_u = \frac{T}{H}$$

(b) $$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$P = wQH \times \eta_o$$
$$= 9\,810\, QH\eta_o\, W = 9.81\, QH\,\eta_o \text{ kW}$$

Thus by substitution, we get

$$N_s = \frac{N}{H^{5/4}} \times (9.81\, QH\eta_o)^{1/2}$$

or $$N_s = 3.13 \times \frac{N}{\sqrt{H}} \times \sqrt{\frac{Q}{\sqrt{H}} \times \eta_o} = 3.13\, N_u \sqrt{Q_u \eta_o}$$

Example 22.8. *In a Kaplan turbine the discharge is 25 m^3/s under a net head of 10 m at the design speed of 250 r.p.m. The runner and hub diameters are 2 m and 1 m respectively. The vapour and barometric pressures are 1.6 m and 10 m respectively. The pressure recorded at inlet to the draft tube is 3 m vacuum, where the diameter of the draft tube is 2 m. The exit diameter of the draft tube is 3 m and the efficiency of the draft tube is 0.80. The Thoma's cavitation parameter for the turbine is given by the relation $\sigma = \eta_d\, \psi^2 + \lambda\, K_u^2$, in which η_d is the efficiency of the draft tube, ψ is flow ratio, K_u is speed ratio and λ is a dimensionless factor defined by $\lambda = \frac{(p_1/w) - (p_{min}/w)}{(u^2/2g)}$ in which u is the tangential velocity, (p_1/w) is the pressure head at inlet to draft tube, and (p_{min}/w) is the minimum pressure head at a point on the blade. Calculate the minimum pressure on the blade. Also calculate N_s, if $\eta_o = 0.90$.*

Solution

$$Q = \frac{\pi}{4}(D^2 - d^2)\,\psi\sqrt{2gH}$$

or $$25 = \frac{\pi}{4}(2^2 - 1^2)\,\psi\,\sqrt{2\times 9.81\times 10}$$

$$\therefore \quad \psi = 0.757$$

$$u = \frac{\pi DN}{60} = K_u\sqrt{2gH}$$

or $$\frac{\pi\times 2\times 250}{60} = K_u\sqrt{2\times 9.81\times 10}$$

$$\therefore \quad K_u = 1.87$$

Also $$u = \frac{\pi DN}{60} = \frac{\pi\times 2\times 250}{60} = 26.18 \text{ m/s}$$

Applying Bernoulli's equation between the inlet of the draft tube and the water surface of the tail race channel, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{p_1}{w} + H_s + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g} + h_f$$

where h_f is the loss of head in the draft tube which is given as

$$h_f = \frac{V_1^2 - V_2^2}{2g} - \eta_d \frac{V_1^2}{2g}$$

and $$\frac{p_2}{w} = \frac{p_{atm}}{w}$$

$$\therefore \quad H_s = \frac{p_{atm}}{w} - \frac{p_1}{w} - \eta_d \frac{V_1^2}{2g}$$

$$V_1 = \frac{25}{\pi(2)^{2/4}} = 7.96 \text{ m/s}$$

$$\frac{p_1}{w} = (10 - 3) = 7 \text{ m absolute}$$

By substitution, we obtain

$$H_s = 10 - 7 - 0.80 \times \frac{(7.96)^2}{2 \times 9.81}$$

$$= 0.42 \text{ m}$$

Thoma's cavitation parameter is also given by

$$\sigma = \frac{H_a - H_v - H_s}{H}$$

Thus $$\sigma = \frac{10 - 1.6 - 0.42}{10} = 0.798$$

As given $$\sigma = \eta_d \psi^2 + \lambda K_u^2$$

Thus by substitution, we have

$$0.798 = 0.80(0.757)^2 + \lambda (1.87)^2$$

$$\therefore \quad \lambda = 0.097$$

but $$\lambda = \frac{(p_1 / w) - (p_{min} / w)}{(u^2 / 2g)}$$

or $$(0.097) \times \frac{(26.18)^2}{2 \times 9.81} = -3 - \frac{p_{min}}{w}$$

or $$\frac{p_{min}}{w} = 6.389 \text{ m vacuum}$$

or $$p_{min} = 62.68 \text{ kN/m}^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further from the Example 22.7, we have

$$N_s = 3.13\, N_u \sqrt{Q_u \eta_o}$$

or
$$N_s = 3.13 \times \frac{250}{\sqrt{10}} \times \sqrt{\frac{25}{\sqrt{10}} \times 0.90} = 660$$

Example 22.9. *Calculate the value of Thoma's cavitation parameter for a Kaplan turbine having non-dimensional specific speed of 19.36 and suction specific speed of 17.6.*

Solution

The non-dimensional specific speed n_s is given by

$$n_s = \frac{N_s}{17.362}$$

$$\therefore \quad N_s = (17.362 \times n_s) = (17.362 \times 19.36)$$

Similarly the non-dimensional suction specific speed is given by

$$\sigma = \frac{S}{17.362}$$

$$\therefore \quad S = (17.362 \times s) = (17.362 \times 17.6)$$

From Eq. 22.26, we have

$$\sigma = \left(\frac{N_s}{S}\right)^{4/5}$$

$$\therefore \quad \sigma = \left(\frac{17.362 \times 19.36}{17.362 \times 17.6}\right)^{4/5} = 1.08$$

Example 22.10. *A model of a turbine 0.3 m diameter when running at 900 r.p.m., under a head of 12 m gave the following results:*

Percentage gate opening	*40*	*60*	*80*	*100*
Output power kW	*8.38*	*13.24*	*17.50*	*19.71.*

The actual turbine is required to develop 7360 kW at 85% gate opening under a head of 35 m. Assuming same efficiency at corresponding gate opening for the model and actual turbines, find (i) diameter and speed of the actual turbine, (ii) output of the actual turbine at 50% gate opening.

Solution

From the plot of percentage gate opening *v/s* output power for the given data, for 85% gate opening the output power of the model is obtained as

$$P_m = 18.20 \text{ kW}$$

Thus from Eq. 20.18 (a), we have

$$\left(\frac{P}{H^{3/2} D^2}\right)_m = \left(\frac{P}{H^{3/2} D^2}\right)_p$$

or
$$\left[\frac{18.20}{(12)^{3/2} (0.3)^2}\right]_m = \left[\frac{7360}{(35)^{3/2} D^2}\right]_p$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore \quad D_p = 2.70$ m

Further from Eq. 22.17 (a), we have

$$\left(\frac{H}{N^2D^2}\right)_m = \left(\frac{H}{N^2D^2}\right)_p$$

or $$\left[\frac{12}{(900)^2(0.3)^2}\right]_m = \left[\frac{35}{N^2(2.7)^2}\right]_p$$

$\therefore \quad N_p = 171$ r.p.m.

From the same plot for 50% gate opening the output power of the model is

$P_m = 11.03$ kW

Thus from Eq. 20.18 (a), we have

$$\left(\frac{P}{H^{3/2}D^2}\right)_m = \left(\frac{P}{H^{3/2}D^2}\right)_p$$

or $$\left[\frac{11.03}{(12)^{3/2}(0.3)^2}\right]_m = \left[\frac{P}{(35)^{3/2}(2.7)^2}\right]_p$$

$\therefore \quad P_p = 4\,450.3$ kW

Example 22.11. *For a large Pelton turbine the design values of K_v, K_u, k and m are respectively 0.97, 0.46, 0.98 and 10.2 (K_v is the ratio of jet velocity to the spouting velocity $\sqrt{2gH}$, K_u is the ratio of the circumferential velocity of the wheel to the spouting velocity, k is the ratio of the relative velocities at exit and inlet of the bucket and m is the ratio of the wheel diameter to the jet diameter).*

Determine the specific speed of the turbine if the bucket angle at exit is 10° and the mechanical efficieny of the turbine is 90.5%.

Solution

Specific speed of Pelton turbine is given by Eq. 22.12 as

$$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{m}$$

The overall efficiency η_o is given by

$$\eta_o = \eta_h \times \eta_m$$

where η_h is hydraulic efficiency and η_m is mechanical efficiency of Pelton turbine.

The hydraulic efficiency of Pelton turbine is given by

$$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2}$$

in which by introducing $u = K_u\sqrt{2gH}$; and

$V = K_v\sqrt{2gH}$, we get

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\eta_h = \frac{2K_u(K_v - K_u)(1 + k\cos\phi)}{K_v^2}$$

$K_u = 0.46;\ K_v = 0.97;\ k = 0.98;$ and $\phi = 10°$

Thus by substitution, we get

$$\eta_h = \frac{2\times 0.46\times(0.97 - 0.46)\times(1 + 0.98\cos 10°)}{(0.97)^2}$$

$$= \frac{2\times 0.46\times(0.97 - 0.46)\times(1 + 0.98\times 0.9848)}{(0.97)^2}$$

$= 0.98$ or 98%

$\eta_m = 90.5\%$ or 0.905 (given)

$\therefore\ \eta_o = 0.98 \times 0.905 = 0.887$

and $m = 10.2$

Thus by substitution, we get

$$N_s = \frac{494.24\times 0.46\times(0.97\times 0.887)^{1/2}}{10.2}$$

$= 20.67$

Example 22.12. *A vertical shaft Kaplan turbine operating under a head of 9.8 m has a runner diameter of 9.3 m. At the optimum regime the turbine runs at 51.7 r.p.m and develops 45000 kW power discharging 535 m^3/s water. Determine the values of unit speed, unit discharge, unit power and specific speed of the turbine.*

If the critical value of cavitation coefficient for the runner is 0.95, determine the location of the runner with respect to the tail race water level. The atmospheric and saturated vapour pressures may be taken as 0.98 kg(f)/cm^2 and 0.25 kg(f)/cm^2 respectively.

Solution

$H = 9.8$ m; $D = 9.3$ m; $N = 51.7$ r.p.m; $P = 45\,000$ kW; and $Q = 535\ m^3/s$

From Eq. 22.5(a), unit speed is given as

$$N_u = \frac{ND}{\sqrt{H}}$$

$$= \frac{51.7\times 9.3}{\sqrt{9.8}} = 153.59$$

From Eq. 22.6 (a), unit discharge is given as

$$Q_u = \frac{Q}{D^2\sqrt{H}}$$

$$\frac{535}{(9.3)^2\sqrt{9.8}} = 1.98$$

From Eq. 22.7(a), unit power is given as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$P_u = \frac{P}{D^2 H^{3/2}}$$

$$= \frac{45000}{(9.3)^2 (9.8)^{3/2}} = 16.96$$

From Eq. 22.11, specific speed is given as

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$= \frac{51.7\sqrt{45000}}{(9.8)^{5/4}} = 632.51$$

From Eq. 22.21, we have

$$\sigma = \frac{H_a - H_v - H_s}{H}$$

$$\sigma = \sigma_c = 0.95; H_a = \frac{0.98 \times 10^4}{1000} = 9.8 \text{ m; and}$$

$$H_v = \frac{0.25 \times 10^4}{1000} = 2.5 \text{ m}$$

Thus by substitution, we get

$$0.95 = \frac{9.8 - 2.5 - H_s}{9.8}$$

$$\therefore \quad H_s = -2.01 \text{ m}$$

The suction head H_s is negative which means that the turbine runner must be located 2.01 m below the tail race water level.

Example 22.13 *The model of a water turbine tested in the laboratory at a head of 20 m just began to cavitate when the static suction head was raised to 7 m. Its optimum hydraulic efficiency was 82%. The prototype turbine is required to work at a head of 50 m. Determine the maximum safe setting for the turbine above the tail race. Atmospheric conditions in the laboratory and at the site are given below:*

	Atmospheric pressure	*Vapour pressure corresponding To water temperature*
At site	1×10^5 *N/m*^{2}	0.035×10^5 *N/m*^{2}
In laboratory	1.012×10^5 *N/m*^{2}	0.033×10^5 *N/m*^{2}

The Reynolds number of model test was $Re = 6 \times 10^5$. *The prototype Reynolds number is* $Re = 1.5 \times 10^8$. *Estimate the prototype efficiency explaining the reasoning behind any formula used.*

Solution

The similarity between the model and the prototype in respect of cavitation characteristics can be ensured if Thoma's cavitation number is same in the model and the prototype. That is

$$(\sigma)_m = (\sigma)_p$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\left(\frac{H_a - H_v - H_s}{H}\right)_m = \left(\frac{H_a - H_v - H_s}{H}\right)_p$$

$$(H_a)_m = \frac{1.012\times10^5}{9810} = 10.316 \text{ m}; \; (H_v)_m = \frac{0.033\times10^5}{9810} = 0.336 \text{ m};$$

$$(H_s)_m = 7 \text{ m; and } (H)_m = 20 \text{ m}$$

$$(H_a)_p = \frac{1\times10^5}{9810} = 10.194 \text{ m}; \; (H_v)_p = \frac{0.035\times10^5}{9810} = 0.357 \text{ m}$$

and $$(H)_p = 50 \text{ m}$$

Thus by substitution, we get

$$\frac{(10.316 - 0.336 - 7)}{20} = \frac{(10.194 - 0.357 - H_s)}{50}$$

or $$\frac{2.98}{20} = \frac{9.837 - H_s}{50}$$

$\therefore$ $$H_s = 2.387 \text{ m}$$

Thus the maximum safe setting for the turbine above the tail race is 2.387 m.

The Reynolds number may be expressed as

$$R_e = \frac{\rho N D^2}{\mu}$$

For the model turbine

$$(\text{Re})_m = \left(\frac{\rho N D^2}{\mu}\right)_m = 6\times10^5$$

or $$(ND)_m = \left(\frac{6\times10^5\times\mu}{\rho D}\right)_m \quad ...(i)$$

For the prototype turbine

$$(\text{Re})_p = \left(\frac{\rho N D^2}{\mu}\right)_p = 1.5\times10^8$$

or $$(ND)_p = \left(\frac{1.5\times10^8\times\mu}{\rho D}\right)_p \quad ...(ii)$$

For similarity between the model and the prototype turbines from Eq. 22.17(a), we have

$$\left(\frac{\sqrt{\eta_o H}}{ND}\right)_m = \left(\frac{\sqrt{\eta_o H}}{ND}\right)_p \quad ...(iii)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Introducing the values of (ND) for the model and the prototype turbines from Eqs (i) and (ii) in Eq. (iii), we get

$$\left(\frac{\sqrt{\eta_o H}\times\rho D}{6\times10^5\times\mu}\right)_m = \left(\frac{\sqrt{\eta_o H}\times\rho D}{1.5\times10^8\times\mu}\right)_p$$

Assuming ρ and μ to be same for the model and the prototype turbines, we get

$$\left(\frac{\sqrt{\eta_o H}\times D}{6\times10^5}\right)_m = \left(\frac{\sqrt{\eta_o H}\times D}{1.5\times10^8}\right)_p$$

or
$$\frac{D_m}{D_p} = \frac{\sqrt{\eta_{op} H_p}}{\sqrt{\eta_{om} H_m}}\times\frac{6\times10^5}{1.5\times10^8} = \frac{\sqrt{\eta_{op} H_p}}{250\times\sqrt{\eta_{om} H_m}} \quad \ldots(iv)$$

From Eq. 22.20 (a), we have

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.2}$$

Introducing the value of (D_m/D_p) from Eq. (iv), we get

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{\sqrt{\eta_{op} H_p}}{250\times\sqrt{\eta_{om} H_m}}\right)^{0.2}$$

η_{om} = 82% = 0.82; H_m = 20 m; and H_p = 50 m

Thus by substitution, we get

$$\frac{1-\eta_{op}}{1-0.82} = \left(\frac{\sqrt{\eta_{op}\times50}}{250\times\sqrt{0.82\times20}}\right)^{0.2}$$

or
$$\frac{1-\eta_{op}}{0.18} = \frac{(\eta_{op})^{0.1}}{2.699}$$

Solving for η_{op} by trial and error, we get

η_{op} = 0.9337 or 93.37%

Thus efficiency of prototype turbine is 93.37%

Equation 22.20 (a) which correlates the efficiencies of the model and the prototype turbines has been used because it is applicable for head less than 150 m.

Example 22.14. *Show that the expressions for the non-dimensional numbers (i) specific speed, (ii) specific diameter, (iii) flow number, and (iv) head number for Pelton wheel are functions of only the diameter ratio (d/D) where d is the jet diameter and D is Pelton wheel diameter. It is assumed that friction effect is neglected and 180° reversal of flow is achieved by the Pelton wheel buckets.*

Find out the values of the above dimensionless numbers if diameter ratio d/D is 0.2.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Assuming $K_v = 1$ and $K_u = 0.5$, then for the given conditions the overall efficiency $\eta_0 = 1$.

(i) The non-dimensional specific speed is given by

$$n_s = \frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}$$

or $$n_s = \frac{N\sqrt{wQH\eta_o}}{(\rho g)^{1/2} g^{3/4} H^{5/4}}$$

or $$n_s = \frac{N\sqrt{Q}}{(gH)^{3/4}}$$

Since $w = \rho g$ and $\eta_o = 1$

Further $$Q = \frac{\pi d^2}{4} \times K_v\sqrt{2gH} = \frac{\pi d^2}{4} \times \sqrt{2gH}$$

Since $K_v = 1$

Also $$u = \frac{\pi DN}{60} = K_u\sqrt{2gH}$$

or $$N = \frac{60K_u\sqrt{2gH}}{\pi D} = \frac{30\sqrt{2gH}}{\pi D}$$

since $K_u = 0.5$

Thus by substitution, we get

$$n_s = \frac{30\sqrt{2gH} \times \sqrt{\left(\frac{\pi d^2}{4}\right)\sqrt{2gH}}}{\pi D \times (gH)^{3/4}}$$

or $n_s = 14.23\ (d/D)$

For $(d/D) = 0.2$

$n_s = (14.23 \times 0.2) = 2.846$

(ii) The non-dimensional specific diameter is given by Eq. 22.9 as

$$D_s' = \left(\frac{D_s}{D}\right) = \left(\frac{P_s}{P_u}\right)^{1/2} = \frac{1}{\sqrt{P_u}}\ ;\ \text{since } P_s = 1 \text{ kW}$$

$$P_u = \left(\frac{wQ_u \times 1}{1000}\right)\eta_o \text{ kW}$$

$$Q_u = \frac{Q}{D^2\sqrt{H}} = \frac{(\pi/4)d^2 \times \sqrt{2gH}}{D^2\sqrt{H}}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus $$D_s' = \frac{1}{\sqrt{\frac{9810}{1000}\times\frac{\pi}{4}\times\left(\frac{d}{D}\right)^2\times\sqrt{2g}}}$$

or $$D_s' = \frac{1}{5.84(d/D)}$$

For $(d/D) = 0.2$

$$D_s' = \frac{1}{5.84\times 0.2} = 0.856$$

(iii) The non-dimensional flow number is given by

$$Q' = \frac{Q}{ND^3} = \frac{(\pi/4)d^2\times\sqrt{2gH}}{\frac{30\times\sqrt{2gH}}{\pi D}\times D^3}$$

or $$Q' = 0.082\,(d/D)^2$$

For $(d/D) = 0.2$

$$Q' = 0.082\times(0.2)^2 = 0.003\,28$$

(iv) The non-dimensional head number is given by

$$H' = \frac{gH}{N^2D^2}$$

or $$H' = \frac{\pi^2N^2D^2}{(30\sqrt{2})^2N^2D^2} = 0.005\,48$$

which is however independent of the ratio (d/D).

Example 22.15. *Calculate the diameter, the speed and N_s of a propeller turbine runner to develop 6250 kW under a head of 5 m, having given that speed ratio K_u based on outer diameter = 2.10, flow ratio $\psi = 0.65$; diameter of boss = 0.35 times external diameter of the runner; overall efficiency $\eta_o = 85\%$.*

Solution

$$\eta_o = 0.85 = \frac{6250\times10^3}{(9810\times Q\times 5)}$$

$\therefore$ $$Q = 149.91\ \text{m}^3/\text{s}$$

$$\psi = 0.65 = \frac{V_f}{\sqrt{2\times9.81\times5}}$$

$\therefore$ $$V_f = 6.44\ \text{m/s}$$

$$Q = 149.91 = \frac{\pi}{4}\left[D^2 - (0.35D)^2\right]\times 6.44$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad D = 5.81 \text{ m}$$

$$K_u = 2.10 = \frac{u}{\sqrt{2 \times 9.81 \times 5}}$$

$$\therefore \quad u = 20.8 \text{ m/s}$$

$$u = 20.8 = \frac{\pi DN}{60} = \frac{\pi \times 5.81 \times N}{60}$$

$$\therefore \quad N = 68.37 \text{ r.p.m.}$$

$$N_s = \left(\frac{N\sqrt{P}}{H^{5/4}}\right)$$

$$= \frac{68.37 \times (6250)^{1/2}}{(5)^{5/4}} = 722.93$$

Example 22.16 *The following data were obtained from the main characteristics of a Kaplan turbine of runner diameter 1 m: $P_u = 30.695$; $Q_u = 108.6$; $N_u = 63.6$. Estimate the runner diameter, the discharge and the speed of a similar runner working under a head of 30 m and developing 2000 kW. Determine the specific speed of the runner.*

Solution

The two runners will be similar if

$$\left(\frac{Q_1}{N_1 D_1^3}\right) = \left(\frac{Q_2}{N_2 D_2^3}\right) ; \left(\frac{N_1 D_1}{\sqrt{H_1}}\right) = \left(\frac{N_2 D_2}{\sqrt{H_2}}\right) ;$$

and

$$\left(\frac{N_1\sqrt{P_1}}{H_1^{5/4}}\right) = \left(\frac{N_2\sqrt{P_2}}{H_2^{5/4}}\right)$$

From these conditions the following identity may be obtained

$$\frac{P_1}{D_1^2 H_1^{3/2}} = \frac{P_2}{D_2^2 H_2^{3/2}}$$

or

$$\frac{30.695}{(1)^2} = \frac{2000}{D_2^2 \times (30)^{3/2}} ; \quad \therefore D_2 = 0.629\,7 \text{ m}$$

$$\frac{N_1 D_1}{\sqrt{H_1}} = \frac{N_2 D_2}{\sqrt{H_2}}$$

or

$$(63.6 \times 1) = \frac{N_2 \times 0.6297}{(30)^{1/2}} ; \quad \therefore N_2 = 553.20 \text{ r.p.m.}$$

Again from the above noted conditions the following identity may be obtained

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{Q_1}{D_1^2\sqrt{H_1}} = \frac{Q_2}{D_2^2\sqrt{H_2}}$$

or $$\frac{108.6}{(1)^2} = \frac{Q_2}{(0.6297)^2 \times (30)^{1/2}}$$

$\therefore$ $$Q_2 = 235.86 \text{ m}^3/\text{s}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = N_u\sqrt{P_u}$$

For the first runner

$$N_s = 63.6\sqrt{30.695} = 352.36$$

For the second runner

$$N_s = \frac{553.20\sqrt{2000}}{(30)^{5/4}} = 352.36$$

The two values agree, as they ought to do.

Example 22.17 *A water turbine develops 130 kW at 230 r.p.m., under a head of 16 m. Determine the scale ratio and the speed of a similar machine which will generate 660 kW when working under a head of 25 m.*

Solution

Since $$\frac{P_1}{D_1^2 H_1^{3/2}} = \frac{P_2}{D_2^2 H_2^{3/2}}$$

or $$\frac{130}{D_1^2 (16)^{3/2}} = \frac{660}{D_2^2 (25)^{3/2}}$$

$\therefore$ Scale ratio $\frac{D_2}{D_1} = 1.612$

Also $$\frac{N_1 D_1}{\sqrt{H_1}} = \frac{N_2 D_2}{\sqrt{H_2}}$$

or $$\frac{230 \times D_1}{\sqrt{16}} = \frac{N_2 \times (1.612 D_1)}{\sqrt{25}}$$

$\therefore$ $$N_2 = 178.35 \text{ r.p.m.}$$

Example 22.18 *The quantity of water available for a hydroelectric station is 275 m^3/s under a head of 18 m. Assuming the speed of the turbines to be 150 r.p.m., and their efficiency 82%, determine the least number of machines, all of the same size, that will be needed if (i) Francis turbines whose N_s must not exceed 395, (ii) Kaplan turbines whose N_s must not exceed 690, are chosen. What would be the individual output of units in the two cases?*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Total power to be developed by the turbines in kW

$$= \left(\frac{wQH}{10^3} \times \eta_o\right)$$

$$= \frac{9810 \times 275 \times 18 \times 0.82}{10^3} = 39\ 819 \text{ kW}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

(i) For Francis turbines considering the limiting value of N_s as 395, we have

$$395 = \frac{150\sqrt{P}}{(18)^{5/4}}$$

$$\therefore \quad P = 9\ 532 \text{ kW}$$

$\therefore$ Number of turbines needed

$$= \frac{39819}{9532} \simeq 5$$

(ii) For Kaplan turbines taking $N_s = 690$, we have

$$690 = \frac{150\sqrt{P}}{(18)^{5/4}}$$

$$\therefore \quad P = 29\ 087 \text{ kW}$$

$\therefore$ Number of turbines needed

$$= \frac{39819}{29087} \simeq 2.$$

Example 22.19. *A 90 MW hydroelectric plant is installed with reaction turbines with head = 26.5 m, overall efficiency = 85%, speed ratio (based on outer diameter) = 2.0, flow ratio = 0.65, diameter of boss = 0.35 × external diameter of runner. The turbine runner is of propeller turbine running at 192 r.p.m. Determine the specific speed of turbine and number of turbines needed.*

Solution

For a propeller turbine from Eq. 22.15, we have

$$N_s = C_1 \sqrt{C_2}\left[K_u \sqrt{\psi(1-n^2)\eta_0}\right]$$

where

$$C_1 = \frac{60\sqrt{2g}}{\pi} = 84.6; \text{ and}$$

$$C_2 = \frac{w\pi\sqrt{2g}}{4} = 34.13$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Also $K_u = 2.0$; $\psi = 0.65$; $n = 0.35$; and $\eta_0 = 0.85$

Thus by substitution, we get

$$N_s = 84.6\sqrt{34.13}\left[2.0\times\sqrt{0.65[1-(0.35)^2]\times 0.85}\right]$$

$$\therefore \quad N_s = 688.27$$

From Eq. 22.11, we have

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$N = 192$ r.p.m; and $H = 26.5$ m

Thus by substitution, we get

$$688.27 = \frac{192\sqrt{P}}{(26.5)^{5/4}}$$

$$\therefore \quad P = 46\,454.7 \text{ kW}$$

Total installed capacity at the plant

$= 90$ MW $= 90\,000$ kW

$\therefore$ Number of turbines needed

$$= \frac{90000}{46454.7} \simeq 2$$

Alternative Method

$$u = K_u\sqrt{2gH} = \frac{\pi DN}{60}$$

Thus by substitution, we get

$$2.0 \times \sqrt{2\times 9.81\times 26.5} = \frac{\pi D\times 192}{60}$$

$$\therefore \quad D = 4.536 \text{ m}$$

$$V_f = \psi\sqrt{2gH}$$

or $$V_f = 0.65 \times \sqrt{2\times 9.81\times 26.5} = 14.821 \text{ m/s}$$

$$Q = \frac{\pi}{4} D^2 (1 - n^2) \times V_f$$

or $$Q = \frac{\pi}{4} \times (4.536)^2 \times [1 - (0.35)^2] \times 14.821$$

or $$Q = 210.165 \text{ m}^3/\text{s}$$

$$\eta_0 = \frac{P}{wQH}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$0.85 = \frac{P}{9810 \times 210.165 \times 26.5}$$

$$\therefore \quad P = 0.85 \times 9810 \times 210.165 \times 26.5 \text{ W}$$
$$= 46.44 \times 10^6 \text{ W}$$
$$= 46.44 \times 10^3 \text{ kW}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$\therefore \quad N_s = \frac{192 \times \sqrt{46.44 \times 10^3}}{(26.5)^{5/4}} = 688.16$$

Number of turbines needed

$$= \frac{90000}{46.44 \times 10^3} \simeq 2.$$

Example 22.20. *A turbine is to operate under a head of 25 m at 200 r.p.m. If the discharge is 9 m^3/s and turbine efficiency is 90%, calculate power generated by the turbine, specific speed of the turbine and performance of the turbine under a head of 20 m. Also state the type of turbine.*

Solution

$$\eta_o = \frac{P}{wQH}$$

or $$P = wQH \times \eta_o$$

Thus by substitution, we get

$$P = 9\,810 \times 9 \times 25 \times 0.9 \text{ W}$$
$$= 198.65 \times 10^4 \text{ W}$$
$$= 1986.5 \text{ kW}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

Thus by substitution, we get

$$N_s = \frac{200\sqrt{1986.5}}{(25)^{5/4}} = 159.46$$

For this value of N_s this can be a Francis turbine.

The performance of the turbine under a head of 20 m is indicated by the following expressions.

$$N_1 = \left[\frac{N\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \left[\frac{200 \times \sqrt{20}}{\sqrt{25}}\right] = 178.89 \text{ r.p.m.}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q_1 = \left[\frac{Q\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \left[\frac{9\times\sqrt{20}}{\sqrt{25}}\right] = 8.05 \text{ m}^3/\text{s}$$

$$P_1 = \left[\frac{PH_1^{3/2}}{H^{3/2}}\right]$$

$$= \left[\frac{1986.5\times(20)^{3/2}}{(25)^{3/2}}\right] = 1421.4 \text{ kW}$$

SUMMARY OF MAIN POINTS

1. The speed, discharge and power developed for a turbine when operating under a unit head are known as unit speed, unit discharge and unit power which may be expressed in terms of the corresponding quantities for the same turbine operating under a head H as indicated below.

   Unit speed $N_u = \dfrac{N}{\sqrt{H}}$

   Unit speed $u_u = \dfrac{u}{\sqrt{H}}$

   Unit discharge $Q_u = \dfrac{Q}{\sqrt{H}}$

   Unit power $P_u = \dfrac{P}{H^{3/2}}$

   The unit quantities facilitate the comparison between the performances of the turbines of the same type but having different outputs and speeds and working under different heads.

2. The turbines of different types may be compared by considering an imaginary turbine called specific turbine. The *specific turbine* is an imaginary turbine which is identical in shape, geometrical proportions, blade angles, gate setting, etc., with the actual turbine but reduced to such a size that it will develop one kilowatt power (or one metric horse power) under unit head.

   The speed of specific runner is known as *specific speed* N_s which may be defined as follows:

   The specific speed of any turbine is the speed in r.m.p of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 kilowatt power (or 1 metric horse power) when working under unit head (i.e., 1 metre). It is given by

   $$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

3. The expressions for all the specific speed in terms of known coefficients for different turbines are as given below.

   (i) For Pelton wheel

   (a) In SI units

   $$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{m}$$

   (b) In metric units

   $$N_s = \frac{(576.2)K_u(K_v\eta_o)^{1/2}}{m}$$

   (ii) For Francis turbine

   (a) In SI units

   $$N_s = 988.45\left[K_u\left(\sqrt{kn\ \psi\ \eta_o}\right)\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) In metric units

$$N_s = 1152.4 \left[K_u\left(\sqrt{kn\ \psi\ \eta_o}\right)\right]$$

(iii) For Kaplan turbine

(a) In SI units

$$N_S = 439.22\ K_u\left(\sqrt{\psi}\right)$$

(b) In metric units

$$N_s = 512.07\ K_u\left(\sqrt{\psi}\right)$$

4. The performance characteristic curves for the turbines are of the following three types;
   (i) Constant head characteristic curves, (or Main characteristic curves)
   (ii) Constant speed characteristic curves, (or operating characteristic curves)
   (iii) Constant efficiency curves, (or Iso-efficiency curves)
5. For the complete similarity to exist between the model and the protoype turbines the following conditions are required to be satisfied.

(i) $\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$

(ii) $\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p$

(iii) $\left(\frac{P}{\rho gHND^3}\right)_m = \left(\frac{P}{\rho gHND^3}\right)_p$

(iv) $\left(\frac{N\sqrt{P}}{H^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{H^{5/4}}\right)_p$

6. The expression relating the efficiencies of the model and the prototype turbines is given as

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{\alpha}\left(\frac{H_m}{H_p}\right)^{\beta}$$

where

$\alpha = 0.20$ and $\beta = 0$ for $H < 150$ m; and
$\alpha = 0.25$ and $\beta = 0.10$ for $H > 150$ m

7. In reaction turbines cavitation may occur at the runner exit or the inlet to the draft tube where the pressure is considerably reduced.

   In order to avoid cavitation, the value of Thoma's cavitation factor should not be reduced belowthe critical cavitation factor σ_c which for different turbines is given as

   (i) When specific speed N_s is in (r.p.m., kW, m) units

   (a) For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{380.78}\right)^2$$

   (b) For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{380.78}\right)^2\right]$$

   For Kaplan turbines the values of σ_c obtained by above equation should be increased by 10 per cent.

   (ii) When specific speed N_s is in (r.p.m, h.p., m)

   (a) For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{444}\right)^2$$

   (b) For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{444}\right)^2\right]$$

   Again for Kaplan turbines the values of σ_c obtained by the above equation should be increased by 10 per cent.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

PROBLEMS

22.1 Explain (a) Unit speed, (b) Unit discharge, (c) Unit Power of a hydraulic turbine. Derive expressions for each of them.

22.2 (a) Explain the terms specific speed, unit speed and unit power as applied to hydraulic turbines. Deduce expressions to indicate their values.

(b) A turbine develops 7355 kW under a head of 24.7 m at 210 r.p.m. What is its specific speed? Indicate the type of turbine suitable for the purpose. If this turbine is tested in the laboratory where the head of water available is only 7.5 m, what power will it develop and at what speed?

[*Ans.* 327 r.p.m; Kaplan or Deriaz; 1231 kW; 115.7 r.p.m.]

22.3 A Francis turbine runner having a diameter of 2.92 m operates at 163.5 r.p.m., under 54 m head, and develops 19 900 kW at an efficiency of 87%. Find the other characteristics if this turbine is operated under 60 m head.

[*Ans.* 172.3 r.p.m.; 45.51 m^3/s; 23 307 kW]

22.4 An impulse turbine at best speed produces 92 kW under a head of 63 m. By what per cent should the speed be increased for a head of 87 m ? [*Ans.* 17.5%]

22.5 In an inward flow reaction turbine the speed ratio is K_u, the flow ratio is ψ, and the breadth ratio is n. If the overall efficiency of the turbine is 85% and the area taken up by the vanes at inlet is 5% of the peripheral area at inlet, then show that $N_s = 888.23\ K_u \left(\sqrt{\psi n}\right)$ (in SI units).

A runner of the above type having $N_s = 185$ is required to develop 6620 kW unde a head of 90 m. Take $\psi = 0.18$, $n = 0.2$, calculate speed and diameter of the runner.

[*Ans.* 630.3 r.p.m., 1.4 m]

22.6 What are the characteristic curves of a hydraulic turbine? How are they useful to practical Engineer? How are small scale models useful in obtaining these curves for a proposed turbine of a hydroelectric installation?

22.7 Explain how the 'best performance curve' can be obtained with the help of characteristic curves of a reaction turbine? If the turbine has to work at a given head and develop given power at maximum efficiency, how this curve can be used to determine the running speed, gate opening, discharge and efficiency.

22.8 What is meant by 'cavitation' ? What is Thoma's cavitation factor, and what its significance for water turbines?

22.9 Calculate the specific speed of a Francis turbine runner for which $K_u = 1.05$, $\psi = 0.25$ and the width of the runner at the outer periphery is 0.25 times the outer diameter. Take h_o as 0.80 and area occupied by vanes as negligible.

[*Ans.* 232.08 r.p.m.]

22.10 The following observations were obtained from a test carried on a turbine working under its designed head of 8.5 m:

Unit speed N_u	56	65	74	84	93	102
Unit power, P_u	8.83	9.19	9.49	9.49	9.34	8.97
Weight flow W (kN/s)	38.85	38.16	37.38	36.54	35.61	34.53

Plot a curve of η_o against N_u. Calculate the turbine N at maximum η_o and hence determine N_s of the turbine. If the head is changed to 10 m, estimate the power developed and find from the graph η_o at a turbine speed of 250 r.p.m.

[*Ans.* 271 r.p.m; 284 r.p.m; 295.4 kW; 75%]

22.11 A model turbine has a runner of diameter 0.61 m. It develops 50 kW under a head of 30 m at a speed of 4000 r.p.m. Compute N_s and N_u for this model. It is required to build a similar turbine to develop 155 kW under a head of 36 m. Calculate the required diameter.

[*Ans.* 402.85 r.p.m.; 730.3 r.p.m.; 0.937 m]

22.12 A 1/4 scale turbine model is tested under a head of 12 m. The prototype turbine is required to work under a head of 30 m at a speed of 425 r.p.m. At what speed the model be run and if it develops 100 kW and uses 1.14 m^3/s of water at this speed, what power will be obtained from the prototype turbine, assuming that its efficiency is 3% better than that of the model? Also indicate the type of runner used in this turbine.

[*Ans.* 1054 r.p.m., 6710 kW; Kaplan (N_s = 496)]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

22.13 Briefly explain the various considerations in the selection of a proper type of turbine for a hydroelectric station, indicating also the conditions where a particular type of turbine is suitable.

22.14 Estimate the maximum height of a conical draft tube for a Kaplan turbine developing 14 710 kW under a net head of 20 m and running at 145 r.p.m. The turbine is installed at a station where the effective atmospheric pressure is 10.4 m of water. The draft tube must sink at least 1 m below the tail race level. [*Ans.* 1.4 m]

22.15 A model having scale ratio $\frac{1}{10}$ is constructed to determine the best design of Kaplan turbine to develop 7355 kW under a net head of 10 m at a speed of 100 r.p.m. If the head available at the laboratory is 6 m and the model efficiency is 88% find: (i) running speed of the model; (ii) the flow required in the laboratory; (iii) the output power of the model; (iv) the specific speed in each case. Assume same efficiency for the model and prototype turbine.

[*Ans.* (i) 774.6 r.p.m.; (ii) 0.66 m^3/s; (iii) 34.19 kW (iv) 482.3 r.p.m., for both]

22.16 Solve Problem 22.15 assuming that the efficiency of the prototype turbine is 4% better than that of the model turbine.

[*Ans.* (i) 757.6 r.p.m.; (ii) 0.617 m^3/s; (iii) 31.96 kW; (iv) $(N_s)_p$ = 482.3 r.p.m.; $(N_s)_m$ = 456.1 r.p.m.]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Reciprocating Pumps

Chapter 23

23.1 INTRODUCTION

In general a pump may be defined as a mechanical device which when interposed in a pipeline, converts the mechanical energy supplied to it from some external source into hydraulic energy and transfers the same to the liquid through the pipeline, thereby increasing the energy of the flowing liquid. Almost all the pumps increase the pressure energy of the liquid which is subsequently converted into potential energy as the liquid is lifted from a lower level to a higher level. The various pumps may broadly be classified into the following two types:

1. Positive-displacement pumps.
2. Rotodynamic pumps (or dynamic-pressure pumps).

The 'positive displacement' pumps are those pumps in which the liquid is sucked and then it is actually pushed or displaced due to the thrust exerted on it by a moving member, which results in lifting the liquid to the required height. These pumps usually have one or more chambers which are alternately filled with the liquid to be pumped and then emptied again. As such the discharge of liquid pumped by these pumps almost wholly depends on the speed of the pump. The most common example of the positive displacement type of pumps is that of reciprocating pumps.

The 'rotodynamic pumps' have a rotating element, called impeller, through which as the liquid passes its angular momentum changes, due to which the pressure energy of the liquid is increased. As such a rotodynamic pump does not push the liquid as in the case of a positive displacement pump. The centrifugal pumps are the most common examples of rotodynamic pumps.

In this chapter only reciprocating pumps have been dealt with. The centrifugal pumps have been discussed in Chapter 24.

23.2 MAIN COMPONENTS AND WORKING OF A RECIPROCATING PUMP

A reciprocating pump essentially consists of a piston or plunger which moves to and fro in a close fitting cylinder, as shown in Fig. 23.1. The cylinder is connected to suction and delivery pipes, each of which is provided with a non-return or one way valve called suction valve and delivery valve respectively. The function of non-return or one way valve is to admit liquid in one direction only. Thus the suction valve allows the liquid only to enter the cylinder and the delivery valve permits only its discharge from the cylinder. The piston or the plunger is connected to a crank by means of a connecting rod. As the crank is rotated at uniform speed by a driving engine or motor, the piston or plunger moves

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

to and fro (or backward and forward) in the cylinder. When the crank rotates from $\theta = 0°$ to $\theta = 180°$, (see Fig. 23.1), the piston or plunger which is initially at its extreme left position (that is, it is completely inside the cylinder), move to its extreme right position, (that is, it moves outwardly in the cylinder). During the outward movement of the piston or plunger a partial vacuum (pressure below atmospheric) is created in the cylinder, which enables the atmospheric pressure acting on the liquid surface in the

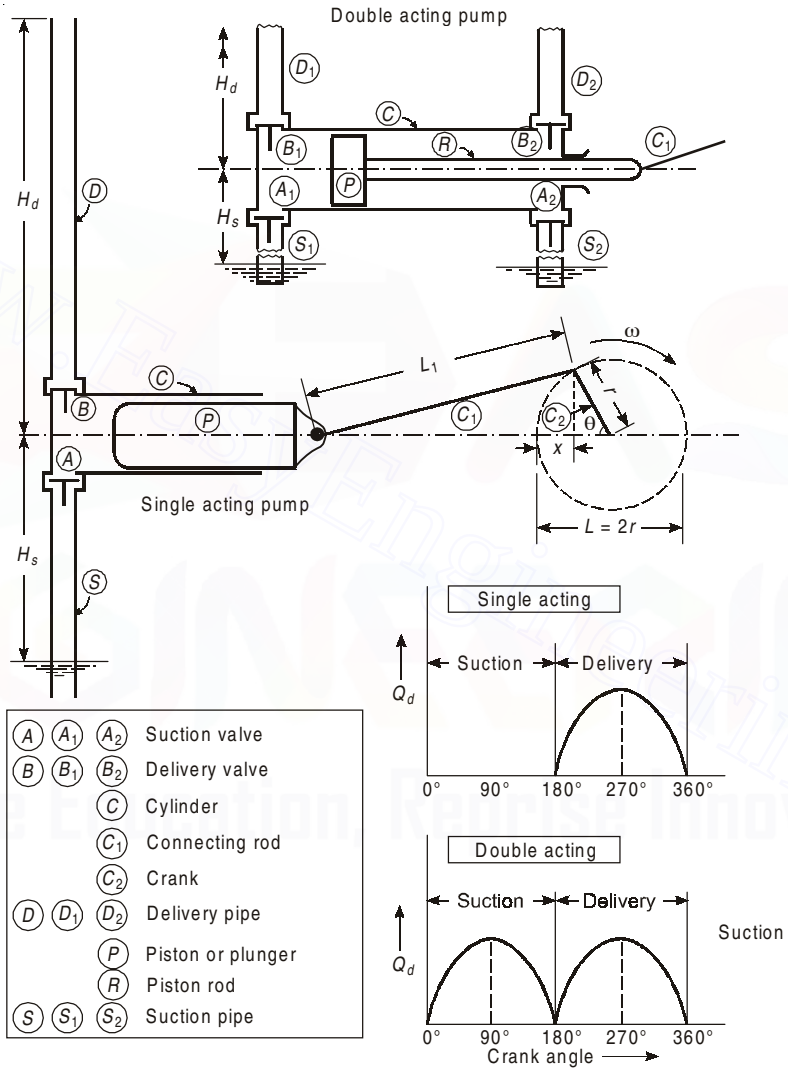


Figure 23.1 *Single acting and double acting reciprocating pumps*

well or sump below, to force the liquid up the suction pipe and fill the cylinder by forcing open the suction valve. Since during this operation of the pump the liquid is sucked from below it is known as its *suction stroke*. Thus at the end of the suction stroke the piston or plunger is at its extreme right position, the crank is at $\theta = 180°$ (i.e., at its outer dead centre), the cylinder is full of liquid, the suction valve is closed and the delivery valve is just at the point of opening.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When the crank rotates from θ = 180° to θ = 360° the piston or plunger moves inwardly from its extreme right position towards left. The inward movement of the piston or plunger causes the pressure of the liquid in the cylinder to rise above atmospheric, due to which the suction valve closes and the delivery valve opens. The liquid is then forced up the delivery pipe and raised to the required height. Since during this operation of the pump the liquid is actually delivered to the required height, it is known as its *delivery stroke*. At the end of the delivery stroke the piston or plunger is at extreme left position, the crank is at θ = 0° or 360° (i.e., at its inner dead centre) so that it has completed one full revolution, and both the suction and the delivery valves are closed. The same cycle is repeated as the crank rotates. This is in general the working principle of a reciprocating pump.

23.3 TYPES OF RECIPROCATING PUMPS

The reciprocating pumps can be classified according to the liquid being in contact with one side or both the sides of the piston or plunger; and according to the number of cylinders provided.

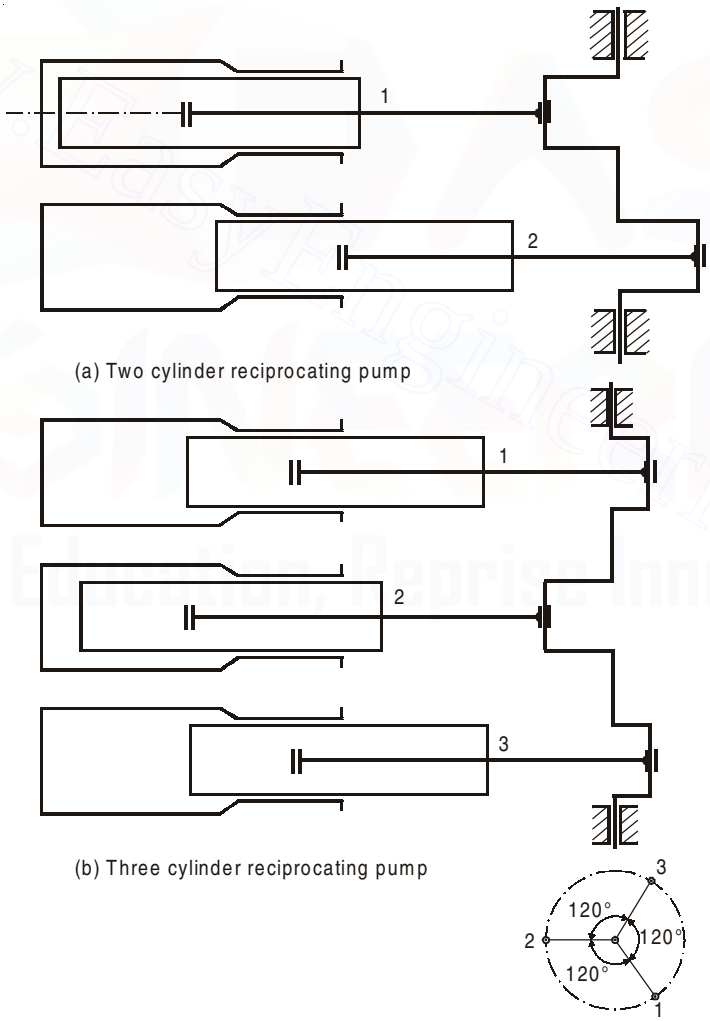


Figure 23.2 *Two and three cylinder reciprocating pumps*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

According to the first basis of classification the reciprocating pumps may be classified as:

(i) Single acting pump. (ii) Double acting pump.

If the liquid is in contact with one side of the piston or plunger only, it is known as *single acting pump*. Thus, as shown in Fig. 23.1 a single acting pump has one suction and one delivery pipe and in one complete revolution of crank there are only two strokes—one suction and one delivery stroke. On the other hand if the liquid is in contact with both the sides of the piston or plunger, it is known as *double acting pump*. As shown in Fig. 23.1 a double acting pump has two suction and two delivery pipes with appropriate valves, so that during each stroke when suction takes place on one side of the piston, the other side delivers the liquid. In this way in the case of a double acting pump in one complete revolution of the crank there are two suction strokes and two delivery strokes.

According to the number of cylinders provided the reciprocating pumps may be classified as:

(i) Single cylinder pump, (ii) Double cylinder pump, (iii) Triple cylinder pump,
(iv) Duplex double acting pump, (v) Quintuplex pump.

As the name suggests a reciprocating pump having only one cylinder is known as *single cylinder pump*. As shown in Fig. 23.1, a single cylinder pump may be either single acting or double acting.

A *double cylinder pump or two throw pump* is the one which has two single acting cylinders, each equipped with one suction and one delivery pipe with appropriate valves and a separate piston or plunger for each cylinder as shown in Fig. 23.2 (a). Both the pistons or plungers are simultaneously driven from cranks set at 180°. A double acting pump having two cylinders in line but only one

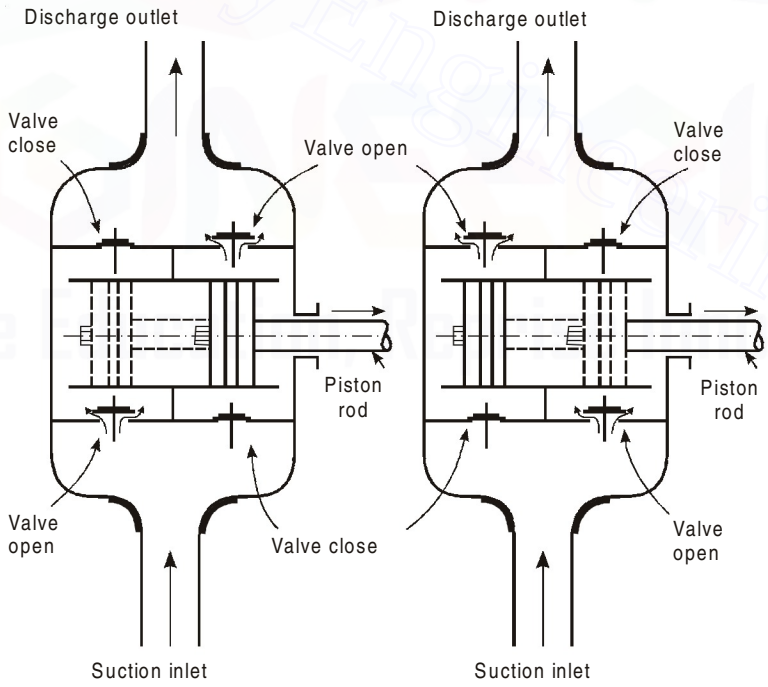


Figure 23.3 *Operation of valves for two throw reciprocating pump*

plunger driven from a single crank is also sometimes manufactured. Another alternative arrangement for a two throw pump may be as shown in Fig. 23.3, in which there is only one suction pipe, one

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

delivery pipe and one piston for both the cylinders. The operation of the valves for such a pump is also shown in Fig. 23.3. The working of a double cylinder pump is similar to that of a double acting single cylinder pump.

A *triple cylinder pump or three throw pump* is the one which has three single acting cylinders each equipped with one suction and one delivery pipe with appropriate valves and a separate piston or plunger as shown in Fig. 23.2 (b). All the three pistons or plungers are simultaneously driven from cranks set at 120°.

A *duplex double-acting pump* is formed by combining in line either two double-acting single cylinder pumps or two double-acting double-cylinder pumps, each driven by a crank and the two cranks being set at 90°. The combination is thus equivalent to a *four-throw pump*.

A *quintuplex or five throw pump* has five single acting cylinders driven from cranks set at 72°.

In general the reciprocating pumps having more than one cylinder are known as *multi-cylinder pumps*.

23.4 WORK DONE BY RECIPROCATING PUMP

The expression for the work done by different types of reciprocating pumps are obtained below.

1. Single-Acting Pump. As explained earlier a single-acting reciprocating pump has only one suction stroke and one delivery stroke for one complete revolution of the crank. Further the liquid is delivered by the pump only during the delivery stroke. Thus if the speed of the crank is N r.p.m., then the number of delivery strokes will be N per minute or $(N/60)$ per second. Also if A is the cross-sectional area of the piston or plunger and L is the length of the stroke, then the theoretical volume of liquid pumped per stroke = (AL); and the theoretical discharge Q_{th} of the pump is given by

$$Q_{th} = \frac{ALN}{60} \qquad ...(23.1)$$

As shown in Fig. 23.1, since $L = 2r$, where r is the radius of the crank, Eq. 23.1 may also be expressed as

$$Q_{th} = \frac{2ArN}{60} \qquad ...(23.2)$$

However, owing to some leakage and time lag in the closure of the valves the actual discharge Q_a of the pump is slightly different from the theoretical discharge Q_{th}, and in almost all the cases Q_a is less than Q_{th}.

Now if H_s is the height of the centre of the cylinder above the liquid surface in the sump below, and H_d is the height to which the liquid is raised above the centre of the cylinder, then H_s and H_d are termed as suction head and delivery head of the pump respectively and $(H_s + H_d)$ is known as its static head, which is the net total vertical height through which the liquid is lifted by the pump.

The theoretical work done by the pump

$$= wQ_{th}(H_s + H_d)$$

$$= \frac{w(ALN)(H_s + H_d)}{60} \qquad ...(23.3)$$

Hence the theoretical power P required to drive the pump is given by

$$P = wQ_{th}(H_s + H_d)$$

$$= \frac{w(ALN)(H_s + H_d)}{60} \qquad ...(23.4)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where w is the specific weight of the liquid.

However, on account of frictional resistance and leakage the actual power required to drive the pump will be more than that given by Eq. 23.4.

The force acting on the piston or plunger during the suction stroke = (wH_sA). ...(23.5)

The force acting on the piston or plunger during the delivery stroke = (wH_dA). ...(23.6)

2. Double Acting Pump. In the case of a double acting reciprocating pump, when there is a suction stroke on one side of the piston or plunger, its other side has a delivery stroke. Thus for one complete revolution of the crank there are two delivery strokes, during which the liquid is delivered by the pump. Now if the speed of the crank is N r.p.m., then the number of delivery strokes will be $2N$ per minute or $(N/30)$ per second. However, due to the presence of the piston rod on one side, the volume of liquid delivered from both the sides of the piston or plunger will not be equal. As such if A_p is the area of cross-section of piston rod then the theoretical volume of liquid pumped per stroke

$$= [AL + (A - A_p)L];$$

and the theoretical discharge Q_{th} of the pump is given by

$$Q_{\text{th}} = \left[\frac{ALN}{60} + \frac{(A - A_p)LN}{60}\right] = \frac{(2A - A_p)LN}{60} \quad ...(23.7)$$

However, if the effect of A_p is neglected then the theoretical discharge Q_{th} is obtained as

$$Q_{\text{th}} = \frac{2ALN}{60} \quad ...(23.8)$$

which is equal to twice the theoretical discharge of a single acting pump of same dimensions and same speed.

Again for these pumps also, due to leakage and the imperfect operation of the valves, the actual discharge Q_a of the pump slightly differs from Q_{th} and in almost all the cases Q_a is less than Q_{th}.

The theoretical work done by the pump

$$= wQ_{\text{th}}(H_s + H_d) = \frac{w(2A - A_p)LN(H_s + H_d)}{60} \quad ...(23.9)$$

$$\approx \frac{2wALN(H_s + H_d)}{60}$$

Hence the theoretical power P required to drive the pump is given by

$$P = wQ_{\text{th}}(H_s + H_d) = \frac{w(2A - A_p)LN(H_s + H_d)}{60} \quad ...(23.10)$$

$$\approx \frac{2w(ALN)(H_s + H_a)}{60}$$

Again on account of frictional resistance and leakage the actual power required to drive the pump will be more than that given by Eq. 23.10.

The force acting on the piston or plunger during outward (left to right) stroke

$$= [wH_sA + wH_d(A - A_p)]. \quad ...(23.11)$$

The force acting on the piston or plunger during inward (right to left) stroke

$$= [wH_dA + wH_s(A - A_p)]. \quad ...(23.12)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

23.5 COEFFICIENT OF DISCHARGE, SLIP, PERCENTAGE SLIP AND NEGATIVE SLIP OF RECIPROCATING PUMP

1. Coefficient of Discharge. As explained in section 23.4, due to leakage and the imperfect operation of the valve the actual discharge Q_a of a reciprocating pump is always slightly different from the theoretical discharge Q_{th}. The ratio between the actual discharge Q_a and the theoretical discharge Q_{th} is defined as the *coefficient of discharge* C_d of the pump. That is

$$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} = \frac{Q_a}{Q_{th}} \qquad ...(23.13)$$

The value of C_d when expressed in terms of percentage is also known as *volumetric efficiency* of the pump.

2. Slip and Percentage Slip. The difference between the theoretical discharge and the actual discharge is known as the *slip* of the pump. That is

$$\text{Slip} = (Q_{th} - Q_a) \qquad ...(23.14)$$

Often the slip is expressed in terms of percentage which is then called *percentage slip* and it is given by

$$\text{Percentage slip} = \frac{Q_{th} - Q_a}{Q_{th}} \times 100 = (1 - C_d) \times 100 \qquad ...(23.15)$$

For the pumps maintained in good condition the percentage slip is of the order of 2 per cent or even less.

3. Negative Slip. For most of the reciprocating pumps the actual discharge Q_a is less than the theoretical discharge Q_{th}, in which case C_d of the pump is less than one and the slip of the pump is positive. However, in some cases the actual discharge of the pump may be more than the theoretical discharge, in which case C_d will be more than one and the slip will be negative, which is then known as *negative slip.*

This may happen in the case of pumps having long suction pipe and low delivery head, especially when these are running at high speed. This is so because for such pumps the inertia pressure in the suction pipe will be large in comparison to the pressure on the outside of the delivery valve, which may cause the delivery valve to open before the suction stroke is completed. Some liquid is thus pushed directly into the delivery pipe even before the delivery stroke is commenced, which results in making the actual discharge more than the theoretical discharge.

23.6 EFFECT OF ACCELERATION OF PISTON ON VELOCITY AND PRESSURE IN THE SUCTION AND DELIVERY PIPES

During the reciprocating motion of the piston or plunger it does not move with uniform velocity for the entire stroke, but as indicated later it has an acceleration developed for the first half of each stroke and for the later half of the stroke there is retardation developed for it. Since the liquid flowing through the pump closely follows the piston or plunger, its acceleration and retarding effects will be correspondingly transmitted to the liquid flowing in the suction and delivery pipes. In other words the velocity of flow of liquid in the suction and delivery pipes will not be uniform but during the same stroke the velocity of flow in these pipes will vary with the crank position. These variations in the velocities of flow of liquid in the suction and delivery pipes give rise to inertia pressures which will cause a variation of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

pressure in the cylinder. The magnitudes of the inertia pressures so developed can be calculated as indicated below.

If the connecting rod is very long as compared with the length of the crank, then it can be assumed that the piston or plunger moves with simple harmonic motion. The assumption considerably simplifies the problem. Let the crank be rotating with an angular velocity ω radians per second and in time t seconds let it turn through an angle θ, from its inner dead centre. Then

$$\theta = \omega t = \frac{2\pi N t}{60}$$

If x is the linear displacement of the piston from the end of the stroke in t seconds then from Fig. 23.1, we have

$$x = r - r\cos\theta = r - r\cos(\omega t)$$

Further if v is the velocity of the piston and f is the acceleration of the piston then, we have

$$v = \frac{dx}{dt} = \omega r \sin(\omega t) = \omega r \sin\theta$$

and

$$f = \frac{dv}{dt} = \omega^2 r \cos(\omega t) = \omega^2 r \cos\theta$$

Now if A represents the cross-sectional area of the piston or plunger and a is the area of the pipe (either suction or delivery pipe), then since the volume of liquid flowing from the pipe per second equals the volume of liquid flowing into the cylinder per second, then the velocity V of flow of liquid in the pipe is given by

$$V = \frac{A}{a}v = \frac{A}{a}\omega r \sin(\omega t) = \frac{A}{a}\omega r \sin\theta \qquad \ldots(23.16)$$

Similarly the acceleration of the liquid in the pipe is given by acceleration $= f\frac{A}{a} = \frac{A}{a}(\omega^2 r)\cos\theta$.

Now if l is the length of the pipe, then the weight of the liquid which is subjected to the above noted acceleration is (wal) and the corresponding mass of the liquid is $\left(\frac{wal}{g}\right)$.

Further if p_a is the intensity of pressure due to acceleration of liquid in the pipe then the accelerating force = $(p_a)\,a$.

But according to Newton's second law of motion

Accelerating force = mass × acceleration

Thus

$$(p_a)\,a = \frac{wal}{g} \times \frac{A}{a}(\omega^2 r)\cos\theta$$

or

$$p_a = \frac{wl}{g} \times \frac{A}{a}(\omega^2 r)\cos\theta \qquad \ldots(23.17)$$

From Eq. 23.17 the pressure head H_a due to acceleration may be obtained as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_a = \frac{p_a}{w} = \frac{l}{g}\frac{A}{a}(\omega^2 r)\cos\theta \quad \ldots(23.18)$$

It may thus be seen from Eq. 23.18 that the pressure head developed due to acceleration acting on the piston or plunger will vary with angle θ.

At the beginning of the stroke when θ = 0°, cos θ = 1, i.e., when piston or plunger is at its inner dead centre, then

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r) \quad \ldots(23.19)$$

At the middle of the stroke when θ = 90°, cos θ = 0, then

$$H_a = 0 \quad \ldots(23.20)$$

At the end of the stroke when θ = 180°, cos θ = –1, i.e., when piston of plunger is at its outer dead centre, then

$$H_a = -\frac{l}{g}\frac{A}{a}(\omega^2 r) \quad \ldots(23.21)$$

Further for 0° < θ < 90°, H_a has positive values and for 90° < θ < 180°, H_a has negative values, thereby indicating that for the first half of the stroke there is acceleration head developed and for the later half of the stroke there is retardation head developed.

The above derived equations for the pressure head H_a may be applied to both the suction and delivery pipes by using either the length l_s and area a_s of the suction pipe; or length l_d and area a_d of the delivery pipe.

However, if the connecting rod is not very long as compared with the length of the crank then simple harmonic motion cannot be assumed for the piston. The pressure head H_a is then given by

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r)\cos\theta\left(\cos\theta + \frac{\cos 2\theta}{n}\right) \quad \ldots(23.22)$$

when n is the ratio of the length of connecting rod to the length of crank.

From Eq. 23.22 at the beginning of the stroke

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r)\left(1 + \frac{1}{n}\right) \quad \ldots(23.23)$$

and at the end of the stroke

$$H_a = -\frac{l}{g}\frac{A}{a}(\omega^2 r)\left(1 - \frac{1}{n}\right) \quad \ldots(23.24)$$

(a) Effect of Variation of Velocity on Friction in Pipes. There will be frictional resistance offered to the liquid flowing through suction and delivery pipes which will cause the loss of head. The loss of head due to friction is given by Darcy-Weisbach equation as

$$h_f = f\frac{l}{d}\frac{V^2}{2g}$$

where f is friction factor, and d is diameter of the pipe.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

By substituting for the instantaneous velocity V of flow in the pipe at any point of the stroke from Eq. 23.16 the head loss due to friction becomes

$$h_f = \frac{fl}{2gd}\left(\frac{A}{a}\omega r \sin\theta\right)^2 \qquad ...(23.25)$$

From Eq. 23.25 it is obtained that due to the fluctuation in the velocity of flow in pipes the loss of head due to friction also varies with the crank position. Thus at the begnning of the stroke, $\theta = 0°$, $\sin\theta = 0$ and hence $h_f = 0$. Similarly at the end of the stroke, $\theta = 180°$, $\sin\theta = 0$ and hence $h_f = 0$. At the middle of the stroke, $\theta = 90°$, $\sin\theta = 1$ and hence h_f has its maximum value as

$$(h_f)_{max} = \frac{fl}{2gd}\left(\frac{A}{a}\omega r\right)^2 \qquad ...(23.26)$$

Further it may be seen from Eq. 23.25 that the variation of h_f with θ is parabolic. Again Eq. 23.25 is applicable to both the suction and delivery pipes and hence by using l_s, d_s, a_s for the suction pipe and l_d d_d, a_d for the delivery pipe the corresponding values of h_f for each of these pipes may be obtained.

(b) Instantaneous and Mean Rates of Discharge. The instantaneous velocity V_d in the delivery pipe may be obtained from Eq. 23.16 by writing subscript d as

$$V_d = \frac{A}{a_d}(\omega r \sin\theta)$$

$$= \frac{\left(\frac{\pi}{4}\right)D^2}{\left(\frac{\pi}{4}\right)d_d^2}(\omega r \sin\theta)$$

or

$$V_d = \left(\frac{D}{d_d}\right)^2(\omega r \sin\theta)$$

$$= \left(\frac{D}{d_d}\right)^2\left(\frac{2\pi Nr}{60}\sin\theta\right) \qquad ...(23.27)$$

where D is the diameter of piston or plunger.

From Eq. 23.27 a curve between V_d and θ can be plotted which will be a sine curve.

For a single acting pump since for one complete revolution of the crank there is only one delivery stroke during which the liquid is delivered, the mean velocity in the delivery pipe can be obtained by integrating Eq. 23.27 as follows

$$(V_d)_{mean} = \frac{1}{2\pi}\left[\int_0^{\pi}\left(\frac{D}{d_d}\right)^2(\omega r)\sin\theta\, d\theta\right]$$

$$= \left(\frac{D}{d_d}\right)^2\frac{\omega r}{\pi} \qquad ...(23.28)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

However, the above expression for the mean velocity may also be obtained by dividing the theoretical discharge Q_{th} of the pump given by Eq. 23.2, by the area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

From Eq. 23.27 when $\theta = 90°$, $\sin\theta = 1$; the velocity V_d has a maximum value given by

$$(V_d)_{\max} = \left(\frac{D}{d_d}\right)^2 \omega r \qquad \ldots(23.29)$$

By dividing Eq. 23.28 by Eq. 23.29 we obtain the ratio between the mean velocity and maximum velocity in the delivery pipe as

$$\frac{(V_d)_{\text{mean}}}{(V_d)_{\max}} = \frac{\left(\frac{D}{d_d}\right)^2 \left(\frac{\omega r}{\pi}\right)}{\left(\frac{D}{d_d}\right)^2 (\omega r)} = \frac{1}{\pi} \qquad \ldots(23.30)$$

The instantaneous rate of discharge Q_d in the delivery pipe may be obtained by using Eq. 23.27 as

$$Q_d = (V_d\, a_d) = A\, \omega r \sin\theta \qquad \ldots(23.31)$$

From Eq. 23.31 a plot of Q_d versus θ can be obtained, which will be a sine curve as shown in Fig. 23.1. For a single acting pump during the first half revolution of the crank i.e., for $\theta = 0°$ to $180°$, there is only suction and no delivery and during the second half revolution of the crank i.e., for $\theta = 180°$ to $360°$ there is delivery of liquid. The same cycle is repeated afterwards. Thus the part of the sine curve below the axis will represent suction and that above it will represent delivery. However, in Fig. 23.1 only that part of the curve is shown which represents the delivery of the liquid.

The mean discharge $(Q_d)_{\text{mean}}$ for a single acting pump can be obtained by integrating Eq. 23.31 as follows:

$$(Q_d)_{\text{mean}} = \frac{1}{2\pi}\left[\int_0^{\pi} A\omega r \sin\theta d\theta\right] = \frac{A\omega r}{\pi} \qquad \ldots(23.32)$$

The above expression for the mean discharge may also be obtained by multiplying $(V_d)_{\text{mean}}$ given by Eq. 23.28 by the area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

Again from Eq. 23.31 for $\theta = 90°$, $\sin\theta = 1$, the discharge Q_d has a maximum value given by

$$(Q_d)_{\max} = A\omega r \qquad \ldots(23.33)$$

Further by dividing Eq. 23.32 by Eq. 23.33, we obtain

$$\frac{(Q_d)_{\text{mean}}}{(Q_d)_{\max}} = \frac{1}{\pi} \qquad \ldots(23.34)$$

For a double acting pump since there are two delivery strokes, for one complete revolution of the crank, the mean velocity of flow of liquid in the delivery pipe may be obtained by integrating Eq. 23.27 as follows

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(V_d)_{\text{mean}} = 2 \times \frac{1}{2\pi}\left[\int_0^{\pi} \left(\frac{D}{d_d}\right)^2 (\omega r)\sin\theta d\theta\right]$$

or
$$(V_d)_{\text{mean}} = 2 \times \left(\frac{D}{d_d}\right)^2 \frac{\omega r}{\pi} \quad ...(23.35)$$

Again the above expression for the mean velocity may also be obtained by dividing the theoretical discharge Q_{th} given by Eq. 23.8 by area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

By dividing Eq. 23.35 by Eq. 23.29 we obtain for a double acting pump

$$\frac{(V_d)_{\text{mean}}}{(V_d)_{\text{max}}} = \frac{2}{\pi} \quad ...(23.36)$$

The instantaneous rate of discharge Q_d for a double acting pump is also given by Eq. 23.31. As such in this case also a plot of Q_d versus θ will be a sine curve. But for a double acting pump during one complete revolution of the crank i.e., for θ = 0° to 360°, there being two delivery strokes, the Q_d versus θ will be a resultant of two sine curves drawn at a phase difference of 180°, as shown in Fig. 23.1, in which only the curves corresponding to the delivery of the liquid are shown. Thus the mean discharge $(Q_d)_{\text{mean}}$ for a double acting pump can also be obtained by integrating Eq. 23.31 as follows:

$$(Q_d)_{\text{mean}} = 2 \times \frac{1}{2\pi}\left[\int_0^{\pi} A\omega r \sin\theta d\theta\right]$$

$$= \frac{2A\omega r}{\pi} \quad ...(23.37)$$

However the above expression for the mean discharge may also be obtained by multiplying $(V_d)_{\text{mean}}$ given by Eq. 23.35 by the area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

Again by dividing Eq. 23.37 by Eq. 23.33 we obtain for double acting pump

$$\frac{(Q_d)_{\text{mean}}}{(Q_d)_{\text{max}}} = \frac{2}{\pi} \quad ...(23.38)$$

However, if the area of piston rod is taken into account then it can be shown that the instantaneous discharges Q_{d1} and Q_{d2} for the two delivery pipes on either side of piston or plunger will be

$$\left.\begin{aligned} Q_{d_1} &= A\omega r \quad \sin\theta \\ Q_{d_2} &= (A - A_p)\omega r[\sin(\pi - \theta)] \end{aligned}\right\} \quad ...(23.39)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

23.7 INDICATOR DIAGRAMS

1. Theoretical Indicator Diagram. Figure 23.4 shows the theoretical indicator diagram of a reciprocating pump which represents the work done by a single acting pump during one complete cycle as given by Eq. 23.3. In this diagram the pressure head on the piston or plunger is plotted along the vertical ordinate and the length of the stroke is represented by the abscissa. The horizontal line *fe* represents the atmospheric pressure. The line *dc* represents the pressure head in the cylinder during the suction stroke, which is below atmospheric pressure by an amount equal to suction head H_s. The line *ab* represents the pressure in the cylinder during the delivery stroke, which is above atmospheric pressure by an amount equal to delivery head H_d. The area *dcef* represents to some scale the work done by the piston or plunger during the suction stroke and the area *abef* represents to the same scale the work done by the piston or plunger during the delivery stroke. The total work done by the piston or plunger during the complete revolution of the crank is then represented to the same scale by area *abcd* which is equal to $(H_s + H_d)L$.

If the pump is double acting, this diagram will represent the pressure head on one side of the piston or plunger only. The work done per revolution will then be represented by twice the area of this diagram, if the area of the piston rod is neglected.

Such a diagram may be obtained automatically by means of an indicator placed on the cylinder and hence it is called an indicator diagram.

2. Effect of Acceleration in Suction Pipe on Indicator Diagram. Due to the acceleration head developed in the suction pipe during the suction stroke, the theoretical indicator diagram will be modified as indicated below. If l_s, d_s, a_s are respectively the length, diameter and the cross-sectional area of the suction pipe, then the acceleration head H_{as} in the pipe is given by Eq. 23.18 as

$$H_{as} = \frac{l_s}{g}\frac{A}{a_s} w^2 r \cos\theta \qquad \ldots(23.40)$$

At the beginning of the suction stroke since $\theta = 0°$, $\cos\theta = 1$, the acceleration head is

$$H_{as} = +\frac{l_s}{g} \times \frac{A}{a_s} \omega^2 r \qquad \ldots(23.41)$$

From Eq. 23.41 it is seen that at the beginning of the suction stroke since H_{as} is positive the liquid in the suction pipe has to be accelerated, for which an additional drop in pressure in the cylinder is required. As a result the pressure head in the cylinder will further drop by an amount equal to H_{as}, so that at the beginning of the suction stroke the pressure head will be $(H_s + H_{as})$ below the atmospheric pressure head.

At the middle of the suction stroke since $\theta = 90°$, $\cos\theta = 0$, the acceleration head $H_{as} = 0$. Thus at the middle of the suction stroke the pressure head will be only H_s below the atmospheric pressure head.

At the end of the suction stroke since $\theta = 180°$, $\cos\theta = -1$, the acceleration head is

$$H_{as} = -\frac{l_s}{g} \times \frac{A}{a_s} w^2 r \qquad \ldots(23.42)$$

From Eq. 23.42 it is seen that at the end of the suction stroke the liquid is to be retarded, corresponding to which a pressure rise in the cylinder is needed. As a result the pressure head in the cylinder will rise by an amount H_{as}, so that at the end of the suction stroke the pressure head will be $(H_a - H_{as})$ below the atmospheric pressure head.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

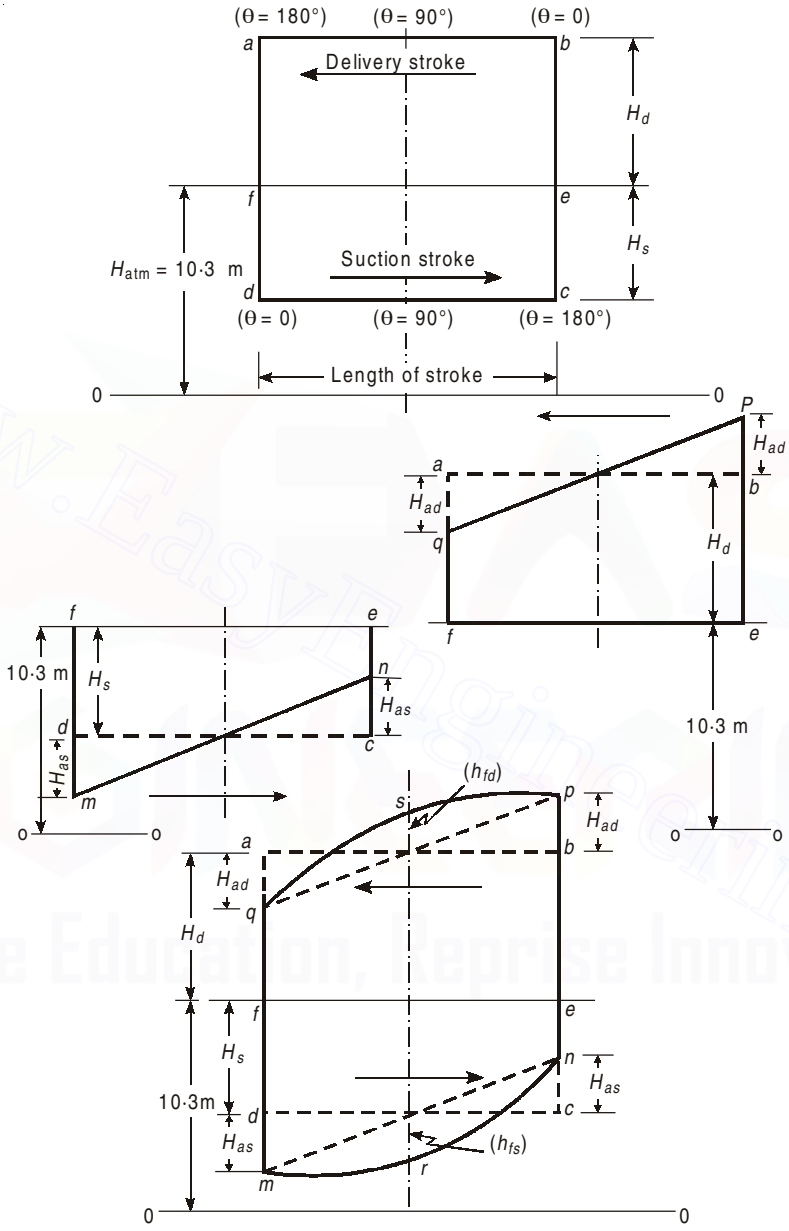


Figure 23.4 *Indicator diagram*

The indicator diagram for the suction stroke will thus be modified as shown in Fig. 23.4. The base of the diagram is now changed from *dc* to *mn*. Also the work done during the suction stroke is now represented by the area *mnef*. However, area *mnef* being equal to the area *dcef*, the net work done during the suction stroke is not altered on account of the accelerating effects in the suction pipe. It is thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

observed that the inertia pressure developed in the suction pipe does not affect the net work done, but merely causes a variation of pressure in the cylinder.

3. Maximum Speed of a Reciprocating Pump. The speed of a reciprocating pump is usually restricted by the pressure corresponding to the point *m* of the indicator diagram. This is so because higher is the speed, the greater will be the acceleration head and lower will be the pressure in the cylinder at the beginning of the suction stroke. The pressure in the cylinder must not be allowed to fall to the value at which dissolved gases are liberated from the liquid, since under these conditions the cavitation may occur and the continuity of the flow may not exist. For water the value of this limiting pressure is about 2.6 m of water absolute or if atmospheric pressure is 10.3 m of water ($H_s + H_{as}$) at any point during the suction stroke must not be greater than (10.3 – 2.6) = 7.7 m of water. The maximum permissible speed of a reciprocating pump may thus be computed from the following expression

$$H_{sep} = H_s + H_{as}$$

or
$$H_{sep} = H_s + \frac{l_s}{g} \times \frac{A}{a_s} w^2 r$$

or
$$H_{sep} = H_s + \frac{l_s}{g} \frac{A}{a_s} \left(\frac{2\pi N}{60} \right)^2 r \qquad ...(23.43)$$

where H_{sep} represents the pressure head below atmospheric pressure head at which the separation of dissolved gases and cavitation may occur.

It may, however, be seen from Eq. 23.43 that the drop of pressure in the cylinder during the beginning of the suction stroke may also be limited by suitably adjusting H_s, l_s and (A/a_s) for the pump.

4. Effect of Acceleration in Delivery Pipe on Indicator Diagram. As in the case of suction pipe the acceleration head will also be developed for the liquid flowing in the delivery pipe, due to which the indicator diagram will be modified. If l_d, d_d and a_d are respectively the length, diameter and the cross-sectional area of the delivery pipe then the acceleration head in the delivery pipe is given by Eq. 23.18 as

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \omega^2 r \cos\theta \qquad ...(23.44)$$

At the beginning of the delivery stroke $\theta = 0°$, $\cos\theta = 1$, the acceleration head is

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \omega^2 r \qquad ...(23.45)$$

Again at the beginning of the delivery stroke since H_{ad} is positive, the liquid in the delivery pipe is to be accelerated, for which an additional pressure head is required to be developed in the cylinder. As a result at the beginning of the delivery stroke the pressure head in the cylinder will be $(H_d + H_{ad})$ above the atmospheric pressure head.

At the middle of the delivery stroke since $\theta = 90°$, $\cos\theta = 0$, the acceleration head $H_{ad} = 0$. Thus at this point of the delivery stroke the pressure head in the cylinder will be only H_d above the atmospheric pressure head.

At the end of the delivery stroke since $\theta = 180°$, $\cos\theta = -1$, the acceleration head is

$$H_{ad} = -\frac{l_d}{g} \times \frac{A}{a_d} \omega^2 r \qquad ...(23.46)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus at the end of the delivery stroke the liquid is to be retarded, corresponding to which a drop of pressure in the cylinder is needed. As a result the pressure head in the cylinder will drop by an amount H_{ad}, so that at the end of the delivery stroke the pressure head will be $(H_d - H_{ad})$ above the atmospheric pressure head.

The indicator diagram for the delivery stroke will thus be modified as shown in Fig. 23.4. The top of the diagram is now changed from *ab* to *pq*. Again the work done during the delivery stroke is now represented by the area *pqfe* which is however equal to the area *abef*. As such during the delivery stroke also the net work done is not altered on account of the accelerating effects in the delivery pipe, but these merely cause a variation of pressure in the cylinder.

During the delivery stroke the minimum pressure head in the cylinder is at the end of the stroke, which is represented by the pressure head at point *q* of the indicator diagram and it is equal to $(H_{atm} + H_d - H_{ad})$ absolute. Again this pressure head must not be less than the absolute separation pressure head of the liquid, otherwise cavitation may occur at the end of the delivery stroke. For water the separation pressure head is about 2.6 m of water absolute, and hence in the limiting condition

$$10.3 + H_d - H_{ad} = 2.6$$

or

$$H_{ad} = 7.7 + H_d \qquad \ldots(23.47)$$

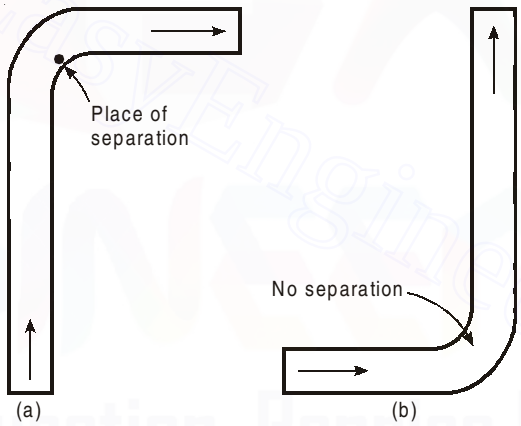


Figure 23.5 *Different arrangements of delivery pipe of a reciprocating pump*

The delivery pipe of a reciprocating pump may have either of the two arrangements as shown in Fig. 23.5. In the arrangement shown in Fig. 23.5(a) the delivery pipe is first vertical and then horizontal. In this case the delivery head H_d will become zero at the bend, after which there is still a long horizontal pipe which will have a considerable value of accelerating head H_{ad}.. Therefore according to Eq. 23.47 the separation may take place at the bend in this case. On the other hand in the arrangement shown in Fig. 23.5 (b) the pipe is horizontal first and then it is vertical. As such there is still considerable H_d available at the bend in this case. Therefore there is no possibility of separation to occur at the bend in this case.

5. Effect of Friction in Suction and Delivery Pipes on Indicator Diagram. A further modification of the indicator diagram results from the effect of friction losses in the suction and delivery pipes. The loss of head due to friction h_f in these pipes is given by Eq. 23.25, which shows that the variation of h_f with θ is parabolic. At the beginning and the end of the suction as well as delivery strokes since $\theta = 0^\circ$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

and $\theta = 180°$, $\sin\theta = 0$, the head loss due to friction h_{fs} and h_{fd} in the suction and delivery pipes respectively will be equal to zero. At the middle of the stroke since $\theta = 90°$, $\sin\theta = 1$, the head loss due to friction h_{fs} and h_{fd} will be

$$h_{fs} = \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r\right)^2 \qquad ...(23.48)$$

and

$$h_{fd} = \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r\right)^2 \qquad ...(23.49)$$

Now if the effect of the loss of head due to friction in the suction pipe is considered then as shown in Fig. 23.4 the base of the diagram will change from *mn* to *mrn*, the parabola *mrn* being the work done against friction in the suction pipe. Similarly by considering the effect of the loss of head due to friction in the delivery pipe the top of the diagram will change from *pq* to *psq* as shown in Fig. 23.4, the parabola *psq* being the work done against friction in the delivery pipe.

Thus the total work done during suction stroke is represented by area (*efmrn*) which is equal to [area (*efdc*) + area (*mrn*)]. Similarly the total work done during delivery stroke is represented by area (*efqsp*) which is equal to [area (*abef*) + area (*psq*)].

As the mean ordinate of a parabola equals two-thirds of its maximum ordinate the total area of the indicator diagram *mrnpsq* is equal to

$$L \left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right]$$

which to some scale represents the total work done during one complete revolution for a single acting pump. The total work done per second for a single acting pump is given as

$$\text{Work done} = \frac{wALN}{60}\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right] \qquad ...(23.50)$$

The absolute pressure head on the piston or plunger during suction stroke for any position of the crank i.e., for any crank angle θ may now be expressed as

$$H_{ts} = H_{\text{atm}} - H_s - H_{as} - h_{fs}$$

or

$$H_{ts} = H_{\text{atm}} - H_s - \left(\frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r\cos\theta\right)$$

$$- \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r\sin\theta\right)^2 \qquad ...(23.51)$$

From Eq. 23.51 it can be shown that the minimum value of the absolute pressure H_{ts} will occur for $\theta = 0°$ i.e., at the beginning of the suction stroke. Therefore if cavitation occurs during the suction stroke, it will do so at the beginning of the stroke, at the junction of suction pipe with the cylinder.

Again the absolute pressure head on the piston or plunger during delivery stroke for any position of the crank i.e., for any crank angle θ may be expressed as

$$H_{td} = H_{\text{atm}} + H_d + H_{ad} + h_{fd}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$H_{td} = H_{\text{atm}} + H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right)$$

$$+ \frac{fl_d}{2gd_d}\left(\frac{A}{a_d} \times \omega r \sin\theta\right)^2 \quad ...(23.52)$$

From Eqs 23.51 and 23.52 it is evident that the acceleration heads H_{as} and H_{ad} have maximum values at the ends of the stroke i.e., when $\theta = 0°$ and $\theta = 180°$, and are equal to zero at the middle of the stroke i.e., when $\theta = 90°$. On the other hand the frictional head losses h_{fs} and h_{fd} are equal to zero at the ends of the stroke and have a maximum value at the middle of the stroke.

23.8 AIR VESSELS

For a single acting pump, the flow rate in the delivery pipe at any instant varies considerably, having a value of zero during the whole of the suction stroke and rising to a maximum during the delivery stroke. In other words the supply of liquid by a single-acting pump will be quite intermittent. To some extent this difficulty may be overcome if the pump is double acting or if it is a multi-cylinder pump so that a continuous supply of liquid may be obtained. But in the case of almost all the pumps the velocity of flow in both suction and delivery pipes vary with the crank position, which results in developing a non-uniform rate of flow of liquid in these pipes. As such in order to obtain a continuous supply of liquid at a uniform rate from a single-acting pump and also to obtain a uniform rate of flow of liquid in both suction and delivery pipes of a reciprocating pump, a large air vessel is fitted to the suction as well as the delivery pipe at a point close to the cylinder. An *air vessel* is a close chamber having an opening at its base through which the liquid may flow into the vessel or it may flow out from the vessel. The top portion of the air-vessel contains compressed air, which will be further compressed when the liquid enters the vessel and it will expand when the liquid flows out from the vessel. Figure 23.6 shows air-vessels fitted to suction and delivery pipes of a single-acting pump, the working of each of these is explained below.

Consider an air vessel fitted to the suction pipe of the pump. During the first half of the suction stroke the piston or plunger moves with acceleration and hence the liquid in the suction pipe has also to be accelerated. In other words the liquid in the suction pipe must have a velocity of flow more than the mean velocity, so that the corresponding flow rate of liquid entering the cylinder may be more than the mean discharge. This excess quantity of liquid required on account of accelerating effects will however be supplied from the air vessel, so that the velocity of flow of liquid in the portion of the suction pipe below the point at which the air vessel is connected to it will be equal to the mean velocity of flow only. During the second half of the suction stroke the piston or plunger moves with retardation and hence the liquid in the suction pipe is also required to be retarded. In other words the liquid in the suction pipe must have a velocity of flow less than the mean velocity, so that the corresponding flow rate of liquid entering the cylinder is less than the mean discharge. Since the velocity of flow of liquid in the portion of the suction pipe below the point at which the air vessel is connected to it is equal to the mean velocity of flow, the excess quantity of liquid flowing in the suction pipe will flow into the air-vessel thus compressing the air inside the vessel. In this way a relatively small quantity of liquid will only be entering the cylinder from the suction pipe. The liquid thus stored in the air-vessel will be supplied during the first half of the next suction stroke and the same cycle will thus be repeated.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From the foregoing discussion it is thus clear that by fitting an air vessel to the suction pipe, throughout the suction stroke a uniform velocity of flow occurs in the portion of the suction pipe below the point at which the air vessel is connected and the fluctuations in the velocity of flow due to the accelerating effects will be limited only to the portion of the suction pipe between the cylinder and the

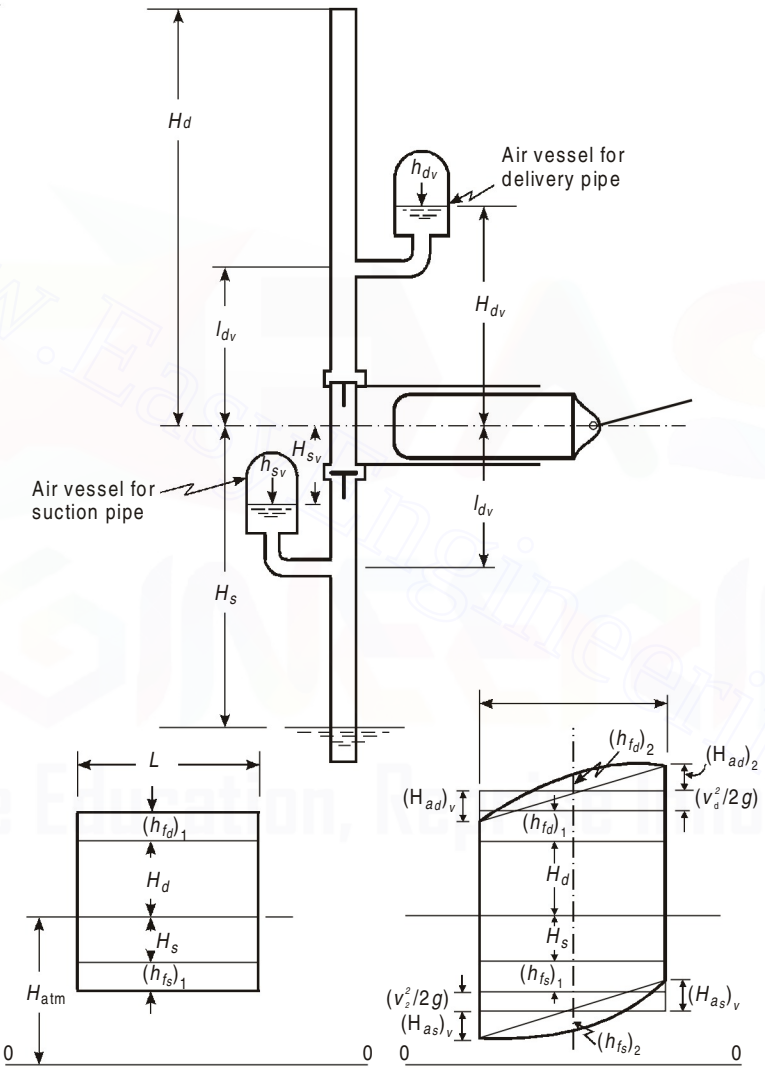


Figure 23.6 *Reciprocating pump with air vessels*

point at which the air vessel is connected to the suction pipe. Since the velocity fluctuations are eliminated for most of the length of the suction pipe, the acceleration pressure heads will also be eliminated and only very small acceleration pressure heads corresponding to the length of the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

suction pipe between the cylinder and the point at which the air vessel is connected to it, will be developed. As such the acceleration pressure heads may be considerably reduced by fitting the air vessel to the suction pipe at a point very close to the cylinder.

Now consider an air vessel fitted to the delivery pipe of the pump. Again during the first half of the delivery stroke the piston or plunger moves with acceleration and hence the liquid is forced into the delivery pipe at a rate of flow greater than the mean discharge. The quantity of liquid in excess of the mean discharge will, however, flow into the air vessel and stored there. During the second half of the delivery stroke, since the piston or plunger moves with retardation, the liquid is forced in the delivery pipe at a rate of flow less than the mean discharge. The liquid stored in the air vessel will then flow into the delivery pipe, thereby making up the deficiency. Moreover, in the case of a single acting pump during the next suction stroke, when the pump discharge is zero, the flow in the delivery pipe will be maintained by the liquid supplied by the air vessel which has been stored in it during the previous delivery stroke.

Thus, by fitting an air vessel to the delivery pipe a uniform rate of flow of liquid will be obtained in the delivery pipe beyond the point at which the air vessel is fitted to it, on account of which the acceleration pressure heads will be eliminated. However, in the portion of the delivery pipe between the cylinder and the point at which the air vessel is fitted to it, the velocity of flow will be fluctuating due to the accelerating effects, corresponding to which small acceleration pressure heads will be developed. Again these acceleration heads may also be considerably reduced by fitting the air vessel to the delivery pipe at a point very close to the cylinder.

As the liquid surface in the air vessel rises and falls some change in the air pressure will take place, which will result in a corresponding variation of the pressure and velocity in the suction and delivery pipes. Such variations of pressure and velocity in these pipes will cause slight variation in the rate of flow of liquid. However, these variations can be considerably reduced by making the air vessels sufficiently large. For a single acting pump it has been observed that an air vessel of volume about six to nine times the volume of the cylinder of the pump is enough. Further the air at the top of the air vessel also gets slowly dissolved in the liquid under pressure. As such the air in the air vessel must be renewed periodically. In the case of larger pumps this is usually done by a special air compressor provided for this purpose. However, in small pumps an automatic air valve called a *snifting valve* is provided which may draw in the necessary quantity of the air.

(a) Rate of Flow of Liquid into and from the Air Vessel. Consider a single acting pump with air vessels fitted to both the suction and delivery pipes. Since a liquid in the suction and delivery pipes beyond the air vessel will move with a constant mean velocity, the mean discharge in the delivery pipe beyond the air vessel or in the suction pipe below the air vessel is given by Eq. 23.32 as

$$(Q_d)_{\text{mean}} = \frac{A\omega r}{\pi}$$

Further the instantaneous discharge to or from the cylinder of the pump is given by Eq. 23.31 as

$$Q_d = A\omega r \sin\theta$$

Obviously the difference of the above two discharge will be equal to the rate of flow of liquid into or from the air vessel.

∴ Rate of flow of liquid into the air vessel

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \left(A\omega r \sin\theta - \frac{A\omega r}{\pi} \right)$$

$$= A\omega r \left(\sin\theta - \frac{1}{\pi} \right) \quad ...(23.53)$$

If Eq. 23.53 is positive, it means that the liquid is flowing into the air vessel fitted to the delivery pipe. But if this equation is negative, then it means that the liquid is flowing from the air vessel.

However, if we consider the flow into or from the air vessel fitted to the suction pipe then the above condition will be reversed. That is if Eq. 23.53 is positive then the liquid is flowing from the air vessel. But if Eq. 23.53 is negative then the liquid is flowing into the air vessel.

For no flow of liquid into or from the air vessel, Eq. 23.53 is put equal to zero. That is,

$$A\omega r \left(\sin\theta - \frac{1}{\pi} \right) = 0 ; \text{ or } \sin\theta = \frac{1}{\pi} = 0.3183$$

$$\therefore \quad \theta = 18° 34' \text{ or } 161° 26'$$

Thus for crank angle $\theta = 18° 34'$ and $161° 26'$ there will be no flow into or from the air vessel. Further for crank angle $\theta = 0°$ to $18° 34'$ and $\theta = 161° 26'$ to $180°$ the instantaneous discharge is less than the mean discharge and for $\theta = 18° 34'$ to $161° 26'$ the instantaneous discharge is more than the mean discharge.

For a 'double acting pump' the mean discharge in the suction pipe below the air vessel or in the delivery pipe beyond the air vessel is given by Eq. 23.37 as

$$(Q_d)_{\text{mean}} = \frac{2A\omega r}{\pi}$$

Since in this case also the instantaneous discharge to or from the cylinder is given by Eq. 23.31 as

$$Q_d = A\omega r \sin\theta$$

Rate of flow of liquid into the air vessel

$$= \left(A\omega r \sin\theta - \frac{2A\omega r}{\pi} \right)$$

$$= A\omega r \left(\sin\theta - \frac{2}{\pi} \right) \quad ...(23.54)$$

Again, if Eq. 23.54 is positive it means that the liquid is flowing into the air vessel fitted to delivery pipe. But if this equation is negative it means that the liquid is flowing from the air vessel.

For the air vessel fitted to the suction pipe the above condition will be reversed. Thus, if Eq. 23.54 is positive then the liquid is flowing from the air vessel. But if this equation is negative then the liquid is flowing into the air vessel.

For no flow of liquid into or from the vessel, Eq. 23.54 is equated to zero. That is,

$$A\omega r \left(\sin\theta - \frac{2}{\pi} \right) = 0 ; \text{ or } \sin\theta = \frac{2}{\pi} = 0.6366$$

$$\therefore \quad \theta = 39° 32' \text{ or } 140° 28'$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus, in the case of a double acting pump for crank angle θ = 39° 32′ and 140° 28′ there will be no flow into or from the air vessel. Again, for crank angle θ = 0° to 39° 32′ and θ = 140° 28′ to 180°, the instantaneous discharge is less than the mean discharge and for θ = 39° 32′ to 140° 28′ the instantaneous discharge is more than the mean discharge.

(b) Pressure Heads in the Cylinder during Delivery Stroke of a Pump with Air Vessel. If l_{dv} is the length of the delivery pipe between the cylinder and the point at which an air vessel is fitted to it as shown in Fig. 23.6 then the acceleration head developed is given by Eq. 23.18 as

$$(H_{ad})_v = \frac{l_{dv}}{g} \times \frac{A}{a_d}\left(\omega^2 r \cos\theta\right) \qquad ...(23.55)$$

Similarly the head loss due to friction in this position of the pipe is given by Eq. 23.25 as

$$(h_{fd})_2 = \left(\frac{fl_{dv}}{2gd_d}\right)\left(\frac{A}{a_d}\omega r \sin\theta\right)^2 \qquad ...(23.56)$$

If l_d is the length of the delivery pipe beyond the air vessel and v_d is the constant mean velocity in the delivery pipe beyond the air vessel then the head loss due to friction in this portion of the delivery pipe is given by Darcy-Weisbach equation as

$$(h_{fd})_1 = \frac{fl_d v_d^2}{2gd_d} \qquad ...(23.57)$$

The constant mean velocity v_d is given by Eq. 23.28 for a single acting pump and by Eq. 23.35 for a double acting pump.

The total pressure head developed during the delivery stroke of a reciprocating pump with air vessel may thus by obtained for any crank angle θ which may be expressed as

$$H_{td} = H_d + \frac{v_d^2}{2g} + (H_{ad})_v + (h_{fd})_1 + (h_{fd})_2 \qquad ...(23.58)$$

and $$(H_{td})_{\text{absolute}} = H_{td} + H_{\text{atm}} \qquad ...(23.59)$$

From Eqs. 23.55 to 23.59, by putting θ = 0°, 90° and 180°, for the beginning, middle and end of the delivery stroke respectively, the corresponding values of H_{td} can be obtained. However, if l_{dv} is very small, $(H_{ad})_v$ and $(h_{fd})_2$ can be neglected. Sometimes the velocity head $(v_d^2/2g)$ is also neglected then

$$H_{td} = H_d + (h_{fd})_1 \qquad ...(23.60)$$

Further the total pressure head of the air in the air vessel on the delivery side reckoned above the centre of the cylinder will be approximately equal to the total pressure head in the delivery pipe above the same datum. Thus if h_{dv} represents the pressure head of the air in the air vessel and H_{dv} is the vertical height of the liquid surface in the air vessel above the centre of the cylinder, then

$$(h_{dv} + H_{dv}) = H_{td} + H_{\text{atm}} = (H_{td})_{\text{absolute}} \qquad ...(23.61)$$

where H_{td} can be obtained from Eq. 23.58 or 23.60.

(c) Pressure Heads in the Cylinder during Suction Stroke of a Pump with Air Vessel. The same analysis as that for the delivery stroke given above in section 23.8 (b) may be applied to compute the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

various pressure heads developed during the suction stroke. Thus, the corresponding relationships for suction stroke may be written by using subscript s in place of d in Eqs. 23.55 to 23.61. Thus

$$(H_{as})_v = \frac{l_{sv}}{g} \times \frac{A}{a_s}(\omega^2 r \cos\theta) \quad ...(23.62)$$

$$(h_{fs})_2 = \frac{fl_{sv}}{2gd_s}\left(\frac{A}{a_s}\omega r \sin\theta\right) \quad ...(23.63)$$

$$(h_{fs})_1 = \frac{fl_s v_s^2}{2gd_s} \quad ...(23.64)$$

in which v_s is the constant mean velocity in the suction pipe below the point at which the air vessel is connected to it, which for single acting pump is given by

$$v_s = \frac{2ArN}{60a_s} = \frac{ALN}{60a_s} = \frac{A\omega r}{\pi a_s}$$

and for a double acting pump by

$$v_s = \frac{4ArN}{60a_s} = \frac{2ALN}{60a_s} = \frac{2A\omega r}{\pi a_s}$$

The total pressure head developed during the suction stroke of a reciprocating pump with air vessel may thus be obtained for any crank angle θ which may be expressed as

$$H_{ts} = H_s + \frac{v_s^2}{2g} + (H_{as})_v + (h_{fs})_1 + (h_{fs})_2 \quad ...(23.65)$$

Again if $(H_{as})_v$, $(h_{fs})_2$ and $(v_s^2/2g)$ are negligibly small, then

$$H_{ts} = H_s + (h_{fs})_1 \quad ...(23.66)$$

Since during the suction stroke the total pressure head developed is below atmospheric pressure or it is vaccum pressure head and hence

$$(H_{ts})_{\text{absolute}} = H_{\text{atm}} - H_{ts} \quad ...(23.67)$$

Again if h_{sv} represents the total pressure head of the air in the air vessel on the suction side then as in the case of delivery stroke

$$h_{sv} - H_{sv} = H_{\text{atm}} - H_{ts} \quad ...(23.68)$$

(d) Work Done by a Reciprocating Pump with Air Vessels. For a pump fitted with air vessel on both suction and delivery pipes, the theoretical work done per second can be expressed as

$$\text{Work done} = wQ_{th}\left[(H_s + H_d) + (h_{fs})_1 + (h_{fd})_1 + \left(\frac{v_{s^2}}{2g} + \frac{v_{d^2}}{2g}\right) + \frac{2}{3}(h_{fs})_2 + \frac{2}{3}(h_{fd})_2\right] \quad ...(23.69)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Since the acceleration heads do not affect the total work done. In Eq. 23.69 the values of $(h_{fs})_2$ and $(h_{fd})_2$ correspond to $\theta = 90°$ i.e., these are respectively the maximum values of the head loss due to friction in the portions of the suction and delivery pipes between the cylinder and the air vessels. However, if all the small quantities are neglected then

$$\text{Work done} = wQ_{th}[(H_s + H_d) + (h_{fs})_1 + (h_{fd})_1)] \quad \ldots(23.70)$$

Equation 23.70 also gives the theoretical power required to drive the pump fitted with air vessels.

(e) Effect of Air Vessels on the Indicator Diagram. Since by fitting air vessels the pressure heads developed during the suction and delivery strokes of the pump will also be modified. Thus if all the quantities of Eqs 23.58 and 23.65 are taken into account then the modified indicator diagram will be as shown in Fig. 23.6. However, if small quantities are neglected then from Eqs 23.60 and 23.66, the indicator diagram will be simply rectangular as shown in Fig. 23.6, since $(h_{fs})_1$ and $(h_{fd})_1$ are independent of the crank angle θ.

(f) Maximum Speed of a Reciprocating Pump with Air Vessel. As indicated in section 23.7 (c) the maximum permissible speed of a reciprocating pump is governed by the drop of pressure in the cylinder at the beginning of the suction stroke. Since the pressure in the cylinder must not drop below H_{sep}, the maximum speed of the pump may be computed by the expression

$$H_{sep} = H_s + (h_{fs})_1 + \frac{v_{s^2}}{2g} + (H_{as})_v$$

or
$$H_{sep} = H_s + \frac{fl_s v_s^2}{2gd_s} + \frac{v_s^2}{2g} + \frac{l_{sv}}{g} \times \frac{A}{a_s}(\omega^2 r)$$

or
$$H_{sep} = H_s + \frac{fl_s}{2gd_s}\left(\frac{A\omega r}{\pi a_s}\right)^2 + \frac{1}{2g}\left(\frac{A\omega r}{\pi a_s}\right)^2 + \frac{l_{sv}}{g} \times \frac{A}{a_s}(\omega^2 r)$$

or
$$H_{sep} = H_s + \frac{fl_s}{2gd_s}\left(\frac{2ArN}{60a_s}\right)^2 + \frac{1}{2g}\left(\frac{2ArN}{60a_s}\right)^2 + \frac{l_{sv}}{g} \times \frac{A}{a_s}\left(\frac{2\pi N}{60}\right)^2 r \quad \ldots(23.71)$$

However, if the small quantities are neglected then the limiting speed of the pump may be computed by the following expression:

$$H_{sep} = H_s + \frac{fl_s}{2gd_s}\left(\frac{2ArN}{60a_s}\right)^2 \quad \ldots(23.72)$$

(g) Work Saved against Friction by Fitting Air Vessel. When a pump is equipped with air vessels, the fluctuations in the velocity of flow in suction and delivery pipes are eliminated, which results in reducing the head loss due to friction in these pipes and hence certain amount of energy is saved. The saving in the energy may be obtained by the following analysis which is applicable to both the suction as well as delivery strokes.

For a single acting pump without air vessel on the pipe the work W_1 lost in friction per stroke is given by

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$W_1 = wAL\left(\frac{2}{3}h_f\right)$$

$$= wA\left(\frac{4r}{3}\right)\left(\frac{fl}{2gd}\right)\left(\frac{A}{a}\omega r\right)^2 \quad \ldots(23.73)$$

Now if an air vessel is fitted to the pipe just outside the cylinder, the velocity of flow in the pipe will be constant being equal to the mean velocity and therefore the work W_2 lost in friction per stroke is given by

$$W_2 = wAL(h_f)_1 = wA\,(2r)\left[\frac{flv^2}{2gd}\right] \quad \ldots(23.74)$$

But from the Eq. 23.32

$$v = \frac{A\omega r}{\pi a}$$

Thus
$$W_2 = wA\,(2r)\left[\frac{fl}{2gd}\right]\left[\frac{A}{a}\frac{\omega r}{\pi}\right]^2 \quad \ldots(23.75)$$

The work saved per stroke by fitting an air vessel to the suction or delivery pipe of a single acting pump is obtained by subtracting Eq. 23.75 from Eq. 23.73 as

$$W_1 - W_2 = (2r)wA\left[\frac{fl}{2gd}\right]\left[\frac{A}{a}\omega r\right]^2\left[\frac{2}{3} - \frac{1}{\pi^2}\right] \quad \ldots(23.76)$$

The percentage of the work saved during the stroke

$$= \frac{W_1 - W_2}{W_1} \times 100 = \frac{\left(\frac{2}{3}\right) - \left(\frac{1}{\pi^2}\right)}{\left(\frac{2}{3}\right)} \times 100$$

$$= 84.8\%$$

For a 'double acting pump' without air-vessel the work W_1 lost in friction per stroke is same as in the case of single acting pump as given by Eq. 23.73.

When an air vessel is fitted to the pipe outside the cylinder, then the work W_2 lost in friction per stroke is also given by Eq. 23.74. But for a double acting pump from Eq. 23.37

$$v = \frac{2A\omega r}{\pi a}$$

$\therefore$
$$W_2 = wA\,(2r)\left[\frac{fl}{2gd}\right]\left[\frac{2A\omega r}{\pi a}\right]^2 \quad \ldots(23.77)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

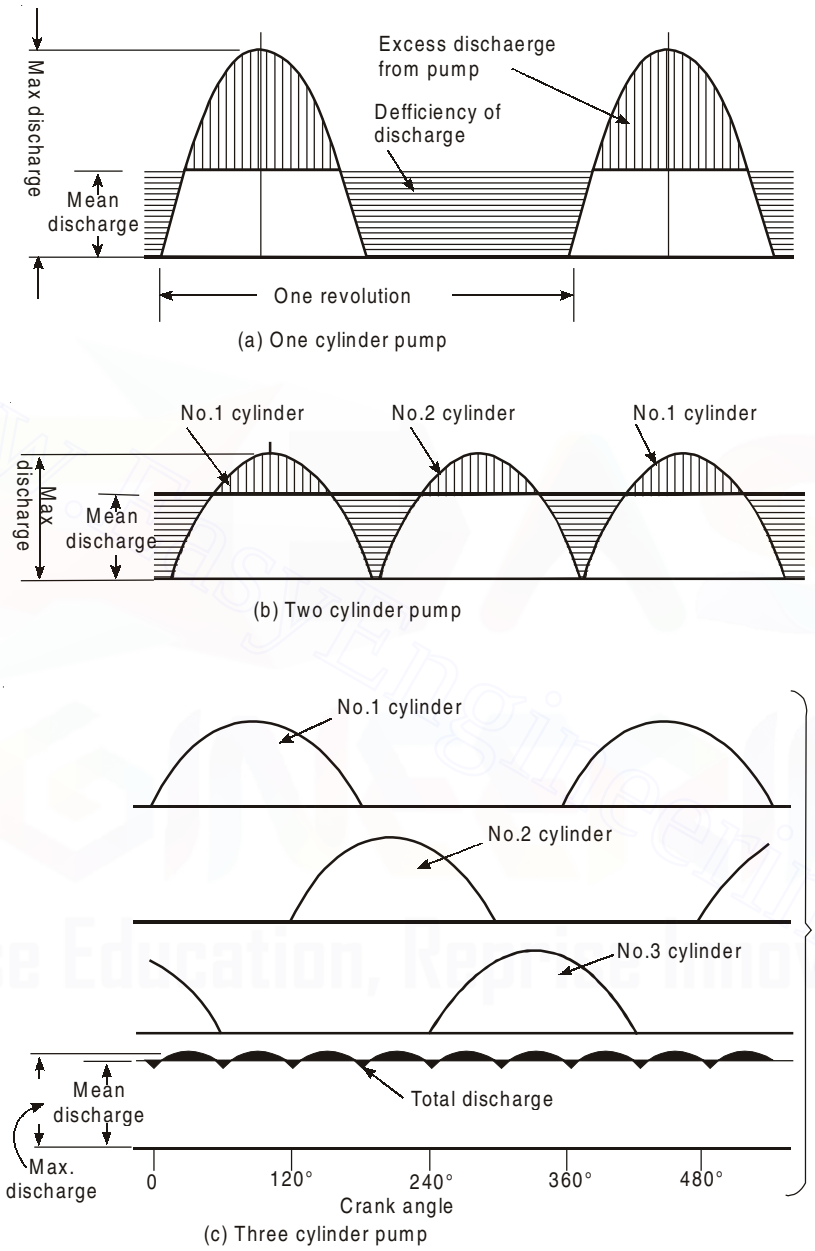


Figure 23.7 *Discharge diagram for one, two, and three cylinder pumps (without air vessels)*

By subtracting Eq. 23.77 from Eq. 23.73 the work saved per stroke by fitting an air vessel to the suction or delivery pipe of a double acting pump is obtained as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$W_1 - W_2 = (2r)wA\left[\frac{fl}{2gd}\right]\left[\frac{A}{a}\omega r\right]^2\left[\frac{2}{3}-\frac{4}{\pi^2}\right] \quad \ldots(23.78)$$

Again the percentage of the work saved during any stroke for a double acting pump

$$= \frac{W_1 - W_2}{W_1} \times 100 = \frac{\left(\frac{2}{3}\right)-\left(\frac{4}{\pi^2}\right)}{\left(\frac{2}{3}\right)} \times 100$$

$$= 39.2\%.$$

23.9 MULTI-CYLINDER PUMPS

As discussed earlier, pumps having more than one cylinder are known as 'multicylinder pumps'. Some of these pumps are shown in Fig. 23.2. The main advantage of multicylinder pumps are that these pumps even without air vessels deliver liquid more uniformly as compared with a single cylinder pump. This is made clear by the various discharge diagrams for different types of pumps shown in Fig. 23.7, in which the instantaneous discharge from each individual cylinder is compared with the mean discharge from the pump. It will be observed that by a suitable disposition of the discharge from the individual cylinder, the total discharge of a multicylinder pump may be made nearly equal to the mean discharge with almost negligible fluctuation. On account of much smaller fluctuations of velocity in both the suction and delivery pipes of a multicylinder pump, no air-vessel is required for the same.

23.10 OPERATING CHARACTERISTIC CURVES OF RECIPROCATING PUMPS

The operating characteristic curves indicating the performance of a reciprocating pump are shown in Fig. 23.8. These curves are obtained by plotting discharge, power input and overall efficiency against the head developed by the pump when it is operating at a constant speed. As shown in Fig. 23.8, under

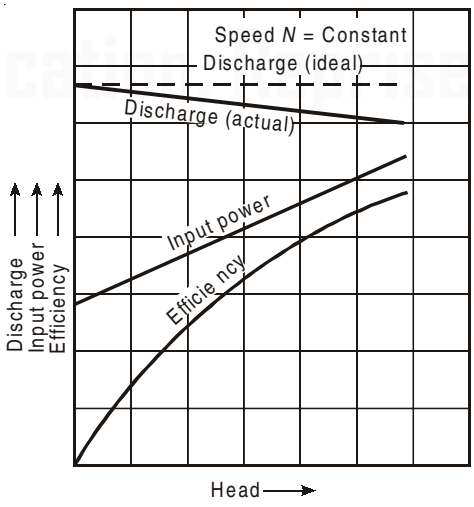


Figure 23.8 *Operating characteristic curves of a reciprocating pump*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

ideal conditions the discharge of a reciprocating pump operating at constant speed is independent of the head developed by the pump. However, in actual practice it is observed that the discharge of a reciprocating pump slightly decreases as the head developed by the pump increases. Further the input power for a reciprocating pump increases almost linearly beyond a certain minimum value with the increase in the head developed by the pump. The overall efficiency of a reciprocating pump also increases with the increase in the head developed by the pump as shown in Fig. 23.8.

In addition to the above noted constant speed characteristic curves, sometimes variable speed characteristic curves of a reciprocating pump are also plotted. For this the pump is operated at different speeds and its discharge corresponding to a constant head is plotted against the speed. It is observed that the discharge of a reciprocating pump increases almost linearly with the increase in the speed of the pump, which is in accordance with the expression for the theoretical discharge given by Eqs. 23.1 and 23.8.

ILLUSTRAITVE EXAMPLES

Example 23.1. *A single acting reciprocating pump has a plunger of diameter 250 mm and stroke of 350 mm. If the speed of the pump is 60 r.p.m. and it delivers 16.5 litres per second of water against a suction head of 5 m and a delivery head of 20 m, find the theoretical discharge, coefficient of discharge, the slip, the percentage slip of the pump and the power required to drive the pump.*

Solution

Theoretical discharge of the pump

$$Q_{th} = \frac{ALN}{60}$$

$$= \frac{\pi}{4}\left(\frac{250}{1000}\right)^2 \times \frac{350}{1000} \times \frac{60}{60}$$

$$= 0.0172 \text{ m}^3/\text{s} = 17.2 \ l/\text{s}$$

Coefficient of discharge

$$C_d = \frac{Q_a}{Q_{th}} = \frac{16.5}{17.2} = 0.96$$

$$\text{Slip of the pump} = (Q_{th} - Q_a)$$

$$= (17.2 - 16.5) = 0.7 \ l/\text{s}$$

$$\text{Percentage slip} = \frac{Q_{th} - Q_a}{Q_h} \times 100$$

$$= \frac{17.2 - 16.5}{17.2} \times 100 = 4.07\%.$$

Power required to drive the pump

$$p = wQ_{th} (H_s + H_d)$$

$$= 9810 \times 0.0172 \times (5 + 20)$$

$$= 4218.3 \text{ W} = 4.2183 \text{ kW}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 23.2. *A single acting reciprocating pump has piston of diameter 150 mm and stroke of length 250 mm. The piston makes 50 double strokes per minute. The suction and delivery heads are 5 m and 15 m respectively. Find (i) discharge capacity of the pump in litres per minute; (ii) force required to work the piston during the suction and delivery strokes if the efficiency of suction and delivery strokes are 60% and 75% respectively; and (iii) power required to operate the pump.*

Solution

(i) Theoretical discharge of the pump

$$Q_{th} = \frac{ALN}{60}$$

$$= \frac{\pi}{4}\left(\frac{150}{1000}\right)^2 \times \frac{250}{1000} \times \frac{50}{60}$$

$$= 0.00368 \text{ m}^3/\text{s}$$

$$= 3.68 \, l/\text{s} = 221 \text{ litres/min}$$

(ii) Force required to work the piston during the suction stroke

$$= \left(\frac{wH_s A}{\eta_1}\right)$$

$$= \frac{9810 \times 5 \times (\pi/4)(150/1000)^2}{0.6}$$

$$= 1444.64 \text{ N}$$

Force required to work the piston during the delivery stroke

$$= \left(\frac{wH_d A}{\eta_2}\right)$$

$$= \frac{9810 \times 15 \times (\pi/4)(150/1000)^2}{0.75}$$

$$= 3467.14 \text{ N}$$

(iii) Power required to operate the pump

$$P = wQ_{th}\left(\frac{H_s}{\eta_1} + \frac{H_d}{\eta_2}\right)$$

$$= 9810 \times 0.00368 \times \left(\frac{5}{0.60} + \frac{15}{0.75}\right)$$

$$= 1023 \text{ W} = 1.023 \text{ kW}$$

Note: Equations 23.5 and 23.6 give only theoretical force acting on the piston or plunger during the suction and delivery strokes respectively and hence in order to obtain the actual force acting on the piston or plunger during these strokes the expressions in these equations are divided by the efficiencies of the respective strokes.

Example 23.3. *A double acting reciprocating pump has piston of diameter 250 mm and piston rod of diameter 50 mm which is on one side only. Length of piston stroke is 350 mm and speed of crank moving the*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

piston is 60 r.p.m. The suction and delivery heads are 4.5 m and 18 m respectively. Determine the discharge capacity of the pump and the power required to operate the pump.

Solution

For a double acting pump the theoretical discharge is given by

$$Q_{th} = \frac{(2A - A_p)LN}{60}$$

$$A = \frac{\pi}{4}\left(\frac{250}{1000}\right)^2 = 0.04909 \text{ m}^2$$

$$A_p = \frac{\pi}{4}\left(\frac{50}{1000}\right)^2 = 0.00196 \text{ m}^2$$

$$L = \frac{350}{1000} = 0.35 \text{ m; N} = 60 \text{ r.p.m}$$

Thus by substitution, we get

$$Q_{th} = \frac{(2 \times 0.04909 - 0.00196) \times 0.35 \times 60}{60}$$

$$= 0.0337 \text{ m}^3/\text{s} = 33.7 \text{ } l/\text{s}$$

Power required to drive the pump

$$= wQ_{th}(H_s + H_d)$$
$$= 9810 \times 0.0337 \ (4.5 + 18)$$
$$= 7438 \text{ W} = 7.438 \text{ kW}$$

Example 23.4. *A double acting reciprocating pump having piston area 0.1 m^2 has a stroke 0.30 m long. The pump is discharging 2.4 m^3 of water per minute at 45 r.p.m. through a height of 10 m. Find the slip of the pump and the power required to drive the pump.*

Solution

Neglecting the area of the piston rod, the theoretical discharge of the pump is

$$Q_{th} = \frac{2ALN}{60} = \frac{2 \times 0.1 \times 0.3 \times 45}{60}$$

$$= 0.045 \text{ m}^3/\text{s}$$

Actual discharge $$Q_a = \frac{2.4}{60} = 0.040 \text{ m}^3/\text{s}$$

$\therefore$ Slip of the pump $$= (Q_{th} - Q_a)$$
$$= (0.045 - 0.040) = 0.005 \text{ m}^3/\text{s}$$

Power required to drive the pump

$$= wQ_{th}(H_s + H_d)$$
$$= 9810 \times 0.045 \times 10$$
$$= 4\,415 \text{ W} = 4.415 \text{ kW}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 23.5. *A single acting reciprocating pump has a piston diameter 0.15 and stroke length 0.30. The centre of the pump is 5 m above the level of water in sump and 33 m below delivery water level. The length of suction and delivery pipes are 6.5 m and 39 m respectively and both the pipes have the same diameter of 75 mm. If the pump is working at 30 r.p.m. find the pressure head on the piston at the beginning, middle and end of both suction and delivery strokes, and find the power required to drive the pump. Take atmospheric pressure as 10.3 m of water and Darcy's friction factor for both the pipes as 0.04.*

Solution

Suction Stroke:

At the ends of the stroke

$$H_{as} = \frac{l_s}{g} \times \frac{A}{a_s} \omega^2 r$$

$$= \frac{6.5}{9.81} \times \left[\frac{\left(\frac{\pi}{4}\right) \times (0.15)^2}{\left(\frac{\pi}{4}\right) \times (0.075)^2} \right] \times \left(\frac{2\pi \times 30}{60} \right)^2 \times 0.15$$

$$= 3.92 \text{ m of water}$$

At the middle of the stroke

$$h_{fs} = \frac{f l_s}{2 g d_s} \left(\frac{A}{a_s} \omega r \right)^2$$

$$= \frac{0.04 \times 6.5}{2 \times 9.81 \times 0.075} \times \left[\frac{\left(\frac{\pi}{4}\right) \times (0.15)^2}{\left(\frac{\pi}{4}\right) \times (0.075)^2} \times \left(\frac{2\pi \times 30}{60} \right) \times 0.15 \right]^2$$

$$= 0.628 \text{ m of water}$$

$\therefore$ Pressure heads in metres of water during suction stroke will be as follows:

At beginning $= (H_s + H_{as}) = (5 + 3.92) = 8.92$ m (vacuum)

$= (10.3 - 8.92) = 1.38$ m (abs)

At middle $= (H_s + h_{fs}) = (5 + 0.628) = 5.628$ m (vacuum)

$= (10.3 - 5.628) = 4.672$ m (abs)

At end $= (H_s - H_{as}) = (5 - 3.92) = 1.08$ m (vacuum)

$= (10.3 - 1.08) = 9.22$ m (abs)

Delivery Stroke:

At the ends of the strokes

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{39}{9.81}\left[\frac{\left(\frac{\pi}{4}\right)\times(0.15)^2}{\left(\frac{\pi}{4}\right)\times(0.075)^2}\right]\times\left(\frac{2\pi\times30}{60}\right)^2\times0.15$$

$$= 23.54 \text{ m of water}$$

At the middle of the stroke

$$h_{fd} = \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r\right)^2$$

$$= \frac{0.04\times39}{2\times9.81\times0.075}\times\left[\frac{\left(\frac{\pi}{4}\right)\times(0.15)^2}{\left(\frac{\pi}{4}\right)\times(0.075)^2}\times\left(\frac{2\pi\times30}{60}\right)\times0.15\right]^2$$

$$= 3.767 \text{ m of water}$$

$\therefore$ Pressure heads in metres of water during suction stroke will be as follows:

At beginning $= (H_d + H_{ad}) = (33 + 23.54) = 56.54$ m (gage)
$= (10.3 + 56.54) = 66.84$ m (abs)

At middle $= (H_d + h_{fd}) = (33 + 3.767) = 36.767$ m (gage)
$= (10.3 + 36.767) = 47.067$ m (abs)

At end $= (H_d - H_{ad}) = (33 - 23.54) = 9.46$ m (gage)
$= (10.3 + 9.46) = 19.76$ m (abs)

Theoretical discharge of the pump

$$Q_{th} = \frac{ALN}{60}$$

$$= \frac{\pi}{4}(0.15)^2 \times (0.30) \times \left(\frac{30}{60}\right)$$

$$= 2.65 \times 10^{-3} \text{ m}^3/\text{s}$$

Work done by the pump

$$= wQ\left[H_s + H_d + \frac{2}{3}h_{fs} + \frac{2}{3}h_{fd}\right]$$

$$= (9810 \times 2.65 \times 10^{-3})\left[5 + 33 + \frac{2}{3}(0.628) + \frac{2}{3}(3.767)\right]$$

$$= 1\,064 \text{ N.m/s}$$

Power required to drive the pump

$$= 1064 \text{ W} = 1.064 \text{ kW}.$$

Example 23.6. *For a pump with no air vessel, stroke length = 300 mm, piston diameter = 125 mm, suction pipe diameter = 75 mm, length of suction pipe = 6 m and suction head = 3 m. Atmospheric pressure = 10.3 m of*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

water and separation may be assumed to occur when the absolute pressure head in the cylinder falls below 2.5 m of water.

Calculate the maximum speed at which the pump may be run if separation is to be avoided.

Solution

From Eq. 23.43, we have

$$H_{sep} = (H_s + H_{as})$$
$$H_{sep} = (10.3 - 2.5) = 7.8 \text{ m of water}$$
$$H_s = 3.0 \text{ m}$$

At the beginning of the suction stroke, we have

$$H_{as} = \frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r$$
$$= \frac{6}{9.81} \times \left[\frac{(\pi/4)\times(0.125)^2}{(\pi/4)\times(0.075)^2}\right] \times \omega^2 \times 0.15$$
$$= 0.255\ \omega^2$$

Thus by substitution, we have

$$7.8 = 3.0 + 0.255\ \omega^2$$

or $$\omega = 4.34 \text{ rad/s}$$

Since $$\omega = \frac{2\pi N}{60}$$

$$4.34 = \frac{2\pi N}{60}$$

$$\therefore \quad N = 41.4 \text{ r.p.m.}$$

Examples 23.7. *A single acting reciprocating pump has a plunger of 80 mm diameter and a stroke of length 150 mm. It takes its supply of water from a sump 3 m below the pump through a pipe 4.5 m long and 30 mm diameter. It delivers water to a tank 12 m above the pump through a pipe 25 mm diameter and 15 m long.*

If separation occurs at 78.48 kN/m^2 below atmospheric pressure, find the maximum speed at which the pump may be operated without separation, assume the plunger to have simple harmonic motion.

Solution

At the beginning of the suction stroke, we have

$$H_{as} = \frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r$$
$$= \frac{4.5}{9.81} \times \left[\frac{(\pi/4)\times(0.08)^2}{(\pi/4)\times(0.03)^2}\right] \times \omega^2 \times 0.075$$
$$= 0.245\ \omega^2$$

From Eq. 23.43, we have

$$H_{sep} = H_s + H_{as}$$
$$H_{sep} = \frac{78.48\times10^3}{9810} = 8 \text{ m of water}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_s = 3 \text{ m}$$

Thus by substitution, we get

$$8 = 3 + 0.245\ \omega^2$$

or $$\omega = \left(\frac{5}{0.245}\right)^{1/2} = 4.518 \text{ rad/s}$$

Since $$\omega = \frac{2\pi N}{60}$$

$$4.518 = \frac{2\pi N}{60}$$

$$\therefore \quad N = 43.14 \text{ r.p.m.}$$

At the end of the delivery stroke, we have

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r$$

$$= \frac{15}{9.81} \times \left[\frac{(\pi/4)\times(0.08)^2}{(\pi/4)\times(0.025)^2}\right] \times \omega^2 \times 0.075$$

$$= 1.174\ \omega^2$$

From the limiting condition, we have

$$H_{ad} - H_d = H_{sep}$$

or $$1.174\ \omega^2 - 12 = 8$$

or $$\omega = \left(\frac{20}{1.174}\right)^{1/2} = 4.127 \text{ rad/s}$$

Since $$\omega = \frac{2\pi N}{60}$$

$$4.127 = \frac{2\pi N}{60}$$

$$\therefore \quad N = 39.41 \text{ r.p.m.}$$

Thus the maximum speed at which the pump may be operated without separation is the minimum of the two speeds obtained above viz., 39.41 r.p.m.

Example 23.8. *Determine the maximum speed at which a double acting reciprocating pump can be operated under the following conditions: (a) no air vessel on the suction side; (b) a very large air vessel on the suction side close to the pump.*

The suction lift is 4 m, length of suction pipe 6.5 m, diameter of suction pipe 100 mm, diameter of piston 150 mm and length of stroke is 0.45 m. Assume simple harmonic motion, atmospheric pressure head as 10.3 m of water and separation occurs at 2.6 m of water absolute. Take Darcy's f = 0.024.

Solution

(a) When no air vessel is fitted, we have from Eq. 23.43

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_{sep} = H_s + H_{as}$$

$$H_{sep} = (10.3 - 2.6) = 7.7 \text{ m of water}$$

$$H_s = 4 \text{ m}$$

$$H_{as} = \frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r$$

$$= \frac{6.5}{9.81} \times \left[\frac{(\pi/4)\times(0.15)^2}{(\pi/4)\times(0.10)^2}\right] \times \omega^2 \times \left(\frac{0.45}{2}\right)$$

$$= 0.335\ \omega^2$$

Thus by substitution, we get

$$7.7 = 4 + 0.335\ \omega^2$$

or $$\omega = \left(\frac{3.7}{0.335}\right)^{1/2} = 3.323 \text{ rad/s}$$

But $$\omega = \frac{2\pi N}{60}$$

or $$3.323 = \frac{2\pi N}{60}$$

$\therefore$ $$N = 31.73 \text{ r.p.m.}$$

(b) With an air vessel fitted very close to the cylinder, we have from Eq. 23.71

$$H_{sep} = H_s + \frac{v_{s_2}}{2g} + (h_{fs})_1$$

$$H_{sep} = (10.3 - 2.6) = 7.7 \text{ m of water}$$

$$H_s = 4 \text{ m}$$

$$Q = \frac{2ALN}{60}$$

$$= 2 \times \frac{\pi}{4} \times (0.15)^2 \times 0.45 \times \left(\frac{N}{60}\right)$$

$$= (2.651 \times 10^{-4} N)$$

$\therefore$ $$v_s = \frac{Q}{a_s}$$

$$= \frac{(2.651\times10^{-4}N)}{(\pi/4)\times(0.10)^2} = (3.375 \times 10^{-2} N)$$

and $$\frac{v_{s^2}}{2g} = \frac{(3.375\times10^{-2}N)^2}{2\times9.81} = (0.58 \times 10^{-4} N^2)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h_{fs1} = \frac{fl_s v_s^2}{2gd_s}$$

$$= \frac{0.024 \times 6.5 \times 0.58 \times 10^{-4} N^2}{0.10} = 0.905 \times 10^{-4} N^2$$

Thus by substitution, we get

$$7.7 = 4.0 + 0.58 \times 10^{-4} N^2 + 0.905 \times 10^{-4} N^2$$

or $$N^2 = \frac{3.7}{1.485 \times 10^{-4}}$$

$$\therefore \quad N = 158 \text{ r.p.m.}$$

Note: It may be noted that by fitting an air vessel the speed of a pump may be increased considerably without causing the separation to occur.

Example 23.9. *A single acting reciprocating pump of 0.2 m diameter and 0.4 m stroke runs at 30 r.p.m. The suction and delivery heads are 3.6 m and 30 m respectively. The length of suction pipe is 9 m and that of delivery pipe is 60 m and the diameter of each pipe is 100 mm. Large air vessels are provided 2 m away from the pump on the suction side and 6 m away on the delivery side, both measured along the pipeline. Darcy's friction factor for the pipes is 0.032 and H_{atm}=10.3 m of water.*

(a) Determine the expressions for total absolute pressure heads H_{ts} and H_{td} in terms of θ.

(b) Calculate the work done per second and the power required to drive the pump.

(c) If the mean level of water in the vessel on delivery side is 1 m above the centreline of the pump, find the pressure of air in the air vessel in metres of water for $\theta = 0$.

Solution

(a) From Eq. 23.65, we have

$$H_{ts} = H_s + \frac{v_{s^2}}{2g} + (H_{as})_v + (h_{fs})_1 + (h_{fs})_2$$

$$H_s = 3.6 \text{ m}$$

$$v_s = \frac{ALN}{60a_s}$$

$$= \frac{(\pi/4)(0.2)^2 \times 0.4 \times 30}{60 \times (\pi/4)(0.1)^2} = 0.8 \text{ m/s}$$

$$\frac{v_{s^2}}{2g} = \frac{(0.8)^2}{2 \times 9.81} = 0.033 \text{ m}$$

$$(H_{as})_v = \frac{l_{sv}}{g} \times \frac{A}{a_s} \times (\omega^2 r \cos\theta)$$

$$= \frac{2}{9.81} \times \frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2} \times \left(\frac{2\pi \times 30}{60}\right)^2 \times 0.2 \cos\theta$$

$$= 1.61 \cos\theta$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$(h_{fs})_1 = \frac{fl_s v_{s^2}}{2gd_s} = \frac{0.032\times(9-2)\times 0.033}{0.1} = 0.074 \text{ m}$$

$$(h_{fs})_2 = \left(\frac{fl_{sv}}{2gd_s}\right)\left(\frac{A}{a_s}\omega r\sin\theta\right)^2$$

$$= \frac{0.032\times 2}{2\times 9.81\times 0.10}\left[\frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2}\times\left(\frac{2\pi\times 30}{60}\right)\times 0.2\sin\theta\right]^2$$

$$= 0.206 \sin^2\theta$$

Thus by substitution, we have

$$H_{ts} = 3.6 + 0.033 + 1.61\cos\theta + 0.074 + 0.206\sin^2\theta$$

or $$H_{ts} = 3.707 + 1.61\cos\theta + 0.206\sin^2\theta$$

From Eq. 23.67, we have

$$(H_{ts})_{\text{absolute}} = H_{\text{atm}} - H_{ts}$$
$$= 10.3 - [3.707 + 1.61\cos\theta + 0.206\sin^2\theta]$$
$$= 6.593 - 1.61\cos\theta - 0.206\sin^2\theta$$

Again from Eq. 23.58, we have

$$H_{td} = H_d + \frac{v_d^2}{2g} + (H_{ad})_v + (h_{fd})_1 + (h_{fd})_2$$

$$H_d = 30 \text{ m}; v_d = v_s = 0.8 \text{ m/s}$$

$$\frac{v_d^2}{2g} = \frac{(0.8)^2}{2\times 9.81} = 0.033 \text{ m}$$

$$(H_{ad})_v = \frac{l_{dv}}{g}\times\frac{A}{a_d}(\omega^2 r\cos\theta)$$

$$= \frac{6}{9.81}\times\frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2}\times\left(\frac{2\pi\times 30}{60}\right)^2\times 0.2\cos\theta$$

$$= 4.83\cos\theta$$

$$(H_{fd})_1 = \frac{fl_d v_d^2}{2gd_d} = \frac{0.032\times(60-6)\times 0.033}{0.1} = 0.57 \text{ m}$$

$$(H_{fd})_2 = \left(\frac{fl_{dv}}{2gd_d}\right)\left(\frac{A}{a_d}\omega r\sin\theta\right)^2$$

$$= \frac{0.032\times 6}{2\times 9.81\times 0.1}\left[\frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2}\times\left(\frac{2\pi\times 30}{60}\right)\times 0.2\sin\theta\right]^2$$

$$= 0.618 \sin^2\theta$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus by substitution, we have

$$H_{td} = 30 + 0.033 + 4.83 \cos\theta + 0.57 + 0.618 \sin^2\theta$$

or $$H_{td} = 30.603 + 4.83 \cos\theta + 0.618 \sin^2\theta$$

From Eq. 23.59, we have

$$(H_{td})_{absolute} = H_{atm} + H_{td}$$
$$= 10.3 + 30.603 + 4.83 \cos\theta + 0.618 \sin^2\theta$$
$$= 40.903 + 4.83 \cos\theta + 0.618 \sin^2\theta$$

(b) $$Q = \frac{ALN}{60}$$

$$= \frac{\pi}{4}(0.2)^2 \times 0.4 \times \left(\frac{30}{60}\right) = 0.006\,28 \text{ m}^3/\text{s}$$

From Eq. 23.69 the work done is given as

Work done $$= 9\,810 \times (0.006\,28)\,[(3.6 + 30) + (0.074 + 0.57) + (0.033 + 0.033) + \frac{2}{3}(0.206 + 0.618)$$

$$= 2148 \text{ N.m/s}$$

$\therefore$ Power required to drive the pump

$$= 2148 \text{ W} = 2.148 \text{ kW}.$$

(c) At $\theta = 0$

$$(H_{td})_{absolute} = (40.903 + 4.83) = 45.733 \text{ m}$$

From Eq. 23.61, we have

$$h_{dv} + H_{dv} = H_{td} + H_{atm} = (H_{td})_{absolute}$$

$\therefore$ $$h_{dv} = (H_{td})_{absolute} - H_{dv}$$
$$= (45.733 - 1.0) = 44.733 \text{ m of water}$$

Example 23.10. *A double acting pump has a bore of 0.2 m and stroke of 0.4 m. The suction pipe has a diameter of 0.1 m and is fitted with an air vessel. Find the rate of flow into or from the air vessel when the crank makes angles of 30°, 90° and 120° with inner dead centre. Determine also the crank angles at which there is no flow to or from the air vessel. Take the speed as 120 r.p.m., and assume that the plunger has simple harmonic motion.*

Solution

From Eq. 23.54 the rate of flow of liquid into the air vessel

$$= A\omega r\left[\sin\theta - \frac{2}{\pi}\right]$$

$$A = \frac{\pi}{4} \times (0.2)^2 = 0.0314 \text{ m}^2$$

$$r = \frac{0.4}{2} = 0.2 \text{ m}$$

$$\omega = \frac{2\pi N}{60} = \frac{2\pi \times 120}{60} = 12.566 \text{ rad/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Thus rate of flow

$$= (0.0314 \times 12.566 \times 0.2)\left[\sin\theta - \frac{2}{\pi}\right]$$

$$= 0.079\left[\sin\theta - \frac{2}{\pi}\right]$$

For $\theta = 30°$

The rate of flow $= 0.079\left[\sin 30° - \frac{2}{\pi}\right]$

$= -0.010\,8\ m^3/s$

Since this is negative and we are considering the air vessel fitted to the suction pipe, then as indicated earlier the flow is taking place into the air vessel.

For $\theta = 90°$

The rate of flow $= 0.079\left[\sin 90° - \frac{2}{\pi}\right]$

$= +0.0287\ m^3/s$

Since this is positive and we are considering the air vessel fitted to the suction pipe, then as indicated earlier the flow is taking place from the air vessel.

For $\theta = 120°$

The rate of flow $= 0.079\left[\sin 120° - \frac{2}{\pi}\right]$

$= +0.0181\ m^3/s$

Since this is positive, the flow is taking place from the air vessel.

For no flow from or into air vessel

$$0.079\left[\sin\theta - \frac{2}{\pi}\right] = 0; \text{ or } \sin\theta = 0.636\,6$$

$$\theta = 39° 32' \text{ and } 140° 28'$$

Example 23.11. *A double acting single cylinder reciprocating pump of 20 cm bore and 40 cm stroke runs at 35 r.p.m. The piston rod is 7.5 cm in diameter. The suction and delivery lifts are respectively 1 m and 30 m. The suction pipe is 2.5 m long and the delivery pipe is 40 m long and both of them are 10 cm in diameter. No air vessel is provided either on the suction or delivery pipe. The local losses can be neglected and for both pipes the friction factor can be taken as 0.03. The motion of the piston can be assumed to be simple harmonic.*

Determine the net force due to fluid pressure on the piston when it has moved through a distance of 10 cm from the inner dead centre.

Solution

In this case when the piston moves from the inner dead centre, on one side of the piston there is suction stroke and on the other side of the piston there is delivery stroke.

During suction stroke the absolute pressure head on the piston for any position of the crank, i.e., for any crank angle θ is given by Eq. 23.51 as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_{ts} = H_{atm} - H_s - \left(\frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r \cos\theta\right) - \frac{f l_s}{2 g d_s}\left(\frac{A}{a_s}\omega r \sin\theta\right)^2$$

$H_s = 1$ m; $A = \frac{\pi}{4}(0.20)^2$; $a_s = \frac{\pi}{4}(0.10)^2$; $\omega = \frac{2\pi \times 35}{60} = 3.67$ rad/s;

$r = \frac{40}{2} = 20$ cm $= 0.2$ m; $\cos\theta = \frac{20-10}{20} = \frac{1}{2}$, i.e., $\theta = 60°$;

$\sin\theta = \frac{\sqrt{3}}{2}$; $f = 0.03$; $l_s = 2.5$ m; $d_s = 10$ cm $= 0.1$ m

The atmospheric pressure is taken as 10.3 m of water.

Thus by substitution, we get

$$H_{ts} = 10.3 - 1 - \left[\frac{2.5}{9.81} \times \frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times (3.67)^2 \times 0.2 \times \frac{1}{2}\right]$$

$$-\left(\frac{0.03 \times 2.5}{2 \times 9.81 \times 0.10}\right)\left[\frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times 3.67 \times 0.2 \times \frac{\sqrt{3}}{2}\right]^2$$

or $H_{ts} = [10.3 - 1 - 1.373 - 0.247] = 7.68$ m

During delivery stroke the absolute pressure head on the piston for any position of the crank, i.e., for any crank angle θ is given by Eq. 23.52 as

$$H_{td} = H_{atm} + H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right) + \frac{f l_d}{2 g d_d}\left(\frac{A}{a_d} \times \omega r \sin\theta\right)^2$$

$H_d = 30$ m; $A = \frac{\pi}{4}(0.20)^2$; $a_d = \frac{\pi}{4}(0.10)^2$; $\omega = 3.67$ rad/s;

$r = 0.2$ m; $\cos\theta = \frac{1}{2}$; $\sin\theta = \frac{\sqrt{3}}{2}$; $f = 0.03$; $l_d = 40$ m;

$d_d = 0.1$ m ; and $H_{atm} = 10.3$ m

Thus by substitution, we get

$$H_{td} = 10.3 + 30 + \left[\frac{40}{9.81} \times \frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times (3.67)^2 \times 0.2 \times \frac{1}{2}\right]$$

$$+\left(\frac{0.03 \times 40}{2 \times 9.81 \times 0.10}\right)\left[\frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times 3.67 \times 0.2 \times \frac{\sqrt{3}}{2}\right]^2$$

$= [10.3 + 30 + 21.968 + 3.954] = 66.222$ m

Total pressure acting on the piston on the side in which suction stroke is taking place is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$P_1 = wH_{ts} \times A$$

$$= 9810 \times 7.68 \times \frac{\pi}{4} \times (0.20)^2 = 2367 \text{ N}$$

Total pressure acting on the piston on the side in which delivery stroke is taking place is

$$P_2 = wH_{td} \times (A - A_p)$$

$$= 9810 \times 66.222 \times \left[\frac{\pi}{4} \times (0.20)^2 - \frac{\pi}{4} \times (0.075)^2\right]$$

$$= 17539 \text{ N}$$

The net force due to fluid pressure acting on the piston is

$$P = (P_2 - P_1)$$

$$= (17539 - 2367) = 15172 \text{ N}$$

Example 23.12. *A single acting reciprocating pump works against a total static lift of 105 m of which 5 m is suction static lift, and when running at 30 r.p.m, it is required to force 45 litres per second of water along a delivery pipe, 25 cm diameter, 110 m long. There are no air vessels in the pump. Darcy's friction factor for the pipe is 0.024. The slip is 3%. The stroke of the plunger is twice the diameter. What would be the diameter of the cylinder and maximum and minimum pressures in the pipe when (i) one cylinder is used; and (ii) two cylinders are used in parallel.*

Solution

(i) *When one cylinder is used*

$$Q_{th} = \frac{ALN}{60}$$

Let D be the diameter of the cylinder, then

$$A = \frac{\pi D^2}{4}; L = 2D \text{ ; and } N = 30 \text{ r.p.m}$$

$$\therefore \quad Q_{th} = \frac{(\pi/4)D^2 \times 2D \times 30}{60} = \frac{\pi D^3}{4}$$

$$Q_a = 45 \text{ } l/\text{s} = 0.045 \text{ m}^3/\text{s}$$

Percentage slip $= \dfrac{Q_{th} - Q_a}{Q_{th}} \times 100$

or $3 = \dfrac{Q_{th} - Q_a}{Q_{th}} \times 100$

or $0.97\, Q_{th} = Q_a$

or $0.97 \times \left(\dfrac{\pi D^3}{4}\right) = 0.045$

or $D^3 = 0.05907$

$\therefore \quad D = 0.39 \text{ m} = 39 \text{ cm}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

During delivery stroke the pressure head developed in the delivery pipe for any position of the crank, i.e., for any crank angle θ is given by the equation

$$H_{td} = H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right) + \frac{f l_d}{2 g d_d}\left(\frac{A}{a_d}\omega r \sin\theta\right)^2$$

$$H_d = 105 - 5 = 100 \text{ m}; \; l_d = 110 \text{ m}; \; A = \frac{\pi}{4} \times (0.39)^2;$$

$$a_d = \frac{\pi}{4} \times (0.25)^2; \; \omega = \frac{2\pi N}{60} = \frac{2 \times \pi \times 30}{60} = \pi \text{ rad/s};$$

$$r = \frac{L}{2} = \frac{2D}{2} = D = 0.39 \text{ m}; \; f = 0.024$$

Thus by substitution, we get

$$H_{td} = 100 + \left[\frac{110}{9.81} \times \frac{(\pi/4) \times (0.39)^2}{(\pi/4) \times (0.25)^2} \times (\pi)^2 \times 0.39 \cos\theta\right]$$

$$+ \left[\frac{0.024 \times 110}{2 \times 9.81 \times 0.25} \times \left\{\frac{(\pi/4) \times (0.39)^2}{(\pi/4) \times (0.25)^2} \times \pi \times 0.39 \sin\theta\right\}^2\right]$$

The maximum pressure will occur at the beginning of the delivery stroke for which θ = 0. Thus, we have

$$(H_{td})_{max} = (100 + 105.04 + 0) \text{ m}$$
$$= 205.04 \text{ m}$$
$$= 205.04 \times 9.81 \text{ kN/m}^2 = 2011.44 \text{ kN/m}^2$$

The minimum pressure will occur at the end of the delivery stroke for which θ = 180°. Thus, we have

$$(H_{td})_{min} = (100 - 105.04 - 0)$$
$$= -5.04 \text{ m}$$
$$= -5.04 \times 9.81 \text{ kN/m}^2 = -49.44 \text{ kN/m}^2$$

(ii) *When two cylinders are used in parallel*

$$Q_{th} = \frac{2ALN}{60}$$

Let D be the diameter of each cylinder, then

$$A = \frac{\pi D^2}{4}; \quad L = 2D; \text{ and } N = 30 \text{ r.p.m.}$$

$$\therefore \quad Q_{th} = \frac{2 \times (\pi/4) \times D^2 \times 2D \times 30}{60} = \frac{\pi D^3}{2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q_a = 45\ l/\text{s} = 0.045\ \text{m}^3/\text{s}$$

$$\text{Percentage slip} = \frac{Q_{\text{th}} - Q_a}{Q_{\text{th}}} \times 100$$

or $$3 = \frac{Q_{\text{th}} - Q_a}{Q_{\text{th}}} \times 100$$

or $$0.97 \times Q_{\text{th}} = Q_a$$

or $$0.97 \times \frac{\pi D^3}{2} = 0.045$$

or $$D^3 = 0.0295$$

$\therefore$ $$D = 0.31\ \text{m} = 31\ \text{cm}$$

During delivery stroke the pressure head developed in the delivery pipe for any position of the crank, i.e., for any crank angle θ is given by the equation

$$H_{td} = H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta \right) + \frac{f l_d}{2 g d_d} \left(\frac{A}{a_d} \omega r \sin\theta \right)^2$$

Thus by substitution, we get

$$H_{td} = 100 + \left[\frac{110}{9.81} \times \frac{(\pi/4) \times (0.31)^2}{(\pi/4) \times (0.25)^2} \times (\pi)^2 \times 0.31 \cos\theta \right]$$

$$+ \left[\frac{0.024 \times 110}{2 \times 9.81 \times 0.25} \times \left\{ \frac{(\pi/4) \times (0.31)^2}{(\pi/4) \times (0.25)^2} \times \pi \times 0.31 \sin\theta \right\}^2 \right]$$

The maximum pressure will occur at the beginning of the delivery stroke for which $\theta = 0$. Thus, we have

$$(H_{td}) = (100 + 52.75 + 0)\ \text{m}$$
$$= 152.75\ \text{m}$$
$$= 152.75 \times 9.81\ \text{kN/m}^2 = 1498.48\ \text{kN/m}^2$$

The minimum pressure will occur at the end of the delivery stroke for which $\theta = 180°$. Thus, we have

$$(H_{td})_{\min} = (100 - 52.75 - 0)\ \text{m}$$
$$= 47.25\ \text{m}$$
$$= 47.25 \times 9.81\ \text{kN/m}^2 = 463.52\ \text{kN/m}^2$$

It may be noted that when one cylinder is used, negative or vacuum pressure is developed at the end of the delivery stroke. On the other hand when two cylinders are used in parallel, no negative or vacuum pressure is developed. In other words by using two cylinders development of negative or vacuum pressure is avoided.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. A reciprocating pump is a positive displacement pump which consists of a piston or plunger which moves to and fro in a close fitting cylinder which is connected to suction and delivery pipes each of which is provided with a non-return or one way value called suction valve and delivery valve respectively.
2. Theoretical discharge of the pump is given as
   (a) For single acting pump

$$Q_{th} = \frac{ALN}{60}$$

   (b) For a double acting pump

$$Q_{th} = \frac{(2A - A_p)LN}{60}$$

   where
   A = cross-sectional area of the piston;
   A_p = area of cross-section of the piston rod;
   L = length of the stroke; and
   N = number of delivery strokes per minute.
3. Theoretical power required to drive the pump is given by
   (a) For a single acting pump

$$P = \frac{W(ALN)(H_s + H_d)}{60}$$

   (b) For a double acting pump

$$P = \frac{w(2A - A_p)LN(H_s + H_d)}{60}$$

   where
   H_s = suction head;
   H_d = delivery head; and
   w = specific weight of the liquid.
4. The difference between the theoretical discharge and the actual discharge is known as the ship of the pump. That is

$$\text{Slip} = (Q_{th} - Q_a) \text{ ; and}$$

$$\text{Percentage slip} = \frac{(Q_{th} - Q_a)}{Q_{th}} \times 100$$

   In some cases the actual discharge of the pump may be more than the theoretical discharge, in which case the coefficient discharge C_d will be more than one and the slip will be negative.
5. The pressure head H_a due to acceleration of liquid in the suction and delivery pipes is given by

$$H_{as} = \frac{l_s}{g}\frac{A}{a_s}(\omega^2 r)\cos\theta$$

$$H_{ad} = \frac{l_d}{g}\frac{A}{a_d}(\omega^2 r)\cos\theta$$

6. The loss of head due to friction in suction and delivery pipes is given by

$$h_{f_s} = \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r \sin\theta\right)$$

$$h_{f_d} = \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r \sin\theta\right)$$

7. Mean velocity of flow is given by
   (a) For a single acting pump

$$V_d = \frac{A}{a_d}\frac{\omega r}{\pi}$$

   (b) For a double acting pump

$$V_d = 2\frac{A}{\alpha_d}\frac{\omega r}{\pi}$$

8. Indicator diagram is a graph between the pressure head in the cylinder and the distance travelled by the piston from the inner dead centre for one complete revolution of the crank.
9. The work done by the pump is proportional to the area of the indicator diagram. Area of ideal indicater diagram is same as the area of the indicator diagram modified due to acceleration of liquid in the suction and delivery pipes.
10. Work done by the pump per second is given by
    (a) For a single acting pump
    Work done =

$$\frac{wALN}{60}\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right]$$

    (b) For a double acting pump

$$\text{Work done} = \frac{\omega(2A - A_p)LN}{60}\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

11. An air vessel is used to obtain a continuous supply of liquid at a uniform rate thereby save a considerable amount of work and run the pump at a high speed without separation.
12. Work done by single acting reciprocating pump fitted with air vessels is given by

$$\text{Work done} = \frac{wALN}{60}\left[H_s + H_d + h_{f_s} + h_{f_d}\right]$$

13. Work saved by fitting air vessels in a single acting reciprocating pump is 84.8% while in a double acting reciprocating pum the work saved is 39.2%.

PROBLEMS

23.1 Why is a reciprocating pump called positive displacement pump?

23.2 Explain the term 'slip' and 'cavitation' with reference to reciprocating pumps.

23.3 Compare discharge curves for single and double acting reciprocating pumps and indicate under what conditions are either type used.

23.4 Explain the working principles of reciprocating pump with sketches.

23.5 Discuss the conditions under which 'cavitation' and 'negative slip' occur and state why air vessels are used in reciprocating pumps.

23.6 Explain with a neat sketch of its main parts, and a diagram of velocity *v/s* stroke, the working of a three throw pump. Is an air vessel necessary for this type of pump? Explain your answer.

23.7 The work saved by fitting an air vessel on a single acting reciprocating pump is much more than in a double acting reciprocating pump. Why?

23.8 Explain the functions of air vessels in a reciprocating pump.

23.9 Show that the maximum inertia head in a reciprocating pump without air vessel is given by

$$H_a = \frac{l}{g} \times \frac{A}{a} \omega^2 r$$

with usual notation.

23.10 What is an indicator diagram of a reciprocating pump? Sketch the theoretical indicator diagram for a single-acting reciprocating pump not fitted with an air vessel. With the help of the diagram explain clearly the effect of acceleration and friction on both suction and delivery strokes. How is this modified if air vessels are provided on both suction and delivery pipes?

23.11 Working from first principles show that, in a single acting pump the work done against pipe friction, if large air vessels are fitted near to the cylinder on both suction and delivery pipes, compared with that done if there are no vessels is. $(3/2\ \pi^2)$.

23.12 A single acting reciprocating pump has a plunger diameter 200 mm and stroke length 300 mm. It is to be driven with S.H.M., at 60 r.p.m., and draws water from a sump 3.6 m below the pump centreline. Find the least diameter of suction pipe, which is 4.5 m long in order to prevent separation at this speed. Take atmospheric pressure as 10.3 m of water, and the separation occurs at 2.6 m of water absolute. [*Ans.* 163 mm]

23.13 A single acting pump has a piston diameter of 0.15 m and a crank radius of 0.30 m. The delivery pipe is 0.10 m in diameter and 30 m long. The water is lifted 20 m above the centre of the pump. Find the maximum speed at which the pump can be run so that no cavitation takes place during the delivery stroke. Take H_{atm} = 10.3 m of water, and separation occurs at 2.6 m of water absolute. [*Ans.* 49.5 r.p.m.]

23.14 A single acting reciprocating pump has a plunger of diameter 0.3 m and a stroke of length 0.4 m. If the speed of the pump if 60 r.p.m., and coefficient of discharge is 0.97, determine the percentage slip and actual discharge of the pump. [*Ans.* 3%; 27.43 *l*/s]

23.15 A single acting reciprocating pump is to raise a liquid of specific weight 11.77 kN/m^3 through a vertical height of 11.5 m, from 2.5 m below the pump axis to 9 m above it. The plunger moves with S.H.M., and has a diameter 125 mm and stroke length 225 mm. The suction and delivery pipes are 75 mm diameter, and 3.5 m and 13.5 m long respectively. There is a large air vessel placed on the delivery pipe near the pump axis but there is no air vessel on the suction pipe. If separation takes place at 88.3 kN/m^2 below the atmospheric pressure, find (a) the maximum speed at which the pump can run without separation taking place; (b) the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

power required to drive the pump if $f = 0.02$. Neglect slip for the pump.

[*Ans.* 64 r.p.m; 0.406 kW]

23.16 A double acting reciprocating pump has a bore of 150 mm and stroke of 300 mm. The suction pipe has a diameter of 100 mm and is fitted with an air vessel. Find the rate of flow into or from the air vessel when the crank makes angles of 30°, 90° and 120° with inner dead centre. The speed is 120 r.p.m., and the plunger has simple harmonic motion.

[*Ans.* 4.55 *l*/s (into air vessel); 12.10 *l*/s (from air vessel); 7.64 *l*/s (from air vessel)]

23.17 If in the Problems 23.16 the delivery pipe has a diameter of 100 mm and it is also fitted with an air vessel, then find the rate of flow into or from the air vessel when the crank makes angles of 30°, 90° and 120° with the outer dead centre and the speed of the pump is 120 r.p.m. The piston moves with S.H.M.

[*Ans.* 4.55 l/s (from air vessel); 12.10 l/s (into air vessel); 7.64 *l*/s(into air vessel)]

23.18 A single acting reciprocating pump runs at 60 r.p.m. The diameter of the plunger is 0.15 m and crank radius is 0.15 m. The suction pipe is 0.1 m in diameter and 5 m long. Calculate the maximum permissible value of suction lift H_s if the separation takes place at 2.6 m of water absolute. [*Ans.* 0.91 m]

23.19 For a single acting reciprocating pump stroke length = 0.2 m, piston diameter = 0.1 m, suction head = 5 m, delivery head = 32 m, diameter of suction and delivery pipes = 50 mm, length of suction pipe = 6 m; length of delivery pipe = 35 m. Darcy's friction factor f for both pipes = 0.02. If the pump is working at 30 r.p.m., find the pressure head in metres of water on the piston at the beginning, middle and end of both suction and delivery strokes. Also find the power required to drive the pump. State if separation may take place at any point.

[*Ans.* 7.41 m, 5.19 m, 2.59 m of water (vacuum) (Suction) 46.09 m, 33.13 m,17.92 m of water (gage) (Delivery); 0.292 kW; No possibility of separation]

23.20 If a single acting pump having diameter of plunger 150 mm and stroke length 225 mm, has for the suction side $d_s = 75$ mm, $l_s = 6$ m, $H_s = 3$ m, and for the delivery side $d_d = 75$ mm, $l_d = 60$ m, $H_d = 27$ m. Find the maximum possible speed at which separation of water could be avoided. Hence find the maximum theoretical discharge at this speed.

[*Ans.* 34 r.p.m.; 2.25 *l*/s]

23.21 A single acting reciprocating pump has a plunger 375 mm diameter and stroke length 600 mm. The delivery pipe is 90 m long and 150 mm diameter. Find the power saved by installing an air vessel near the pump, if the pump runs at 50 r.p.m. Take $f = 0.032$.

[*Ans.* 28.92 kW]

23.22 A single acting reciprocating pump running at 60 r.p.m., has a plunger diameter of 0.25 m and stroke length of 0.5 m. The delivery pipe is 0.1 m diameter and 50 m long. If the motion of the piston is simple harmonic, find the power required to overcome friction of the delivery pipe, when (a) no air vessel is fitted, (b) a large air vessel is fitted at the centreline of the pump.Take $f = 0.04$.

[*Ans.* (a) 15.77 kW; (b) 2.4 kW]

23.23 (a) Derive an expression for pressure head due to acceleration of the piston of a reciprocating pump, assuming motion of the piston to be S.H.M.

(b) Find the power required to drive a double acting pump and the quantity of water delivered per hour under the following conditions: head of water excluding friction = 16 m; diameter of main =1 m; length of main = 2000 m; diameter of cylinder = 0.75 m; speed of pump = 24 r.p.m.; length of stroke = 1.5 m; positive slip = 4%; mechanical efficiency = 80%. Assume $f = 0.01$.

[*Ans.* 104.62 kW; 1832 m^3/hr]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Centrifugal Pumps

Chapter 24

24.1 INTRODUCTION

Centrifugal pumps are classified as rotodynamic type of pumps in which a dynamic pressure is developed which enables the lifting of liquids from a lower to a higher level. The basic principle on which a centrifugal pump works is that when a certain mass of liquid is made to rotate by an external force, it is thrown away from the central axis of rotation and a centrifugal head is impressed which enables it to rise to a higher level. Now if more liquid is constantly made available at the centre of rotation, a continuous supply of liquid at a higher level may be ensured. Since in these pumps the lifting of the liquid is due to centrifugal action, these pumps are called 'centrifugal pumps'. In addition to the centrifugal action, as the liquid passes through the revolving wheel or impeller, its angular momentum changes, which also results in increasing the pressure of the liquid. As such centrifugal pumps behave quite differently from positive displacement pumps. A centrifugal pump does not push the liquid as in the case of a positive displacement pump, but it modifies the hydraulic gradient such that the liquid is lifted to a higher level.

According to the general direction of flow of liquid within the passage of the rotating wheel or impeller the rotodynamic pumps are classified as,

(i) Centrifugal pumps,

(ii) Half axial or screw or mixed flow pumps,

(iii) Axial flow or propeller pumps.

In the impeller of a centrifugal pump the liquid flows in the outward radial direction, while the flow of liquid in a propeller pump impeller is in the axial direction, parallel to the rotating shaft. The mixed flow pump impeller has an intermediate form so that the flow of liquid is in between the radial and axial directions. However, there are no rigid boundaries separating these three types of pumps, and often all the three types of pumps are called centrifugal pumps.

In general all the rotodynamic pumps closely resemble reaction type of hydraulic turbines and they may be regarded as reversed reaction turbines. Thus the action of a centrifugal pump is just the reverse of a radially inward flow reaction turbine. Similarly the axial flow pumps are reverse of propeller or Kaplan turbines and the mixed flow pumps are the reverse of mixed flow type turbines such as Francis turbine. In the present chapter only centrifugal pumps have been described.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

24.2 ADVANTAGES OF CENTRIFUGAL PUMPS OVER RECIPROCATING PUMPS

The main advantage of a centrifugal pump is that its discharging capacity is very much greater than that of a reciprocating pump which can handle relatively small quantity of liquid only. A centrifugal pump can be used for lifting highly viscous liquids such as oils, muddy and sewage water, paper pulp, sugar molasses, chemicals etc. But a reciprocating pump can handle only pure water or less viscous liquids free from impurities as otherwise its valves may cause frequent trouble. A centrifugal pump can be operated at very high speeds without any danger of separation and cavitation. As such it can be coupled directly through flanged coupling to electric motor. On the other hand as explained in Chapter 23 the maximum speed of a reciprocating pump is limited from the considerations of separation and cavitation. As such reciprocating pumps can be operated at low speeds only and for that these pumps are mostly belt driven. The maintenance cost of a centrifugal pump is low and only periodical check up is sufficient. But for a reciprocating pump the maintenance cost is high because the parts such as valves etc. may need frequent replacement. However, a reciprocating pump can build up very high pressures as high as 69×10^6 N/m^2 [700 kg(f)/cm^2] or even more and hence these pumps are used for lifting oil from very deep oil wells.

24.3 COMPONENT PARTS OF A CENTRIFUGAL PUMP

Figure 24.1 shows the main component parts of a centrifugal pump which are described below:

1. Impeller. It is a wheel or rotor which is provided with a series of backward curved blades or vanes. It is mounted on a shaft which is coupled to an external source of energy (usually an electric motor) which imparts the required energy to the impeller thereby making it to rotate.

The impellers may be classified as (a) closed or shrouded impeller, (b) semi-open impeller; and (c) open impeller, which are all shown in Fig. 24.1. A 'closed or shrouded impeller' is that whose vanes are provided with metal cover plates or shrouds on both sides. These plates or shrouds are known as crown plate and lower or base plate as shown in Fig. 24.1. The closed impeller provides better guidance for the liquid and is more efficient. However, this type of impeller is most suited when the liquid to be pumped is pure and comparatively free from debris.

If the vanes have only the base plate and no crown plate, then the impeller is known as 'semi-open type impeller'. Such an impeller is suitable even if the liquids are charged with some debris.

An 'open impeller' is that whose vanes have neither the crown plate nor the base plate. Such impellers are useful in the pumping of liquids containing suspended solid matter, such as paper pulp, sewage and water containing sand or grit. These impellers are less liable to clog when handling liquids charged with a large quantity of debris.

2. Casing. It is an airtight chamber which surrounds the impeller. It is similar to the casing of a reaction turbine. The different types of casings that are commonly adopted are described later.

3. Suction Pipe. It is a pipe which is connected at its upper end to the inlet of the pump or to the centre of the impeller which is commonly known as eye. The lower end of the suction pipe dips into liquid in a suction tank or a sump from which the liquid is to be pumped or lifted up.

The lower end of the suction pipe is fitted with a *foot valve and strainer*. The liquid first enters the strainer which is provided in order to keep the debris (such as leaves, wooden pieces and other rubbish) away from the pump. It then passes through the foot valve to enter the suction pipe. A 'foot valve' is a non-return or one-way type of valve which opens only in the upward direction. As such the liquid will pass through the foot valve only upwards and it will not allow the liquid to move downwards back to the sump.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

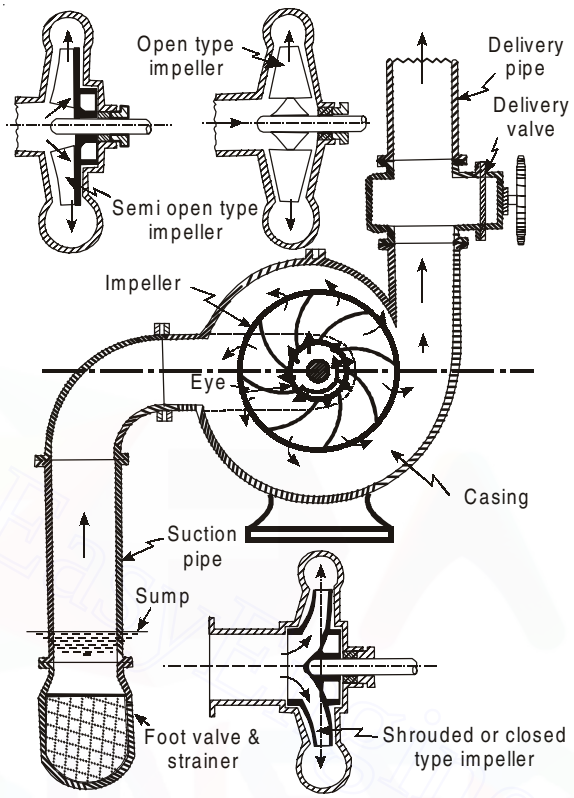


Figure 24.1 *Component parts of a centrifugal pump*

4. Delivery Pipe. It is a pipe which is connected at its lower end to the outlet of the pump and it delivers the liquid to the required height. Just near the outlet of the pump on the delivery pipe a delivery valve is invariably provided. A delivery valve is a regulating valve which is of sluice type and is required to be provided in order to control the flow from the pump into delivery pipe.

24.4 WORKING OF CENTRIFUGAL PUMP

The first-step in the operation of a centrifugal pump is priming. *Priming* is the operation in which the suction pipe, casing of the pump and the portion of the delivery pipe upto the delivery valve are completely filled with the liquid which is to be pumped, so that all the air (or gas or vapour) from this portion of the pump is driven out and no air pocket is left. It has been observed that even the presence of a small air pocket in any of the portion of pump may result in no delivery of liquid from the pump. The necessity of priming a centrifugal pump is due to the fact that the pressure generated in a centrifugal pump impeller is directly proportional to the density of the fluid that is in contact with it. Hence if an impeller is made to rotate in the presence of air, only a negligible pressure would be produced with the result that no liquid will be lifted up by the pump. As such it is essential to properly prime a centrifugal pump before it can be started. The various methods used for priming a centrifugal pump are discussed later.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

After the pump is primed, the delivery valve is still kept closed and the electric motor is started to rotate the impeller. The delivery valve is kept closed in order to reduce the starting torque for the motor. The rotation of the impeller in the casing full of liquid produces a forced vortex which imparts a centrifugal head to the liquid and thus results in an increase of pressure throughout the liquid mass. The increase of pressure at any point is proportional to the square of the angular velocity and the distance of the point from the axis of rotation. Thus if the speed of rotation of the impeller of the pump is sufficiently high, the pressure in the liquid surrounding the impeller is considerably increased. Now as long the delivery valve is closed and the impeller is rotating, it just churns the liquid in the casing. When the delivery valve is opened the liquid is made to flow in an outward radial direction thereby leaving the vanes of the impeller at the outer circumference with high velocity and pressure. At the eye of the impeller due to the centrifugal action a partial vacuum is created. This causes the liquid from the sump, which is at atmospheric pressure, to rush through the suction pipe to the eye of the impeller thereby replacing the liquid which is being discharged from the entire circumference of the impeller. The high pressure of the liquid leaving the impeller is utilized in lifting the liquid to the required height through the delivery pipe.

As the liquid flows through the rotating impeller it receives energy from the vanes which results in an increase in both pressure and velocity energy. As such the liquid leaves the impeller with a high absolute velocity. In order that the kinetic energy corresponding to the high velocity of the leaving liquid is not wasted in eddies and efficiency of the pump thereby lowered, it is essential that this high velocity of the leaving liquid is gradually reduced to a lower velocity of the delivery pipe, so that the larger portion of the kinetic energy is converted into useful pressure energy. Usually this is achieved by shaping the casing such that the leaving liquid flows through a passage of gradually expanded area. The gradually increased cross-sectional area of the casing also helps in maintaining uniform velocity of flow throughout, because as the flow proceeds from the tongue *T* (Fig. 24.2) to the delivery pipe, more and more liquid is added from the impeller. There are different types of casings that are adopted for this purpose and on the basis of the type of casing used, the centrifugal pumps are classified into different types as described in the next section.

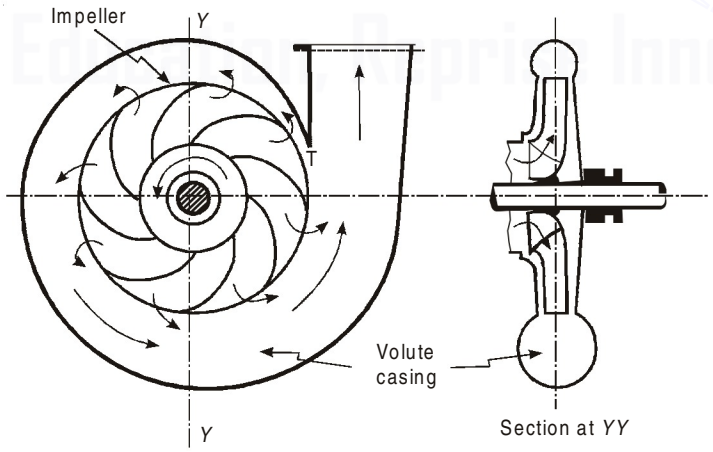


Figure 24.2 *Volute pump*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

24.5 TYPES OF CENTRIFUGAL PUMPS

According to the type of casing provided, centrifugal pumps are classified into the following two classes:

(1) Volute pump.

(2) Diffuser or turbine pump.

1. Volute Pump. In a volute pump the impeller is surrounded by a spiral shaped casing which is known as *volute chamber*. As shown in Fig. 24.2 the shape of the casing is such that the sectional area of flow around the periphery of the impeller gradually increases from the tongue *T* towards the delivery pipe. This increase in the cross-sectional area results in developing a uniform velocity throughout the casing, because as the flow progresses from the tongue *T* towards the delivery pipe, more and more liquid is added to the stream from the periphery of the impeller. The volute casing may be designed to have the velocity of flow approximately equal to that of the liquid leaving the impeller. If the casing is designed according to this consideration then the loss of energy is considerably reduced, but the conversion of kinetic energy into useful pressure energy will not be possible. On the other hand if the casing is so designed that the casing velocity may be kept down to the value of the velocity in the delivery pipe, then there will be considerable loss of energy due to the difference between the casing velocity and that of the liquid discharged from the impeller. As such a compromise design is often used in which the casing is gradually enlarged so that the velocity is gradually reduced, from the velocity of the liquid leaving the impeller to that in the delivery pipe.

However, in actual practice it has been found that in volute type of casing there is only a slight increase in the efficiency of the pump, because a considerable loss of energy takes place in eddies developed in the casing.

A subsequent improvement over the simple volute casing was made by J. Thomson by prodiving a circular chamber between the impeller and the volute chamber as shown in Fig. 24.3. The circular chamber is known as *vortex* or *whirlpool chamber* and such a pump is known as *volute pump with vortex chamber*.

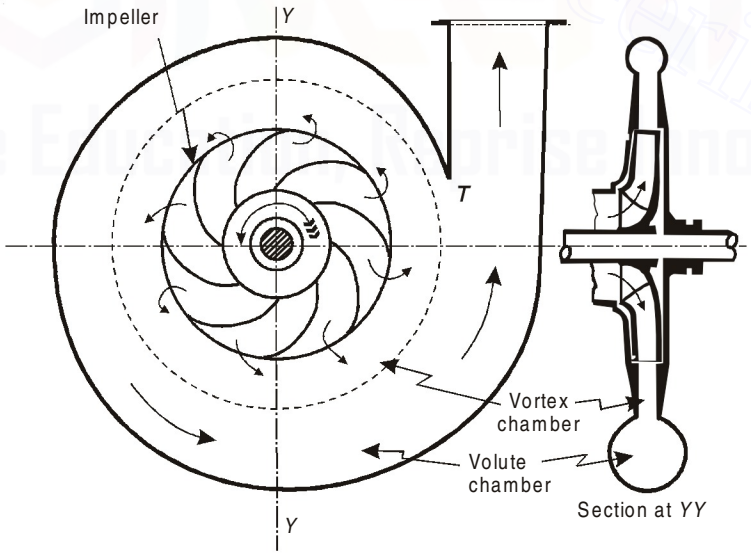


Figure 24.3 *Volute pump with vortex chamber*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The vortex chamber is usually formed as a part of the casing with its side walls parallel or nearly parallel as shown in Fig. 24.3. It acts as a diffuser wherein the conversion of kinetic energy into pressure energy takes place as explained below. The liquid after leaving the impeller enters the vortex chamber with a whirling motion, that is the liquid particles move radially away from the centre following a rotary path while passing through this chamber. Since no work is done on the liquid as it passes through this chamber, its energy remains constant (except for the slight loss by friction). Therefore the torque produced for the liquid does not change and hence a free vortex is formed as the liquid passes through the vortex chamber. Since for a free vortex the velocity of whirl varies inversely as its radial distance from the centre, there is a reduction in velocity of flow of liquid as it passes through the vortex chamber. The reduction in velocity is accompanied by an increase in pressure. As such a vortex chamber serves a dual purpose of reducing the velocity and increasing the efficiency of the pump by converting a large amount of kinetic energy into pressure energy. The liquid after leaving the vortex chamber passes through the volute chamber surrounding it, which further increases the efficiency of the pump.

2. Diffuser or Turbine Pump. In the diffuser pump, the impeller is surrounded by a series of guide vanes mounted on a ring called diffuser ring as shown in Fig. 24.4. The diffuser ring and the guide vanes are fixed in position. The adjacent guide vanes provide gradually enlarged passages for the flow of liquid. The liquid after leaving the impeller passes through these passages of increasing area, wherein the velocity of flow decreases and the pressure increases. The guide vanes are so designed that the liquid emerging from the impeller enters these passages without shock. This condition may however be achieved by making the tangent to the guide vane at the inlet tip to coincide with the direction of the absolute velocity of liquid leaving the impeller. After passing through the guide vanes the liquid flows into the surrounding casing which may be circular, and concentric with the impeller or it may be volute-shaped like that of volute pump. However, the common practice is to adopt circular casings for these pumps.

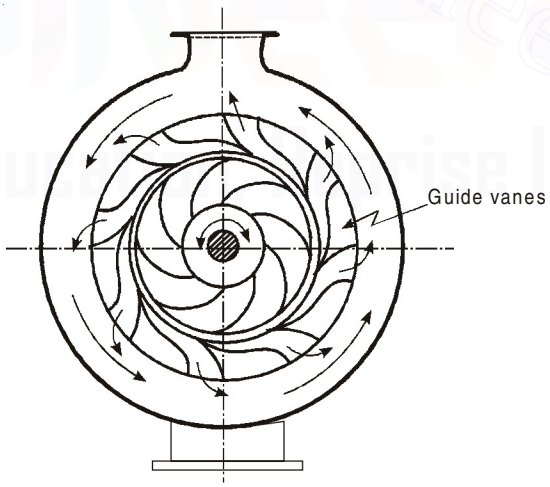


Figure 24.4 *Diffuser (or turbine) pump*

These pumps which are provided with diffuser ring and guide vanes very much resemble a reversed turbine and hence they are also known as *turbine pumps*. It has been found from tests that a well

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

designed diffuser pump is capable of converting as much as 75 percent of the kinetic energy of the liquid discharged from the impeller into pressure energy. However these pumps will work with maximum efficiency only for one rate of discharge at given impeller speed. This is so because the guide vanes will be correctly set or shaped for one rate of discharge only and for other discharges a loss of energy by shock or turbulence will occur at the entrance to the guide vanes, thereby resulting in a low efficiency. Moreover turbine pumps are more costly than the simple volute pumps. As such the arrangement of diffuser ring is usually employed only in multistage pumps.

The centrifugal pumps may also be classified on the basis of certain other factors as indicated below:

(a) Number of impellers per shaft.

(b) Relative direction of flow of liquid through impeller.

(c) Number of entrances to the impeller.

(d) Disposition of shaft, and

(e) Working head.

According to the number of impellers provided the pumps may be classified as single-stage and multi-stage. A *single stage centrifugal pump* has only one impeller mounted on the shaft. A *multi-stage centrifugal pump* has two or more impellers connected in series, which are mounted on the same shaft and are enclosed in the same casing.

On the basis of the direction of flow of the liquid through the impeller the pump may be classified as radial flow pump, mixed flow pump and axial flow pump.

A *radial flow pump* is that in which the liquid flows through the impeller in the radial direction only. Ordinarily all the centrifugal pumps are provided with radial flow impellers. In *mixed flow pumps* the liquid flows through the impeller axially as well as radially, that is there is a combination of radial and axial flows. A mixed flow impeller is just a modification of radial flow type in this respect that the former is capable of discharging a large quantity of liquid. As such mixed flow pumps are generally used where a large quantity of liquid is to be dicharged to low heights. In *axial flow pumps* the flow of liquid through the impeller is in the axial direction only. Axial flow pumps are usually designed to deliver very large quantities of liquid at relatively low heads. However, it is not justified to call axial flow pumps as centrifugal pumps, because there is hardly any centrifugal action in their operation.

Depending on the number of entrances to the impeller the centrifugal pumps may be classified as single suction pump and double suction pump. In a *single suction* (*or entry*) *pump* liquid is admitted from a suction pipe on one side of the impeller. In a *double suction* (*or entry*) *pump* liquid enters from both sides of the impeller. A double suction pump has an advantage that by this arrangement the axial thrust on the impeller is neutralised. Further it is suitable for pumping large quantities of liquid since it provides a large inlet area.

The centrifugal pumps may be designed with either horizontal or vertical disposition of shafts. Generally the pumps are provided with horizontal shafts. However, for deep wells and mines the pumps with vertical shafts are more suitable because the pumps with vertically disposed shafts occupy less space.

According to the head developed, the centrifugal pumps may be classified as *low head*, *medium head* and *high head* pumps. A *low head* pump is the one which is capable of working against a total head upto 15 m. A *medium head* pump is that which is capable of working against a total head more than 15 m but upto 40 m. A *high head* pump is the one which is capable of working against a total head above 40 m. Generally high head pumps are multi-stage pumps.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

24.6 WORK DONE BY THE IMPELLER

The expression for the work done (or the energy supplied) by the impeller of a centrifugal pump on the liquid flowing through it may be derived in the same way as for a turbine. The liquid enters the impeller at its centre and leaves at its outer periphery. Figure 24.5 shows a portion of the impeller of a Centrifugal pump with one vane and the velocity triangles at the inlet and the outlet tips of the vane. For the sake of convenience the same system of notation is employed as that for turbines. Thus, V is absolute velocity of liquid, u is peripheral (or tangential) velocity of the impeller V_r is relative velocity of liquid, V_f is velocity of flow of liquid, and V_w is velocity of whirl of the liquid at the entrance to the impeller. Similarly V_1, u_1 V_{r1}, V_{f1} and V_{w1} represent their counterparts at the exit point of the impeller. Further θ represents the impeller vane angle at the entrance and ϕ represents the impeller vane angle at the outlet. Similarly α is the angle between the direction of the absolute velocity of entering liquid and the peripheral velocity of the impeller at the entrance, and β is the angle between the absolute velocity of leaving liquid and the peripheral velocity of the impeller at the exit point.

At the entrance to the impeller since there are no guide vanes (as in the case of turbines) the direction of the absolute velocity of liquid at this point of the impeller is not directly known. However, for best efficiency of the pump it is commonly assumed that the liquid enters the impeller radially that is the absolute velocity of the liquid at the entrance to the impeller (or at the inlet tip of the impeller vane) is radial in direction. Thus, in this case $\alpha = 90°$ and the velocity of whirl V_w at inlet is equal to zero. Further it is desired that the liquid enters and leaves the vane without shock. This can be ensured if the inlet and outlet tips of the vane are parallel to the direction of the relative velocities at the two tips. As such it is assumed that the relative velocities V_r and V_{r1} are parallel to the tangents to the vane at the inlet and outlet tips respectively as shown in Fig. 24.5.

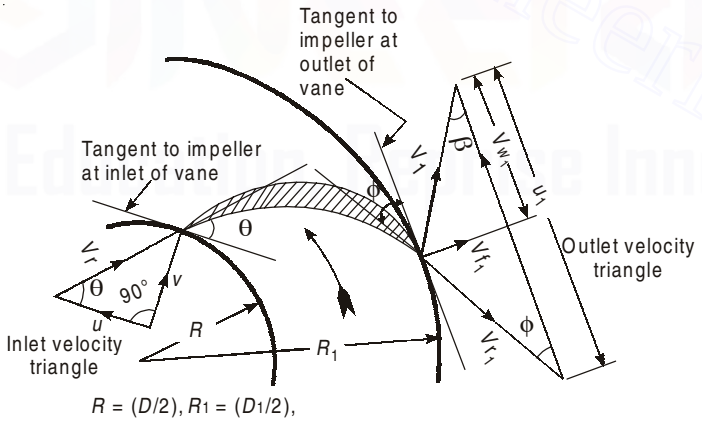


Figure 24.5 *Velocity triangles for an impeller vane*

In the case of a radially inward flow reaction turbine the work done per second by the liquid on the runner = (W/g) $(V_w u \pm V_{w1} u_1)$; where W is weight of liquid striking the runner per second. Since a centrifugal pump is just the reverse of a radially inward flow reaction turbine, the same analysis as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

used for turbines may be applied to pumps. Thus, the work done per second by the impeller on the liquid may be written as

$$\text{Work done} = \frac{W}{g}(V_{w1}u_1 - V_w u) \qquad \ldots(24.1)$$

where W is the weight of liquid per second that passes through the impeller. Since in this case as stated earlier the liquid enters the impeller radially, $\alpha = 90°$ and hence $V_w = 0$.

Thus Eq. 24.1 becomes

$$\text{Work done} = \frac{W}{g}(V_{w1}u_1) \qquad \ldots(24.2)$$

and work done per unit weight of liquid

$$= \frac{1}{g}(V_{w1}u_1) \qquad \ldots(24.3)$$

Thus Eq. 24.3 represents the head imparted by the impeller to the liquid.

Further from the outlet velocity triangle of Fig. 24.5, $V_{w1} = (u_1 - V_{f1}\cot\phi)$. As in the case of turbines u_1 and V_{f1} can be expressed in terms of the speed ratio K_u and the flow ratio ψ as follows:

$$u_1 = K_u\left(\sqrt{2gH_m}\right) \quad \text{and} \quad V_{f1} = \psi\left(\sqrt{2gH_m}\right)$$

where H_m is the total (or manometric) head as defined later. The usual range of values for the speed ratio and the flow ratio of impellers are given in Table 24.1.

TABLE 24.1

	Low N_s Impeller		*High N_s Impeller*
Speed ratio K_u	0.95	to	1.25
Flow ratio ψ	0.10	to	0.25

Moreover, Eq. 24.1 can be transformed in the following form:

Work done per kg per second

$$= \left(\frac{V_1^2 - V^2}{2g} + \frac{u_1^2 - u^2}{2g} + \frac{V_r^2 - V_{r_1}^2}{2g}\right) \qquad \ldots(24.4)$$

Equation 24.4 shows that work done on the liquid consists of three parts. The first part represents the change in kinetic energy of the liquid, the second part represents effect of centrifugal head and the last part indicates the change in static pressure energy of the liquid if the losses in the impeller and the effect of difference in elevations of the inlet and the outlet points of the impeller are neglected. Usually Eq. 24.4 is known as the *fundamental equation* of centrifugal pump.

24.7 HEAD OF PUMP

The head of a centrifugal pump may be expressed in the following two ways:

(a) Static head,

(b) Manometric head (or total head or gross head or effective head).

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) Static Head. As shown in Fig. 24.6 the static head is the vertical distance between the liquid surfaces in the sump and the tank to which the liquid is delivered by the pump. Thus if h_s is the vertical height of the centre line of the pump shaft above the liquid surface in the sump from which the liquid is being raised; and h_d is the vertical height of the liquid surface in the tank to which the liquid is delivered above the centre line of the pump shaft, then the static head (or lift) H_s may be expressed as

$$H_s = h_s + h_d \qquad \ldots(24.5)$$

The term h_s is known as static suction lift, and h_d is known as static delivery lift. Thus static head (or lift) is the net total vertical height through which the liquid is lifted by the pump.

(b) Manometric Head. It is the total head that must be produced by the pump to satisfy the external requirements. If there are no energy losses in the impeller and the casing of the pump, then the manometric head H_m will be equal to the energy given to the liquid by the impeller, which for radial entry to the impeller = $(V_{w1}u_1/g)$ N-m/N. But if the losses occur in the pump then

$$H_m = \frac{V_{w1}u_1}{g} - (\text{losses of head in the pump}) \qquad \ldots(24.6)$$

The various losses of head that may occur in the pump have been described in the next section.

The manometric head may also be expressed in other forms which are as indicated below:

Applying Bernoulli's equation between the points, 0 at the liquid surface in the sump and 1 in the suction pipe just at the inlet to the pump (i.e., at the centre line of the pump), the following expression is obtained if the liquid surface in the sump is taken as datum.

$$0 = \frac{p_s}{w} + \frac{V_s^2}{2g} + h_s + h_{fs}$$

or

$$\frac{p_s}{w} = -\left(\frac{V_s^2}{2g} + h_s + h_{fs}\right) \qquad \ldots(24.7)$$

where p_s is the pressure at point 1; V_s is the velocity of flow in the suction pipe; h_s is the suction lift and h_{fs} is the head loss in the suction pipe which includes the head loss due to friction and the inlet and the other minor losses. It may however be pointed out that if the pump is situated below the level of the liquid surface in the sump, h_s will be negative. Equation 24.7 indicates that at the inlet to the pump there is always a suction or vacuum pressure developed which will be recorded by the vacuum gage provided at this point as shown in Fig. 24.6. The head expressed by Eq. 24.7 is called the *suction head* of the pump.

Also, applying Bernoulli's equation between points 1 and 2, which is just at the outlet of the impeller and is assumed to be at the same level as point 1, then since the impeller imparts a head equal to $(V_{w1}u_1/g)$ to the liquid the following expression is obtained.

$$\frac{p_s}{w} + \frac{V_s^2}{2g} + \frac{V_{w1}u_1}{g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_{Li} \qquad \ldots(24.8)$$

where p_2 is the pressure and V_1 is the absolute velocity of the liquid leaving the impeller and h_{Li} is the loss of head in the impeller.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

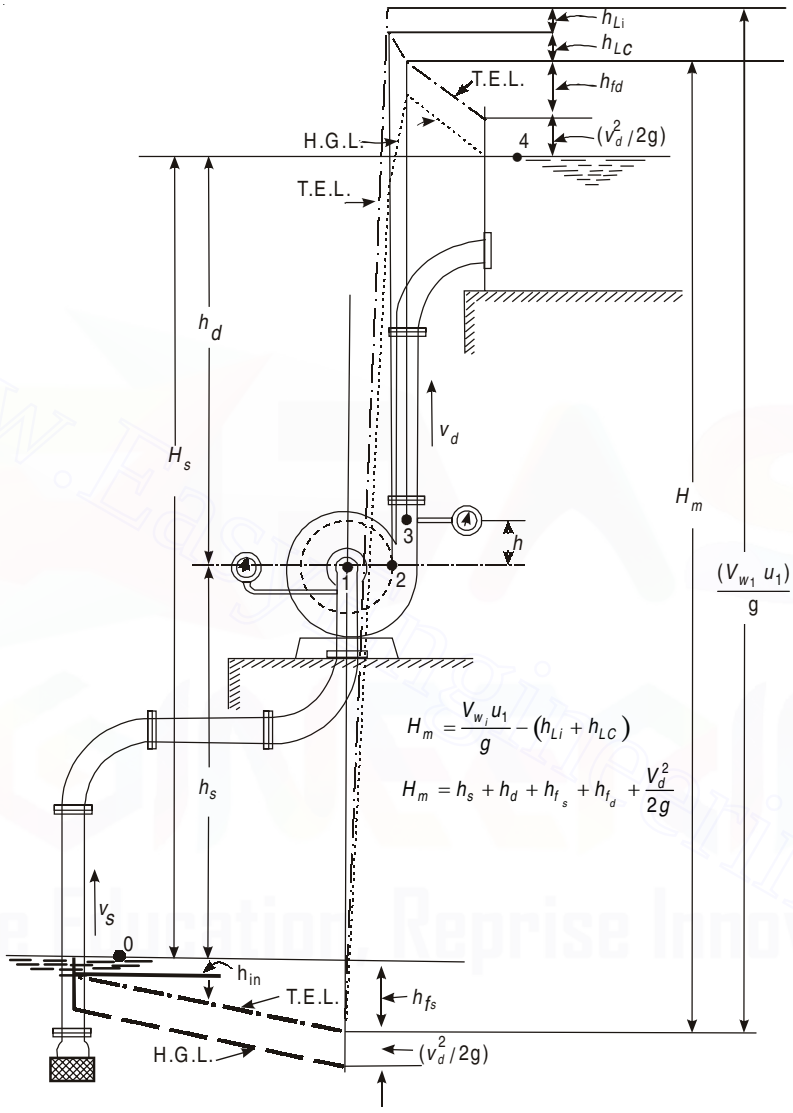


Figure 24.6 *Head on a centrifugal pump*

Further applying Bernoulli's equation between points 2 and 3, which is [as shown in Fig. 24.6] lying in the delivery pipe just at the exit from the pump, the following expression is obtained.

$$\frac{p_2}{w} + \frac{V_1^2}{2g} = \frac{p_d}{w} + \frac{V_d^2}{2g} + h + h_{LC} \qquad \ldots(24.9)$$

where p_d is the pressure in the delivery pipe at point 3 ; V_d is the velocity of flow in the delivery pipe; h

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

is the height of the point 3 above the centre line of the pump and h_{LC} is the loss of head in the casing. The pressure in the delivery pipe can be recorded by the pressure gage provided at point 3, as shown in Fig. 24.6.

Adding Eqs 24.8 and 24.9 the following expression is obtained

$$\frac{p_s}{w} + \frac{V_s^2}{2g} + \frac{V_{w1}u_1}{g} = \frac{p_d}{w} + \frac{V_d^2}{2g} + h + h_{Li} + h_{LC} \quad \ldots(24.10)$$

Equation 24.10 can however be obtained directly by applying Benoulli's equation between the points 1 and 3. Rearranging the terms of Eq. 24.10 it may be written as

$$\frac{V_{w1}u_1}{g} - (h_{Li} + h_{LC}) = \left(\frac{p_d}{w} + \frac{V_d^2}{2g} + h\right) - \left(\frac{p_s}{w} + \frac{V_s^2}{2g}\right)$$

or $\frac{V_{w1}u_1}{g}$ – (losses of head in the pump)= $\left(\frac{p_d}{w} + \frac{V_d^2}{2g} + h\right) - \left(\frac{p_s}{w} + \frac{V_s^2}{2g}\right)$

From Eq. 24.6, left hand side of the above equation is equal to H_m. Thus

$$H_m = \left(\frac{p_d}{w} + \frac{V_d^2}{2g} + h\right) - \left(\frac{p_s}{w} + \frac{V_s^2}{2g}\right) \quad \ldots(24.11)$$

Equation 24.11 thus indicates that 'the menometric head is equal to the difference between the total energies of the liquid at inlet and exit from the pump'.

If the gages are at the same level, $h = 0$, then Eq. 24.11 becomes

$$H_m = \left(\frac{p_d}{w} + \frac{V_d^2}{2g}\right) - \left(\frac{p_s}{w} + \frac{V_s^2}{2g}\right) \quad \ldots(24.12)$$

Furthermore if the suction and delivery pipes are of the same diameter, $V_s = V_d$, then manometric head is equal to difference of the pressure heads at the inlet and the exit from the pump, that is

$$H_m = \left(\frac{p_d}{w} - \frac{p_s}{w}\right) \quad \ldots(24.13)$$

The suction pipe is usually slightly larger than the delivery pipe, but as the velocities are not excessive in either of the pipes, the difference in velocity heads is small and is usually neglected, in that case also Eq. 24.13 will hold good. Moreover the suction head (p_s/w) being negative, it is indicated by Eq. 24.13 that if suction and delivery pipes are of the same size then the manometric head is equal to the arithmetical sum of the readings of the vacuum and pressure gages placed at the same level on the suction and the delivery pipes just at the inlet and the exit from the pump respectively. However, if the vacuum and the pressure gages are not placed at the same level then the manometric head is equal to the arithmetic sum of the readings of the vacuum and pressure gages plus the vertical difference in the levels of the vacuum and pressure gages. i.e.,

$$H_m = \frac{p_d}{w} - \frac{p_s}{w} + x \quad \ldots(24.13a)$$

where x is the vertical difference in the levels of the vacuum and pressure gages.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equation 24.9 may also be written as

$$\frac{p_d}{w} - \frac{p_2}{w} = \frac{V_1^2}{2g} - \frac{V_d^2}{2g} - h - h_{LC}$$

The left hand side of the above equation represents the increase in the pressure head between the point 2 at the outlet of the impeller and the point 3 at the outlet of the pump. This increase in pressure head is due to partial conversion of velocity head of liquid into pressure head and is equal to $\left(\frac{V_1^2}{2g} - \frac{V_d^2}{2g}\right)$, where V_1 is the velocity with which the liquid leaves the impeller and V_d is the velocity at point 3 in the delivery pipe. Generally this gain in pressure head is expressed as $k(V_1^2/2g)$, where k is a coefficient, the value of which varies from 0.4 for volute casing to 0.7 for a turbine pump. Thus,

$$\frac{V_1^2}{2g} - \frac{V_d^2}{2g} = k\frac{V_1^2}{2g} \qquad \ldots(24.14)$$

Equation 24.8 may be written as

$$\frac{V_{w1}u_1}{g} - h_{Li} - h_{LC} = \left(\frac{p_2}{w} - \frac{p_s}{w}\right) + \frac{V_1^2}{2g} - \frac{V_s^2}{2g} - h_{LC}$$

or

$$H_m = \left(\frac{p_2}{w} - \frac{p_s}{w}\right) + \left(\frac{V_1^2}{2g} - \frac{V_d^2}{2g}\right) + \frac{V_d^2}{2g} - \frac{V_s^2}{2g} - h_{LC}$$

Now if it is assumed that $V_d = V_s$ and Eq. 24.14 is introduced in the above expression then it becomes

$$H_m = \left(\frac{p_2}{w} - \frac{p_s}{w}\right) + k\frac{V_1^2}{2g} - h_{LC} \qquad \ldots(24.15)$$

The Eq. 24.15 also provides another expression for the manometric head according to which

H_m = the actual pressure rise in the impeller + the pressure rise in the volute chamber or the guide blade ring (i.e., between the points 2 and 3) – the loss of head in the casing.

Finally the manometric head may also be expressed by another expression as indicated below.

Applying Bernoulli's equation between points 3 and 4 (point 4, as shown in Fig. 24.6, is at the liquid surface in the delivery tank), the following expression is obtained if the datum is considered at point 3,

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = h_d + h_{fd} + \frac{V_d^2}{2g}$$

or

$$\frac{p_d}{w} = h_d + h_{fd} \qquad \ldots(24.16)$$

where h_d is the delivery lift and h_{fd} is the head loss in the delivery pipe which includes the head loss due to friction and the other minor losses.

The head $\left(h_d + h_{fd} + \frac{V_d^2}{2g}\right)$ is called the *delivery head* of the pump.

Now introducing Eqs 24.7 and 24.16 in Eq. 24.12 it becomes

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_m = h_s + h_d + h_{fs} + h_{fd} + \frac{V_d^2}{2g} \quad \ldots(24.17)$$

Equation 24.17 represents another expression for the manometric head. It is observed from Eq. 24.17 that all the manometric head is not used to lift the liquid against the static head, since some of it is used to overcome the various losses in the pipes. Hence

H_m= static head + friction and minor head loss in suction and delivery pipes + the velocity head in the delivery pipe.

If the velocity head in the delivery pipe ($V_d^2/2g$) is relatively small it may be neglected and then Eq. 24.17 becomes

$$H_m = h_s + h_d + h_{fs} + h_{fd} \quad \ldots(24.17a)$$

24.8 LOSSES AND EFFICIENCIES

(A) **Losses**. The various losses occurring during the operation of a centrifugal pump may be classified as follows:

(1) Hydraulic losses.

(2) Mechanical losses.

(3) Leakage losses.

(1) Hydraulic Losses. The *hydraulic losses* that may occur in a centrifugal pump installation may be grouped as

(a) Hydraulic losses in the pump.

(b) Other hydraulic losses.

The hydraulic losses that may occur within the pump consist of the following:

(i) Shock or eddy losses at the entrance to and the exit from the impeller.

(ii) Friction losses in the impeller.

(iii) Friction and eddy losses in the guide vanes (or diffuser) and casing.

It can be seen from Fig. 24.5 that for the given values of the blade angles θ and ϕ, and the speed of rotation, there can be only one rate of discharge that will ensure tangential entry to the impeller and tangential exit from the impeller. But often the pump is required to operate under varying conditions which results in the variation in the rate of discharge. As such at the entrance and the exit of the impeller the shock losses generally occur. Furthermore at the exit from the impeller there occurs a loss of energy due to the more or less abrupt change in the direction of the velocity of liquid as it enters the casing.

The other hydraulic losses consist of the following:

(i) Friction and other minor losses in the suction pipe.

(ii) Friction and other minor losses in the delivery pipe.

(2) Mechanical Losses. The *mechanical losses* occur in the centrifugal pump on account of the following:

(i) Disc friction between the impeller and the liquid which fills the clearance spaces between the impeller and the casing.

(ii) Mechanical friction of the main bearings and glands.

(3) Leakage Losses. In centrifugal pumps as ordinarily built, it is not possible to provide a completely water tight seal between the delivery and suction spaces. As such there is always a certain amount of liquid which slips or leaks from the high pressure to the low pressure points in the pump and it never

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

passes through the delivery pipe. The liquid which escapes or leaks from a high pressure zone to a low pressure zone carries with it energy which is subsequently wasted in eddies. This loss of energy due to leakage of liquid represents the *leakage loss*.

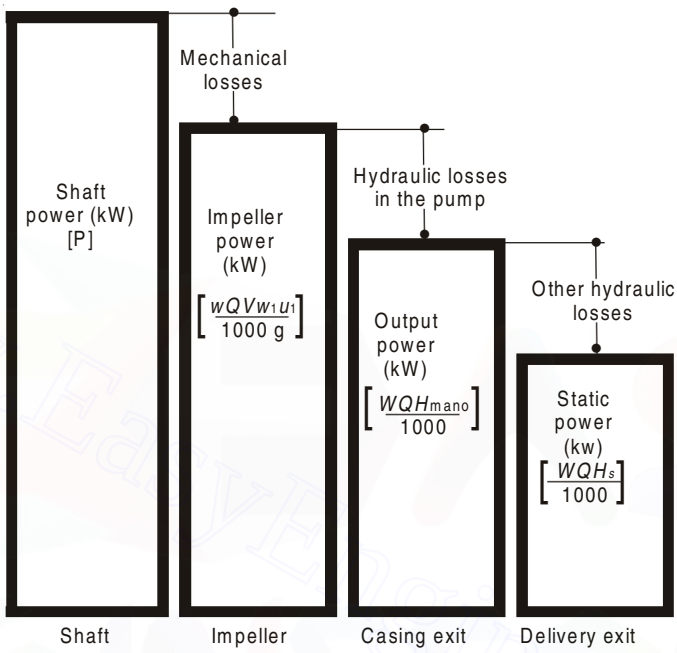


Figure 24.7 *Head losses in centrifugal pump*

(B) **Efficiencies**. The efficiency of a centrifugal pump is expressed in the following form :

(i) Manometric efficiency,

(ii) Volumetric efficiency,

(iii) Mechanical efficiency,

(iv) Overall efficiency.

(i) Manometric Efficiency. The manometric efficiency η_{mano} is defined as the ratio of the manometric head developed by the pump to the head imparted by the impeller to the liquid. Thus

$$\eta_{mano} = \frac{H_m}{(V_{w1}u_1 / g)} = \frac{gH_m}{V_{w1}u_1} \qquad \ldots(24.18)$$

or

$$\eta_{mano} = \frac{H_m}{H_m + \text{Losses in the pump}}$$

If Q is the volume of liquid actually delivered per second by the pump and w is the specific weight of the liquid then

$$\eta_{mano} = \frac{wQH_m}{wQ(V_{w1}u_1 / g)} \qquad \ldots(24.19)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$\eta_{mano} = \frac{\text{Power actually delivered by the pump}}{\text{Power imparted by the impeller}}$$

or $$\eta_{mano} = \frac{\text{Output of the pump}}{\text{Power imparted by the impeller}} \quad \ldots(24.20)$$

[Power imparted by the impeller = Power delivered by the motor to the shaft (or shaft power) – Power lost in mechanical losses]

(ii) Volumetric Efficiency. The volumetric efficiency η_v is defined as the ratio of the quantity of liquid discharged per second from the pump to the quantity passing per second through the impeller. As stated earlier these two quantities differ by the rate ΔQ at which the liquid from the impeller leaks through the clearances between the impeller and the casing and finds its way back to the eye of the impeller. Thus

$$\eta_v = \frac{Q}{(Q + \Delta Q)} \quad \ldots(24.21)$$

where Q is the quantity of liquid actually discharged per second from the pump.

(iii) Mechanical Efficiency. The mechanical efficiency η_{mech} is defined as the ratio of the power actually delivered by the impeller to the power supplied to the shaft by the prime mover or motor. Thus

$$\eta_{mech} = \frac{w(Q + \Delta Q)(V_{w_1} u_1 / g)}{\text{Power given to the shaft}} \quad \ldots(24.22)$$

or $$\eta_{mech} = \frac{(V_{w1} u_1 / g)}{\text{Energy head given to the shaft}}$$

or $$\eta_{mech} = \frac{(V_{w1} u_1 / g)}{(V_{w1} u_1 / g) + (\text{mechanical head losses in bearing})} \quad \ldots(24.22a)$$

(iv) Overall Efficiency. The overall efficiency η_o of the pump is defined as the ratio of the power output from the pump to the power input from the prime mover driving the pump. Thus

$$\eta_o = \frac{wQH_m}{\text{Power given to the shaft}} \quad \ldots(24.23)$$

The overall efficiency is also equal to the product of all the three efficiencies described above. That is

$$\eta_o = (\eta_{mano}) \times (\eta_v) \times (\eta_{mech})$$

$$= \frac{H_m}{(V_{w1} u_1 / g)} \times \frac{Q}{(Q + \Delta Q)} \times \frac{w(Q + \Delta Q)(V_{w1} u_1 / g)}{\text{Power given to the shaft}}$$

$$= \frac{wQh_m}{\text{Power given to the shaft}}$$

which is same as Eq. 24.23.

(C) **Effect of Vane Angle ϕ on Manometric Efficiency.** If the loss of head in the pump is neglected then the manometric head may be expressed as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$H_m = \frac{V_{w1}u_1}{g} - \frac{V_1^2}{2g} \qquad \ldots(24.24)$$

From the outlet velocity triangle shown in Fig. 24.5,

$$V_1^2 = \left(V_{w1}^2 + V_{f1}^2\right) \text{ and } V_{w1} = (u_1 - V_{f1}\cot\phi).$$

Then

$$H_m = \frac{u_1^2 - V_{f1}^2 \operatorname{cosec}^2\phi}{2g} \qquad \ldots(24.25)$$

Under the ideal condition assumed above the manometric efficiency of the pump will become

$$\eta_{\text{mano}} = \frac{gH_m}{V_{w1}u_1} = \frac{(u_1^2 - V_{f_1}^2\operatorname{cosec}^2\phi)}{2u_1(u_1 - V_{f_1}\cot\phi)} \qquad \ldots(24.26)$$

Now assuming a constant value for the flow ratio $\psi\left[= V_{f1}/\sqrt{2gH_m}\right] = 0.25$ and computing the value of u_1 in terms of H_m from Eq. 24.25 for different values of ϕ, it will be observed that as the value of ϕ varies from 90° to 20°, the value of η_{mano} increases from 0.47 to 0.73. It is obvious from equation 24.26 that a further decrease in the value of ϕ will result in the increase in the efficiency, but it is impracticable to have the values of ϕ less than 20°, since it will result in the long and narrow passages causing high frictional losses. As such in practice the value of ϕ is not reduced below 20°, but some other devices in the form of guide vanes etc., are adopted which will convert a part of the velocity energy into useful pressure energy, thereby increasing the efficiency of the pump.

(D) **Effect of Finite Number of Vanes of the Impeller on Head and Efficiency.** Eq. 24.3 or 24.4 represents the head imparted by the impeller to the liquid only if there are infinite number of vanes (or blades) in the impeller and it is usually known as *Euler's head*. Similarly the velocity triangles shown in Fig. 24.5 will be developed only if there are infinite number of vanes in the impeller and these velocity triangles are also known as *Euler's velocity triangles*. However, in practice the impeller has only a finite number of vanes. Therefore although the vanes are designed in accordance with the consideration of Euler's velocity triangles, but the actual velocity triangles developed in the case of impellers with finite number of vanes are different from the Euler's velocity triangles. On account of secondary or circulatory flow effects developed in the impeller the actual velocity of whirl V_{w1} developed in the case of impeller with finite number of vanes is always less than that corresponding to the Euler's velocity triangles. Consequently the actual head H_i imparted by the pump impeller (with finite number of vanes) to the liquid is always less than the Euler's head H_e (given by Eq. 24.3 or 24.4). The difference between the actual head H_i imparted by the impeller with finite number of vanes to the liquid and the Euler's head H_e is not considered as loss of head. Further the ratio (H_i/H_e) is usually termed as *vane efficiency* or *vane effectiveness* ε (Greek 'epsilon') i.e.,

$$\varepsilon = \frac{H_i}{H_e}$$

Experiments made on impellers of identical size, speed and vane angle but with different number of vanes have indicated that as the number of vanes is increased the value of ε increases and approaches unity. In other words, it means that the actual head imparted by the impeller (with finite number of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

vanes) to the liquid, approaches the Euler's head as the number of vanes is increased. In addition to the number of vanes, the value of ε also depends on the shape of the vane and the outlet vane angle. In general for radial flow pumps the value of ε varies from 0.6 to 0.8 as the number of vanes is increased from 4 to 12. However, for impellers with vanes more than 24 the value of ε may be taken as unity.

A portion of the head H_i actually imparted by the impeller (with finite number of vanes) to the liquid will be lost in the pump and hence the net manometric head H_m available from the pump may be expressed as

$$H_m = H_i - \text{losses of head in the pump.}$$

Further the ratio of the manometric head H_m available from a pump and the head H_i actually imparted by the impeller to the liquid is termed as *hydraulic efficiency* η_h of the pump i.e.,

$$\eta_h = \frac{H_i - \text{losses of head in the pump}}{H_i}$$

or
$$\eta_h = \frac{H_m}{H_i}$$

However, the manometric efficiency η_{mano} of a pump has been defined earlier as (Eq. 24.18)

$$\eta_{mano} = \frac{H_m}{(V_{w1}u_1/g)} = \frac{H_m}{H_e}$$

Thus combining the above noted expressions the manometric efficiency becomes

$$\eta_{mano} = \frac{H_m}{H_i} \times \frac{H_i}{H_e} = (\eta_h \times \varepsilon)$$

Again for $\varepsilon = 1; H_i = H_e$ and $\eta_{mano} = \eta_h$

As stated earlier since the actual shape of the velocity triangles for impellers with finite numbers of vanes being not known, the impeller vanes are designed in accordance with the Euler's velocity triangles. As such in the various illustrative examples the value of ε has been assumed as 1 and accordingly H_i has been considered to be same as H_e.

24.9 MINIMUM STARTING SPEED

When the pump is started, there will be no flow of water until the pressure difference in the impeller is large enough to overcome the gross or manometric head. If the impeller is rotating, but there is no flow, then the water is rotating in a forced vortex. Therefore a centrifugal head or pressure head caused by the centrifugal force on the rotating water will be $(u_1^2 - u^2)/2g$. The flow will commence only if

$$[(u_1^2 - u^2)/2g] \geq H_m$$

or
$$\left[\frac{(\pi D_1 N/60)^2}{2g} - \frac{(\pi DN/60)^2}{2g}\right] \geq H_m$$

or
$$\left[\left(\frac{\pi N}{60}\right)^2 (D_1^2 - D^2)\right] \geq (2gH_m)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

For determining the minimum speed required for the pump to commence the flow, the above expression may be written as

$$\left[\left(\frac{\pi N}{60}\right)^2 (D_1^2 - D^2)\right] = (2gH_m) \qquad \ldots(24.27)$$

from which the required value of N may be computed.

24.10 LOSS OF HEAD DUE TO REDUCED OR INCREASED FLOW

The efficiency of a pump will be maximum only when it is running and discharging at its designed speed. But if the discharge is either reduced or increased, then there will be a loss of head due to shock at the entrance to the impeller, which will result in lowering the efficiency of the pump.

As shown in Fig. 24.8, let *abd* be the inlet velocity triangle for the pump when running under normal conditions. The vanes at inlet tip will be parallel to *ab*. Now if the radial flow through the pump is reduced (or increased) from *bd* to *cd*, while the speed of rotation remains the same, the velocity triangle will be represented by *acd*, *ac* being the new relative velocity. But the angle of the vane at inlet will be the same as before, the relative velocity, therefore, will no longer be parallel to the vane and the shock will occur.

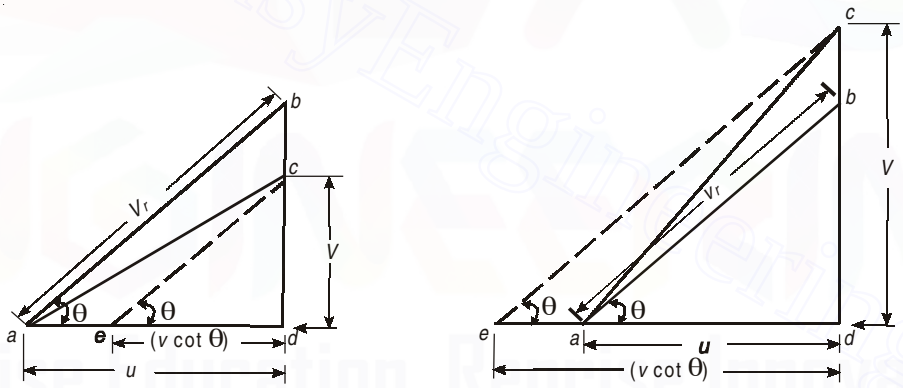


Figure 24.8 *Inlet velocity triangle with reduced and increased flow*

Now as the new velocity of flow *cd* is fixed and also the water must pass along the vane, it follows that the velocity diagram will be triangle *ecd*, *ec* being parallel to *ab*. Therefore a tangential change of velocity *ae* will suddenly take place resulting in the shock causing the head loss. The loss of head due to sudden change in velocity

$$= \left[\frac{(\text{change in velocity})^2}{2g}\right]$$

∴ Loss of head at the entrance

$$= \left[\frac{(ae)^2}{2g}\right] = \left[\frac{(u - V\cot\theta)^2}{2g}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

24.11 DIAMETERS OF IMPELLER AND PIPES

(a) Outside Diameter of Impeller. If the outside diameter of the impeller is D_1 and the speed of the pump shaft is N r.p.m.,then

$$u_1 = (\pi D_1 N/60)$$

Further if H_m is the total head, then

$$u_1 = K_u\left(\sqrt{2gH_m}\right)$$

where K_u is the speed ratio for the pump.

$$\therefore \quad D_1 = \frac{60k_u\sqrt{2gH_m}}{\pi N} = \frac{(84.60)K_u(\sqrt{H_m})}{N} \qquad \ldots(24.28)$$

The value of K_u usually ranges from 0.95 to 1.25 as shown in Table 24.1. Equation 24.28 can, however, be used to determine the head which a pump can develop if D_1 and N are known. This will serve at a check for an existing pump.

(b) Inlet Diameter of Impeller. The inlet diameter D is $\frac{2}{3}$ to $\frac{1}{3}$ of D_1depending upon the specific speed N_s or total head H_m. However in most of the cases, $D = (0.5)\, D_1$.

(c) Least Diameter of Impeller. An expression for the minimum diameter of an impeller can be derived on the basis of the fact that for the pump to start pumping, the centrifugal head must be equal to the total head H_m.Thus

$$\left[\frac{(u_1^2 - u^2)}{2g}\right] = H_m$$

or $$\left[\left(\frac{\pi D_1 N}{60}\right)^2 - \left(\frac{\pi DN}{60}\right)^2\right] = 2gH_m$$

or $$(D_1^2 - D^2) = \left(\frac{60}{\pi N}\right)^2 \times (2\, gH\, m)$$

Taking $D = 0.5\, D_1$; and solving for D_1

$$D_1 = \frac{2\times 60\times(\sqrt{2gH_m})}{(\sqrt{3})\pi N} = \frac{(97.68)(\sqrt{H_m})}{N} \qquad \ldots(24.29)$$

A comparison of Eqs 24.28 and 24.29 indicates that for speed ratio K_u equal to $\left(2\sqrt{3}\right)$ both the equations become same.

(d) Diameter of Suction Pipe. If d_s is the diameter of the suction pipe and V_s is the velocity of flow in suction pipe, then the amount of water to be pumped is given by

$$Q = \left[\frac{(\pi d_s^2)V_s}{4}\right]$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$d_s = \left[\sqrt{\frac{(4Q)}{(\pi V_s)}}\right] \quad \ldots(24.30)$$

Usually V_s is 1.5 to 3 m/s.

(e) Diameter of Delivery Pipe. If d_d is the diameter of the delivery pipe and V_d is velocity of flow in delivery pipe, then

$$Q = \left[\left(\frac{\pi}{4}\right) d_d^2 V_d\right]$$

or $$d_d = \left[\sqrt{\frac{(4Q)}{(\pi V_d)}}\right] \quad \ldots(24.31)$$

Usually V_d is 1.5 to 3.5 m/s. The value of V_d is generally equal to or or slightly higher than that of V_s.

24.12 SPECIFIC SPEED

In order to compare the performance of different pumps, it is necessary to have some term which will be common to all the centrifugal pumps. The term ordinarily used for this purpose is the 'specific speed'. This is a type characteristic and can be used to predict the behaviour of one pump based on tests of similar, but different sized pumps.

The specific speed of a centrifugal pump may be defined as *the speed in revolutions per minute of a geometrically similar pump of such a size that under corresponding conditions it would deliver 1 litre of liquid per second against a head of 1 metre.* It is represented by N_s, and it may be expressed by a relation as derived below:

Discharge $$Q = (k\pi B_1 D_1)\, V_{f1}$$

or $$Q \propto B_1 D_1 V_{f1}$$

or $$Q \propto D_1^2 V_{f1}\ ;\ \text{since } B_1 \propto D_1$$

Also $$V_{f1} = \psi\left(\sqrt{2gH_m}\right);\ \text{or } V_{f1} \propto \left(\sqrt{H_m}\right)$$

Thus $$Q \propto D_1^2\left(\sqrt{H_m}\right)$$

or $$\frac{Q}{D_1^2(\sqrt{H_m})} = \text{constant} \quad \ldots(24.32)$$

Further $$u_1 = (\pi D_1 N/60)$$

or $$D_1 \propto (u_1/N)$$

Also $$u_1 = K_u\left(\sqrt{2gH_m}\right),\ u_1 \propto \left(\sqrt{H_m}\right)$$

Thus $$D_1 \propto \left(\sqrt{H_m}/N\right)$$

or $$\frac{\sqrt{H_m}}{D_1 N} = \text{constant} \quad \ldots(24.33)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Substituting the value of D_1 from Eq. 24.33 in Eq. 24.32 it becomes

$$Q \propto (H_m^{3/2}/N^2) \quad \ldots(24.34)$$

or $$N^2 \propto (H_m^{3/2}/Q)$$

or $$N \propto \left(H_m^{3/4}\sqrt{Q}\right)$$

or $$N = C\left(H_m^{3/4}\sqrt{Q}\right); \text{ where } C \text{ is a constant}$$

or $$\frac{N\sqrt{Q}}{H_m^{3/4}} = C \quad \ldots(24.35)$$

Now according to the definition of the specific speed, putting $Q = 1$ litre/second and $H_m = 1$ m, then $C = N = N_s$. Thus

$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}} \quad \ldots(24.36)$$

Equation 24.36 represents the expression for the specific speed of a centrifugal pump, which is based on the unit discharge. It is the most commonly adopted expression for the specific speed of the pumps. The value of N_s usually varies from about 300 to 15,000 for a single impeller. Further the value of N_s is same is SI and metric system of units and it is equal to 0.67 times N_s in F.P.S system, where Q is in gallons per minute and H_m is in feet.

The values of Q and H_m to be substituted in Eq. 24.36 are those corresponding to the maximum efficiency of the pump at its normal working speed.

For a multi-stage pump the value of H_m to be used in Eq. 24.36 is obtained by dividing the total head developed by the number of stages. Similarly for a double suction pump half the actual discharge delivered by the pump is taken as Q.

Sometimes another definition of the specific speed of a centrifugal pump is used, which is based on unit power, though it is not very common. According to this basis the specific speed is defined as 'the speed in revolutions per minute of a geometrically similar pump of such a size that under corresponding conditions it would absorb 1 kW power when working against a head of 1 metre'. If the power required to drive the pump is P, then

$$P = [(wQH_m)/(\eta_0)]$$

Assuming η_0 to be constant and substituting for Q from Eq. (24.34), we have

$P \propto (H_m^{5/2}/N^2)$; or $P = \text{constant}\ (H_m^{5/2}/N^2)$ $\quad \ldots(24.37)$

Hence $$N = [C_1 H_m^{5/4}/P^{1/2}) \text{ where } C_1 \text{ is a constant.}$$

Now according to the definition of the specific speed, putting $H_m = 1$ and $P = 1$, then $C_1 = N = N_s$. Thus

$$N_s = \frac{N\sqrt{P}}{H_m^{5/4}} \quad \ldots(24.38)$$

Equation 24.38 is, however, not very common with the pumps. As such if nothing is mentioned, then the specific speed of a pump generally refers to that obtained on the basis of the unit discharge criterion, as represented by Eq. 24.36.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Centrifugal pumps may also be classified on the basis of the specific speed. Table 24.2 gives the different types of centrifugal pumps with the corresponding ranges of specific speeds. The values of N_s given in this table are as given by Eq. 24.36.

TABLE 24.2

S. No.	*Type of pump*	*Specific speed* N_s
1	Slow speed radial flow	300 to 900
2	Medium speed radial flow	900 to 1500
3	High speed radial flow	1500 to 2400
4	Mixed flow or screw type	2400 to 5000
5	Axial flow or propeller type	3400 to 15000

24.13 MODEL TESTING OF PUMPS

The large-sized pumps are usually manufactured only after testing their small scale models, because from the test results of the models it would be possible to predict the performance of the prototype in advance and also it would be possible to make necessary alterations in the design of the prototype pump. The performance of the prototype pump will be correctly predicted by its model tests only if the model bears a complete similitude with its prototype. The necessary conditions for the complete similarity to exist between the model of a pump and its prototype may be obtained as indicated below.

The variables involved in this case are discharge Q, manometric or total head H, speed of rotation of impeller N, impeller diameter D, power P, mass density ρ and viscosity μ of the fluid pumped. Again as in the case of turbines considering gH as the variable in place of H, these variables may be grouped into the dimensionless parametres viz., discharge number $\left(\frac{Q}{ND^3}\right)$, head number $\left(\frac{gH}{N^2D^2}\right)$, power number $\left(\frac{p}{\rho gHND^3}\right)$ and Reynolds number $\left(\frac{\mu}{\rho ND^2}\right)$. By dividing the square root of discharge number by head number raised to power (3/4) the following dimensionless parameter n_s is obtained

$$n_s = \frac{N\sqrt{Q}}{(gH)^{3/4}}$$

The parameter n_s is known as *dimensionless specific speed* or *shape number* of the pump.

For complete similarity to exist between the model and the prototype pumps the above noted parameters must have the same value for the model and the prototype pumps. However, as described in Chapter 22 (section 22.6) since the flow in the prototype pump is turbulent, if the flow in the model pump is also turbulent then even if the Reynolds numbers are not equal for the model and the prototype pumps, the similarity between them can be ensured. Thus for complete similarity to exist between the model and the prototype pumps the necessary conditions are as given below:

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p \quad \ldots(24.39)$$

$$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p \quad \ldots(24.40)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left.\begin{aligned} \left(\frac{P}{\rho g Q H}\right)_m &= \left(\frac{P}{\rho g Q H}\right)_p \\ \text{or} \quad \left(\frac{P}{\rho g H N D^3}\right)_m &= \left(\frac{P}{\rho g H N D^3}\right)_p \\ \text{or} \quad \left(\frac{P}{\rho N^3 D^5}\right)_m &= \left(\frac{P}{\rho N^3 D^5}\right)_p \end{aligned}\right\} \quad \ldots(24.41)$$

$$\left[\frac{N\sqrt{Q}}{(gH)^{3/4}}\right]_m = \left[\frac{N\sqrt{Q}}{(gH)^{3/4}}\right]_p \quad \ldots(24.42)$$

Since in most of the cases $g_m = g_p$ and $\rho_m = \rho_p$ the above conditions may be simplified.

The above noted conditions for establishing the similarity between the model and the prototype pumps are based on the assumption that the efficiency of the model is equal to that of the prototype. However, the efficiencies of the model and the prototype pumps are not equal. This is so because on account of the losses in the model and the prototype pumps being disproportionate, the model efficiency is usually lower than that of the prototype. As such in order to correctly predict the performance of the prototype pump on the basis of the model test results the above noted expressions for the similarity conditions needs to be modified and the modified expressions are as indicated below.

$$\left(\frac{gH}{\eta_{mano} N^2 D^2}\right)_m = \left(\frac{gH}{\eta_{mano} N^2 D^2}\right)_p \quad \ldots(24.40\ a)$$

$$\left.\begin{aligned} \left(\frac{P\eta_0}{\rho g Q H}\right)_m &= \left(\frac{P\eta_0}{\rho g Q H}\right)_p \\ \text{or} \quad \left(\frac{P\eta_0}{\rho g H N D^3}\right)_m &= \left(\frac{P\eta_0}{\rho g H N D^3}\right)_p \\ \text{or} \quad \left(\frac{P\eta_0}{\rho N^3 D^5}\right)_m &= \left(\frac{P\eta_0}{\rho N^3 D^5}\right)_p \end{aligned}\right\} \quad \ldots(24.41\ a)$$

$$\left[\frac{N\sqrt{Q}\ \ \eta_{mano}^{3/4}}{gH^{3/4}}\right]_m = \left[\frac{N\sqrt{Q}\ \ \eta_{mano}^{3/4}}{gH^{3/4}}\right]_p \quad \ldots(24.42\ a)$$

Further the efficiency of the prototype pump being not equal to that of the model pump, by applying suitable correction factor, the efficiency of the prototype pump can be predicated from the efficiency obtained for its model. Based on the test efficiencies of single stage centrifugal pumps, G.F. Wislicenus has suggested the following expression relating the efficiencies of the model and the prototype pumps.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{0.95 - \eta_m}{0.95 - \eta_p} = \left[\frac{0.658 + \log_{10} Q_p}{0.658 + \log_{10} Q_m}\right]^2 \quad \ldots(24.43)$$

in which Q is the discharge in litres per minute.

24.14 PUMP IN SERIES—MULTI-STAGE PUMPS

The head produced by a centrifugal pump depends on the rim speed of the impeller. To increase the rim speed, either the rotative speed or the diameter of the impeller or both must be increased. Increasing either of these has the effect of increasing the stress in the impeller material. For this reason it is usually not possible to produce very high head with one impeller. Normally a pump with a single impeller can be used to deliver the required discharge against a maximum head of about 100 m. But if the liquid is required to be delivered against a still larger head then it can be done by using two or more *pumps in series*. The pumps in series may be used in boiler feeding system to satisfy high head demand, in multi-stations systems along pipelines such as long oil pipelines, for boosting the liquid like pumping installations interposed in the water mains, etc. The higher heads may also be produced by using multi-stage pumps.

A multi-stage pump consists of two or more identical impellers mounted on the same shaft, and enclosed in the same casing. All the impellers are connected in series, so that liquid discharged with increased pressure from one impeller passes through the connecting passages to the inlet of the next impeller and so on, till the discharge from the last impeller passes into the delivery pipe. The impellers are surrounded by guide vanes which are generally provided within the connecting passages, and are meant for the recuperation of the velocity energy of the liquid leaving the impeller into pressure energy. According to the number of impellers fitted in the casing a multi-stage pump is designated as two stage, three-stage, etc. Figure 24.9 shows a three-stage centrifugal pump.

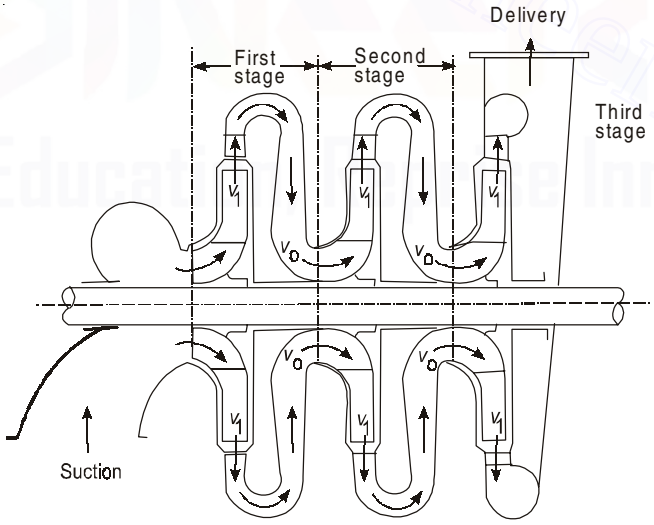


Figure 24.9 *Three-stage centrifugal pump*

Figure 24.10 indicates the variations of the velocity and the pressure head of the liquid as it passes through the pump. As the liquid passes through each impeller the absolute velocity of the liquid

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

increases to V_1 and in each connecting passage the absolute velocity decreases again to V_0; but the pressure head continuously increases.

If H_1 and H_2 are the pressure heads gained by the liquid in each impeller and the surrounding guide vanes respectively, then the pressure head impressed on the liquid at each stage is $H_m = (H_1 + H_2)$. Now, if there are n impellers then since at each stage the pressure head will be raised by the same amount, the total head H developed by this multi-stage pump will be

$$H = n(H_m) = n(H_1 + H_2) \qquad \ldots(24.44)$$

Since the same liquid flows through each impeller, the discharge of a multi-stage pump is same as the discharge passing through each impeller of the series.

The number of stages to be adopted depends on the limitations of speed of the driving motor and also upon the discharge and the total head. If these factors are known, the head per stage can be fixed on the basis of the fact that specific speed N_s per impeller should not be less than about 700, with a provisional maximum limit of head per stage of 160 m.

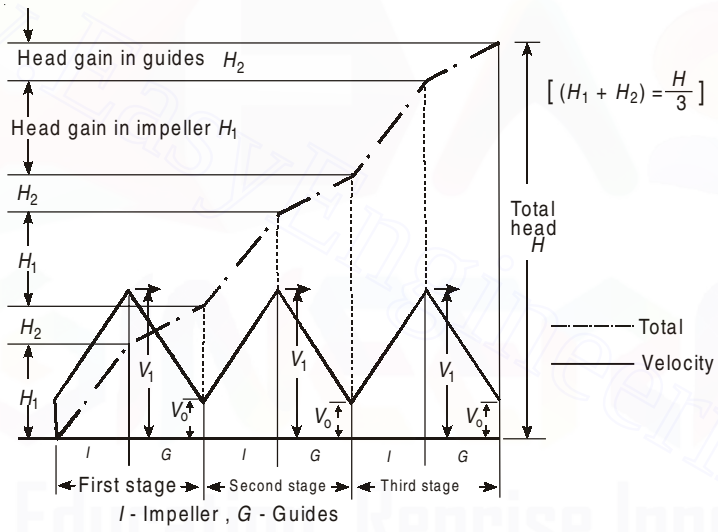


Figure 24.10 *Variations of velocity and pressure head in a three-stage pump*

24.15 PUMPS IN PARALLEL

The multi-stage pumps or the pumps in series as described earlier are employed for delivering a relatively small quantity of liquid against very high heads. However, when a large quantity of liquid is required to be pumped against a relatively small head, then it may not be possible for a single pump to deliver the required discharge. In such cases two or more pumps are used which are so arranged that each of these pumps working separately lifts the liquid from a common sump and delivers it to a common collecting pipe through which it is carried to the required height (Fig. 24.11). Since in this case each of the pumps delivers the liquid against the same head, the arrangement is known as *pumps in parallel*. If $Q_1, Q_2, Q_3 \ldots\ldots\ldots Q_n$ are the discharging capacities of n pumps arranged in parallel then the total discharged delivered by these pumps will be

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$Q_t = (Q_1 + Q_2 + Q_3 ..... Q_n) \quad \ldots(24.45)$$

If the discharging capacity of each of the n pumps is same, equal to Q then the total discharge delivered by these pumps will be

$$Q_t = nQ \quad \ldots(24.46)$$

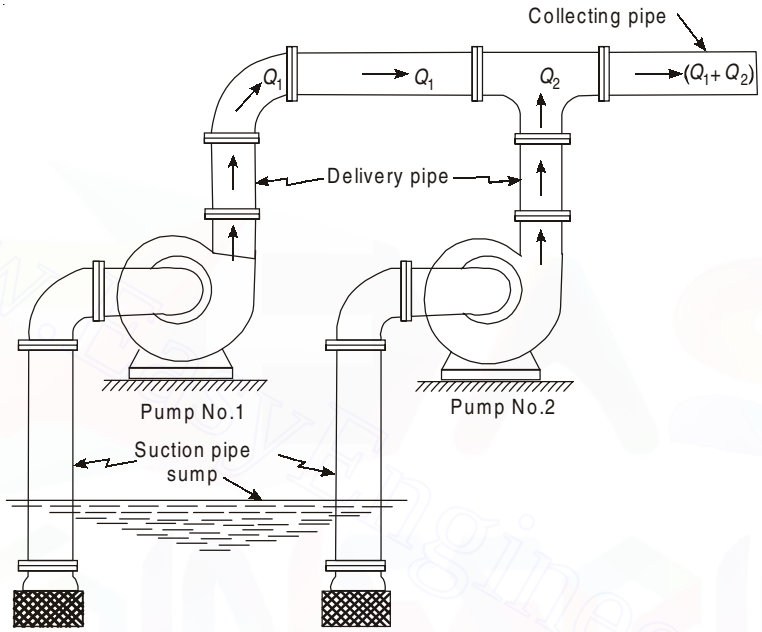


Figure 24.11 *Two centrifugal pumps arranged in parallel*

24.16 PERFORMANCE OF PUMPS—CHARACTERISTIC CURVES

A pump is usually designed for one speed, flow rate and head, but in actual practice the operation may be at some other condition of head or flow rate, and for the changed conditions the behaviour of the pump may be quite different. As indicated earlier, if the flow through the pump is less than the designed quantity, the value of the velocity of flow of liquid through the impeller will be changed, and as a result, the values of V_w and V_{w1} will be altered, thereby changing the head developed by the pump, and at the same time the losses will increase so that the efficiency of the pump will be lowered. Therefore, in order to predict the behaviour and performance of a pump under varying conditions, tests are performed, and the results of the tests are plotted. The curves thus obtained are known as the characteristic curves of the pump. The following three types of characteristic curves are usually prepared for the centrifugal pumps:

(a) Main and operating characteristics.

(b) Constant efficiency or Muschel curves.

(c) Constant head and constant discharge curves.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) Main and Operating Characteristics In order to obtain the main characteristic curves of a pump it is operated at different speeds. For each speed the rate of flow Q is varied by means of a delivery valve and for the different values of Q the corresponding values of manometric head H_m, shaft power P, and overall efficiency η_0, are measured or calculated. The same operation is repeated for different speeds of the pump. Then H_m v/s Q; P v/s Q and η_0 v/s Q curves for different speeds are plotted, so that three sets of curves, as shown in Fig. 24.12 are obtained, which represent the *main characteristics* of a pump. The main characteristics are useful in indicating the performance of a pump at different speeds.

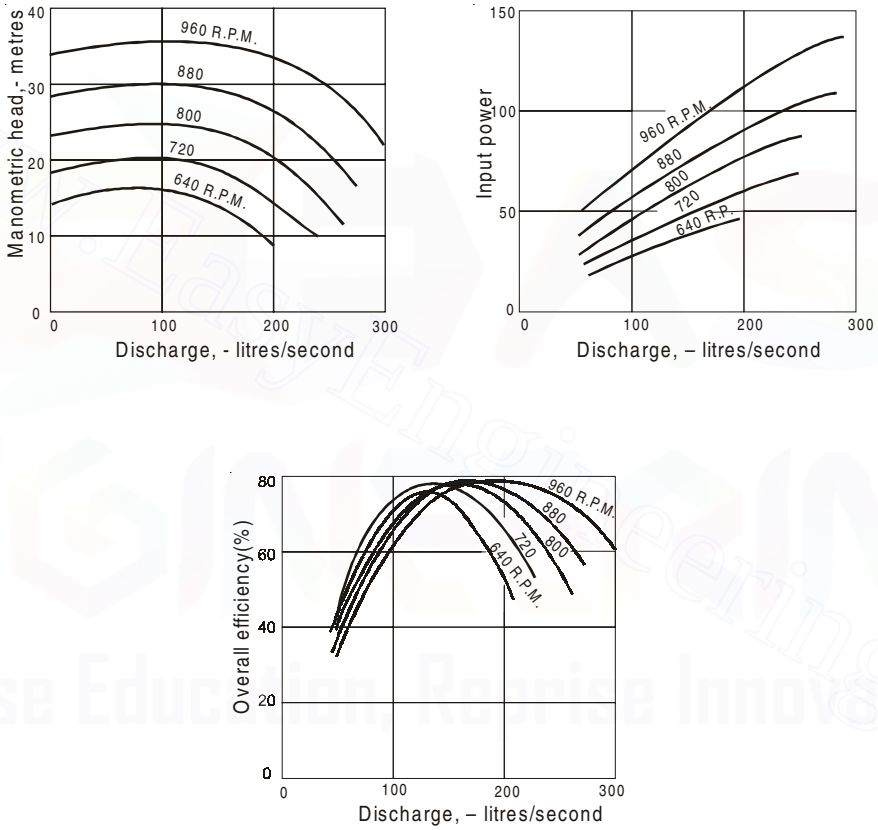


Figure 24.12 *Main characteristics of a centrifugal pump*

During the operation a pump is normally required to run at a constant speed, which is its designed speed (same as the speed of the driving motor). As such that particular set of main characteristics which corresponds to the designed speed is mostly used in the operations of a pump and is, therefore, known as the *operating characteristics.* A typical set of such characteristics of a pump is shown in Fig. 24.13.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

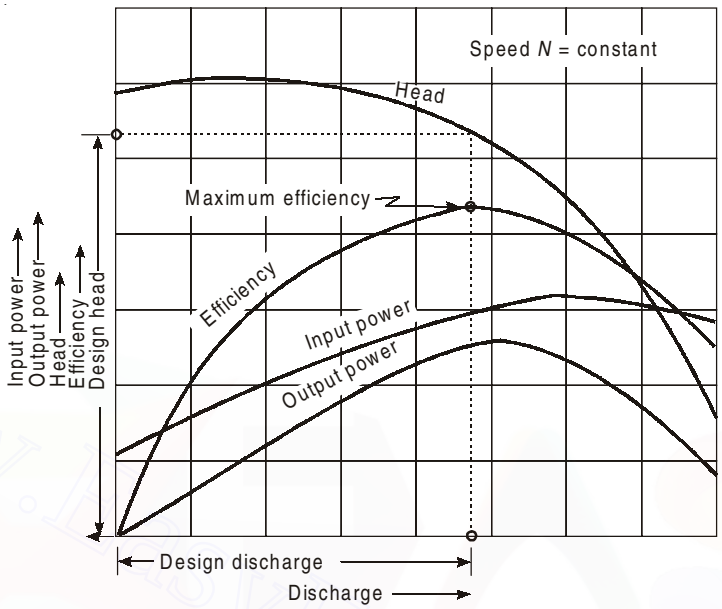


Figure 24.13 *Operating characteristic curves of a centrifugal pump*

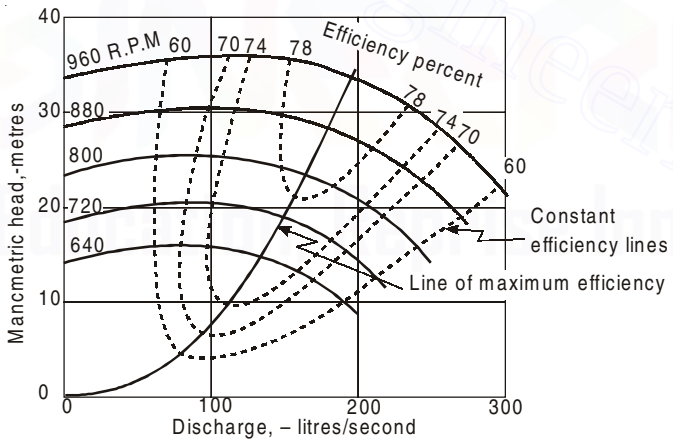


Figure 24.14 *Constant efficiency curves of a centrifugal pump*

The values of the head and the discharge corresponding to the maximum efficiency are known as the normal (or designed) head and the normal (or designed) discharge of a pump. The head corresponding to zero or no discharge is known as the *shut-off head* of the pump. From these characteristics it is possible to determine whether the pump will handle the necessary quantity of liquid against the desired head and what will happen if the head is increased or decreased. The

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

P *v/s* Q curve will show what size motor will be required to operate the pump at the required conditions and whether or not the motor will be overloaded under any other operating conditions.

(b) Constant Efficiency Curves As shown in Fig. 24.14 the constant or iso-efficiency curves may be obtained from H_m *v/s* Q and η_0 *v/s* Q curves of Fig. 24.12. In order to plot the iso-efficiency curves, horizontal lines representing constant efficiencies are drawn on the η_0 *v/s* Q curves. The points, at which these lines cut the efficiency curves at various speeds, are transferred to the corresponding H_m *v/s* Q curves. The points corresponding to the same efficiency are then joined by smooth curves which represent the iso-efficiency curves. From these curves the line of maximum efficiency as shown in Fig. 24.14 may be obtained. The iso-efficiency curves facilitate the direct determination of the range of operation of a pump with a particular efficiency.

(c) Constant Head and Constant Discharge Curves These curves are useful in determining the performance of a variable speed pump for which the speed constantly varies. In such cases if the head H_m is maintained constant then as the speed N varies the rate of flow Q will vary. As such a plot of Q *v/s* N can be prepared which can be used to determine the speeds required to discharge varying amounts of liquid at a constant pressure head. It is obvious from Eq. 24.39 that $Q \sim N$ and hence Q *v/s* N plot will be a straight line as shown in Fig. 24.15. Similarly if a constant rate of flow Q is to be maintained then as N varies, H_m will vary. Thus a plot of H_m *v/s* N can be prepared which can be used to determine the speeds required to discharge a certain quantity of liquid at different pressure heads. From Eq. 24.40 it is evident that $H_m \sim N^2$ and hence H_m *v/s* N plot is a parabolic curve as shown in Fig. 24.15. Similarly it is obvious from Eq. 24.41 that $P \sim N^3$, as shown in Fig. 24.15.

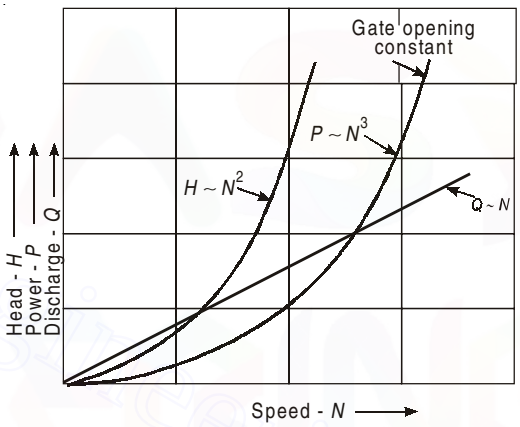


Figure 24.15 *Q v/s N; H_m v/s N; and P v/s N curves of a centrifugal pump*

24.17 LIMITATION OF SUCTION LIFT

From Eq. 24.7 the absolute pressure head at the inlet to the pump may be expressed as

$$\frac{p_a}{w} + \frac{p_s}{w} = \frac{p_a}{w} - \left[\frac{V_s^2}{2g} + h_s + h_{fs}\right] \qquad \ldots(24.47)$$

where p_a is the atmospheric pressure acting on the free liquid surface in the sump, p_s is the gage pressure at the inlet to the pump, V_s is the velocity of flow in the suction pipe, h_s is the suction lift and h_{fs} is the head lost in the foot valve, strainer and the suction pipe. It is not possible to create at the pump inlet, an absolute pressure lower than the vapour pressure of the liquid. Thus if p_v is the vapour pressure of the liquid in absolute units, then in the limiting case $(p_a + p_s) = p_v$, and hence from Eq. 24.47 the limiting value of the suction lift h_s is obtained as

$$h_s = \left(\frac{p_a - p_v}{w}\right) - \frac{V_s^2}{2g} - h_{fs} \qquad \ldots(24.48)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The suction lift should in no case be more than that given by Eq. 24.48, because a greater h_s may result in a rapid vaporization of the liquid due to the reduction of the pressure, which may ultimately lead to cavitation.

24.18 NET POSITIVE SUCTION HEAD (NPSH)

The *net positive suction head* (or NPSH) is defined as the absolute pressure head at the inlet to the pump, minus the vapour pressure head (in absolute units) corresponding to the temperature of the liquid pumped, plus the velocity head at this point. Thus

$$\text{NPSH} = \left(\frac{p_a}{w} + \frac{p_s}{w}\right) - \frac{p_v}{w} + \frac{V_s^2}{2g} \qquad \ldots(24.49)$$

Introducing the value of (p_s/w) from Eq. 24.47 in the above expression it becomes

$$\text{NPSH} = \frac{p_a}{w} - \frac{p_v}{w} - h_s - h_{fs} \qquad \ldots(24.50)$$

The right hand side of Eq. 24.50 also represents the *total suction head* H_{sv}, i.e.,

$$H_{sv} = \frac{p_a}{w} - \frac{p_v}{w} - h_s - h_{fs} \qquad \ldots(24.51)$$

and hence

$$\text{NPSH} = H_{sv} \qquad \ldots(24.51a)$$

In other words, NPSH may also be defined as the head required to make the liquid to flow through the suction pipe to the impeller.

The term NPSH is very commonly used in the pump industry. For any pump installation a distinction is usually made between the required NPSH and the available NPSH. The required NPSH varies with the pump design, the speed of the pump and its capacity. The required NPSH for a pump may be determined experimentally as indicated below and its magnitude must be given by the manufacturer of the pump. When the pump is installed the available NPSH can be determined from the above noted expression. Both the available and the required NPSH vary with the discharge in such a manner that with the increase in the discharge the available NPSH decreases but the required NPSH increases. In order to have cavitation free operation of a centrifugal pump the available NPSH should be more than the required NPSH. It may also be mentioned that the NPSH requirements for a given pump as determined experimentally, by definition are tied to the size of the suction pipe. Therefore, the available NPSH of a given installation should be based on the same size of the suction pipe.

For determining the required NPSH, the pump is operated at the designed speed with different suction lifts and for each setting of the pump, the head v/s discharge and efficiency v/s discharge characteristics are obtained. In each case it will be observed that with the decrease in the head the discharge increases upto a certain value of the head. With further reduction in the head there is only slight increase in the discharge till a certain value of discharge is reached, when further reduction in head causes no increase in the discharge. In other words, at a certain value of discharge there is an abrupt reduction in the head developed by the pump. Further corresponding to this discharge an abrupt reduction in the efficiency of the pump also occurs. The abrupt reduction in the head as well as the efficiency of the pump indicates the inception of the cavitation. As such the value of the discharge at which there is an abrupt reduction in the head as well as the efficiency of the pump is the maximum

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

discharge which the pump is able to provide without cavitation. For the different settings of the pump the values of such maximum discharges are determined and corresponding to each of these discharges the value of NPSH is also computed by Eq. 24.49, which thus represents the required NPSH for each setting of the pump. Usually the values of the required NPSH are plotted against the corresponding values of the discharge to obtain a curve of required NPSH for the pump. Further if a pump is required to be operated at different speeds then by adopting the same procedure, for each of the different speeds a curve for required NPSH may be obtained.

24.19 CAVITATION IN CENTRIFUGAL PUMPS

If the pressure at the suction side of the pump drops below the vapour pressure of the liquid then the cavitation may occur. The cavitation in a pump can be noted by a sudden drop in efficiency and head.

24.19.1 Thoma's Cavitation Factor

As in the case of turbines, for pumps also, Thoma's cavitation factor is used to indicated whether cavitation will occur. For pumps, Thoma's cavitation factor is defined as

$$\sigma = \frac{[(p_a - p_v)/w] - (h_s + h_{fs})}{H_m} \quad \ldots(24.52)$$

By comparing Eq. 24.52 with 24.50 and 24.51, cavitation factor may also be expressed as

$$\sigma = \frac{\text{NPSH}}{H_m} = \frac{H_{sv}}{H_m} \quad \ldots(24.53)$$

The cavitation will occur if the value of σ is less than the critical value, σ_c at which the cavitation just begins. The critical value σ_c is that at which a drop in the efficiency or head or some other property begins. The value of σ_c for any specified operating conditions depends upon the design of the particular pump, and in any important installation it should be determined experimentally from the model tests. However, σ_c depends on the specific speed of a pump and for the pumps of normal design the following empirical relation may be used to determine σ_c.

$$\sigma_c = 0.103\ (N_s/1000)^{4/3} \quad \ldots(24.54)$$

where N_s is the specific speed of the pump defined by Eq. 24.36.

Equation 24.52 indicates that the value of σ can be increased by reducing the suction lift h_s. In some cases, especially when hot liquids are to be handled, the pump may have to be installed either at the liquid surface or even below the liquid surface in the sump. In the first case, $h_s = 0$, but in the second case, $h_s < 0$, which indicates that there is positive pressure at the inlet of the pump. A higher value of σ also ensures that there is sufficient NPSH available.

24.10.2 Suction Specific Speed

In addition to Thoma's cavitation factor, suction specific speed is another cavitation parameter, which is commonly used in the case of pumps to indicate whether the cavitation will occur. The *suction specific speed S* may be defined as the speed in r.p.m. of a geometrically similar pump which under corresponding conditions would deliver 1 litre of liquid per second when the total suction head is 1 m (in absolute units). According to this definition the expression for suction specific speed may be obtained by replacing the manometric head H_m in Eq. 24.36 for the specific speed by the total suction head H_{sv}. Thus

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$S = \frac{N\sqrt{Q}}{H_{sv}^{3/4}} \qquad \ldots(24.55)$$

Further by combining Eqs 24.36, 24.53 and 24.55, the following relationship between σ, N_s and S may be obtained

$$\sigma = \left(\frac{N_s}{S}\right)^{4/3} \qquad \ldots(24.56)$$

For most of the centrifugal and propeller pumps for cavitation free flow the limiting value of the suction specific speed have been found to lie between 4700 and 6700 when S is expressed in [r.p.m./(litres/second)$^{1/2}$/m$^{3/4}$] units.

By having the same value of the Thoma's cavitation factor σ or the suction specific speed S for the model and the prototype pumps the similarity in respect of cavitation can be established. However for the pumps the suction specific speed is a more pertinent cavitation parameter for ensuring similarity in the model and the prototype pumps in respect of cavitation. This is so because although both σ and S depend on the inlet flow conditions, σ also depends on the specific speed N_s of the pump. As such a constant value of σ can be obtained only if in addition to the inlet flow conditions, the N_s is also held constant and only then it will ensure similar cavitation conditions. On the other hand suction specific speed is independent of the specific speed of the pump and therefore a constant value of S alone will ensure similar cavitation conditions in the model and the prototype pumps. For instance, if the specific speed of pump is changed by changing the outside diameter of the impeller, leaving the inlet flow conditions unchanged, it can be assumed that the cavitation conditions remain the same. The suction specific speed S will not change since the inlet flow conditions remain the same, but σ will change since its value depends on the head H and hence on the outer diameter of the impeller. As such even if the value of σ is not same, as long as the value of S is same for the model and prototype pumps, similar cavitation conditions can be ensured.

24.20 COMPUTATION OF THE TOTAL HEAD OF PUMPING—SYSTEM HEAD CURVES

The total head of pumping consists of the following components.

(i) Static head H_s, which is the vertical distance between the liquid surfaces in the sump and the tank to which the liquid is delivered by the pump.

(ii) Velocity head ($V^2/2g$) at the point of discharge.

(iii) Head loss H_L, which includes the head loss due to friction in suction and delivery pipes as well as head loss in valves, bends and all such appurtenances both on the suction and the delivery sides.

Thus the total head H against which the pump has to operate is given by

$$H = H_s + H_L + \frac{V^2}{2g} \qquad \ldots(24.57)$$

With the pump running, if the discharge of the pump is more than the inflow, the level of the liquid in the sump would keep falling. By this the suction head component in the total head would keep increasing. Reverse will be the case when the inflow is more than the discharge by the pump.

By varying the size of the suction and delivery pipes in a pumping system the velocity of flow would vary which in turn would cause a change in the velocity head which varies as the square of the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

velocity of flow. This also causes a variation in the frictional losses because these also vary as the square of the velocity of flow or flow rate.

Thus the total head of pumping varies with the flow rate or discharge of the pump. At the stage of planning it is therefore necessary to compute the total head of pumping over a range of flow rates or discharges, for different variations in the levels of liquid in the sump and for different options of piping sizes and layouts. The values of the total heads of pumping so computed are plotted against the flow rates or discharges. A plot of total head of pumping versus flow rate of discharge is known as *system head curve* for the pumping system. Figure 24.16 shows a typical system head curve for a pumping system with a given static head and a given piping size and layout.

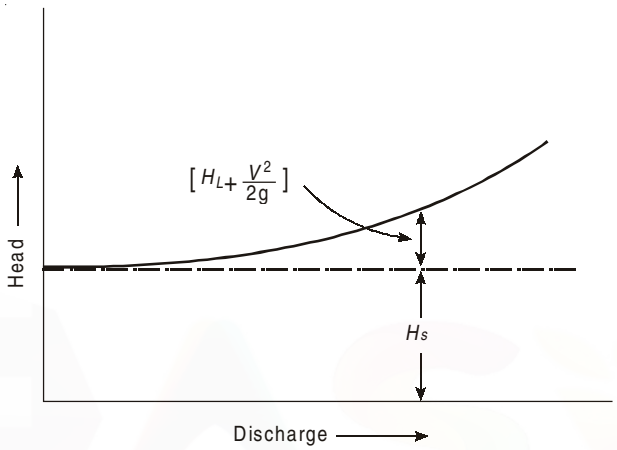


Figure 24.16 *System head curve for a pumping system.*

With an increase only in the static head the new system head curve will be a curve shifted parallel upwards as shown in Fig. 24.17. Further for a smaller size of piping the parabolic portion in the system head curve will be steeper as shown in Fig. 24.18.

From the system head curves for a pumping system with variations in static levels and different piping sizes and layouts, one can know what the total head would be for the most average operating conditions, which then can be specified as the total head of pumping.

24.20.1 Operating Point or Operating Range of a Centrifugal Pump

The *operating point* of a centrifugal pump is the point of intersection of the system head curve with the head-discharge (H v/s Q) characteristic curve of the pump. Thus for obtaining the operating point the

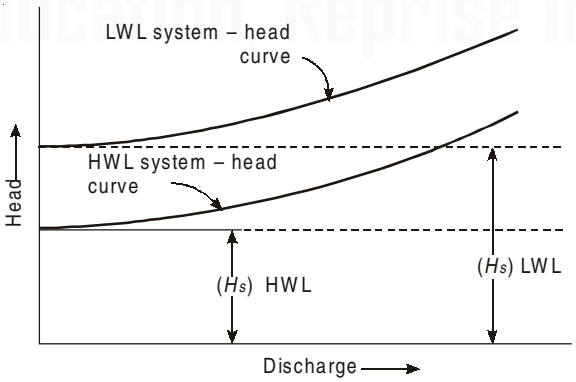


Figure 24.17 *System head curves for low water level (LWL) and high water level (HWL) in the sump.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

system head curve and the head-discharge characteristic curve of the pump are plotted on the same sheet as shown in Fig. 24.19. The head-discharge characteristic curve of a centrifugal pump is a drooping parabola with the pump discharge being less when the head is more. When the pump is put into a system it meets the head as demanded by the system, which is as per the system head curve. At the point of intersection *P* of the system head curve with the head discharge curve of the pump, since the head demanded by the system for a certain flow rate or discharge is equal to the operating head of the pump at the same flow rate or discharge, point *P* represents the operating point of the pump.

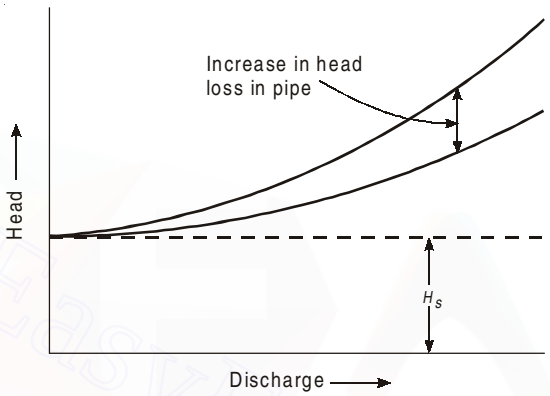


Figure 24.18 *System head curves with change in pipe sizes*

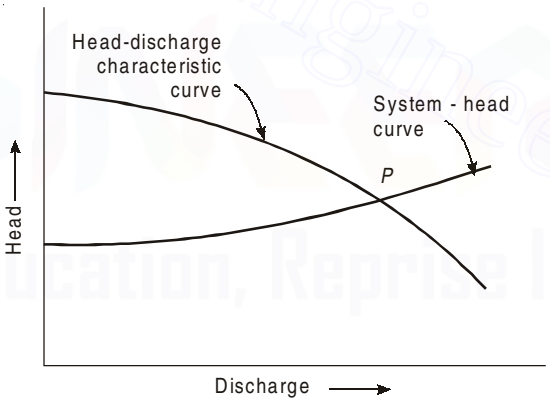


Figure 24.19 *Operating point of a centrifugal pump*

As indicated earlier the system head curve of a pumping system will change due to any change made in the system. If the system head curve changes the operating point will shift. The most significant change that may occur in a pumping system is due to variation in the level of liquid in the sump. There will be different system head curves corresponding to low water level (LWL) and high water level (HWL) in the sump as shown in Fig. 24.20. Thus if the level of liquid in the sump depletes during pumping from HWL to LWL, the operating point of the pump would vary from a low-head and high-discharge point P_1 to high-head and low-discharge point P_2 as shown in Fig. 24.20. The range between points P_1 and P_2 is known as the *operating range* of the pump due to fluctuations in the level of liquid in the sump.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

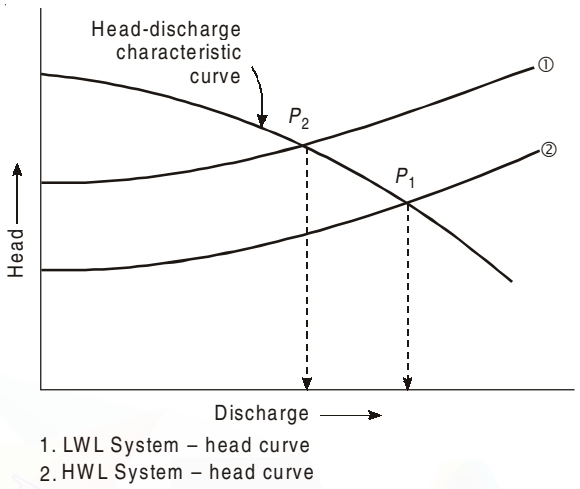


Figure 24.20 *Operating range of a centrifugal pump.*

24.20.2 Selection of a Pumping Unit

With a given or known system head curve it is required to select a pump to deliver the anticipated flow at the required height. For this the system head curve and the head-discharge characteristic curve of the pump are plotted on the same sheet and the operating point is found (see Fig. 24.19). The operating point P gives the head and discharge or flow rate at which the pump will be operating. As such the pump selected should be such that the operating point is as close as possible to the best efficiency point.

24.20.3 Pumps Operated in Series

When pumps are operated in series for producing higher head, the operating point is the point of intersection of the combined head-discharge characteristic curve with the system head curve as shown in Fig. 24.21. The combined head-discharge characteristic curve of the pumps operated in series is obtained by reading against different discharges, the values of the heads obtained from the head-discharge characteristic curves of the individual pumps, and plotting the sum of the head values against the respective discharges as shown in Fig. 24.21. The total head is thus equal to the head added by each pump corresponding to discharge at the operating point. It may, however, be noted that since the point of intersection of the system head curve with the combined head-discharge characteristic curve is at a discharge larger than that at the point of intersection of system head curve with the head-discharge characteristic curve of a single pump, the head developed by each of the pumps when operating in series at the point of operation gets reduced. Thus as shown in Fig. 24.21 when only one pump is operating the head at the operating point P_1 is H_1, and when two identical pumps are operated in series the point of operation gets shifted upwards to P_2 corresponding to which the total head developed is H_2, but $H_2 < 2H_1$. It means that to double the head it is not adequate to commission two identical pumps in series operation. For this one must study what combination of pumps with different head-discharge characteristic curves can give such a combined head-discharge characteristic curve which will have its intersection with the system head curve at a point having total head as desired.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

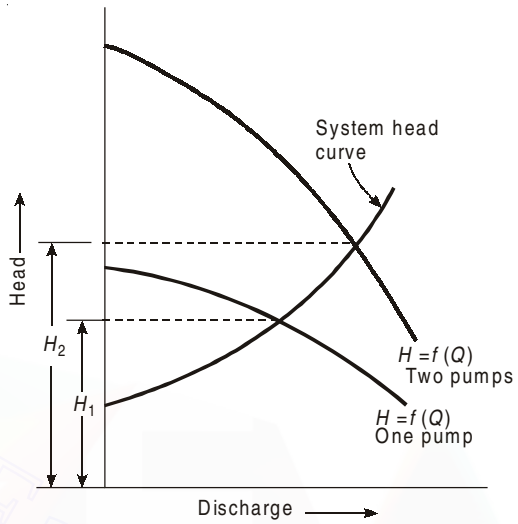


Figure 24.21 *Head-discharge characteristic curves for pumps operated in series*

24.20.4 Pumps Operated in Parallel

When pumps are operated in parallel for deliverying relatively large quantity of liquid against relatively small heads, the operating point is again the point of intersection of the combined head-discharge characteristic curve with the system head curve as shown in Fig. 24.22. The combined head-discharge characteristic curve of the pumps operated in parallel is obtained by reading against different heads, the values of the discharges obtained from the head-discharge characteristic curves of the individual pumps, and plotting the sum of the discharge values against the respective heads, as shown in Fig. 24.22. The total discharge is thus equal to the discharge added by each pump corresponding to head at the operating point. It may, however, be noted that since the point of intersection of the system head curve with the combined head-discharge characteristic curve is at a head higher than that at the point of intersection of system head curve with the head-discharge characteristic curve of a single pump, the discharge of each of the pumps when operating in parallel at the point of operation gets reduced. Thus as shown in Fig. 24.22 when only one pump is operating the discharge at the operating point P_1 is Q_1, and when two identical pumps are operated in parallel the point of operation gets shifted upwards to P_2 corresponding to which the total discharge is Q_2, but $Q_2 < 2Q_1$. It means that to double the

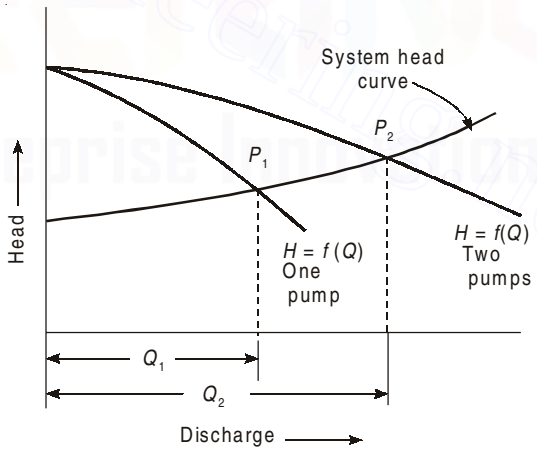


Figure 24.22 *Head-discharge characteristic curves for pumps operated in parallel*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

discharge capacity of pumping, it is not adequate to commission two identical pumps in parallel operation. For this one must study what combination of pumps with different head-discharge characteristic curves can give such a combined head-discharge characteristic curve which will have its intersection with the system head curve at a point having total discharge as desired.

24.21 PRIMING DEVICES

As stated earlier, before starting a centrifugal pump, it must be primed. Small pumps are usually primed by pouring the liquid through a funnel into the casing from some external source. The air vent provided in the casing is opened to facilitate the exit of the air. When all the air has been removed from the suction pipe and the pump casing, the air vent is closed and the pump is primed.

Large pumps are usually primed by evacuating the casing and the suction pipe with the aid of an air pump or a stream ejector, the liquid is thus sucked into the suction pipe from the sump.

In some pumps, their internal construction is such that special arrangements containing a supply of liquid are provided in the suction pipe, which facilitate automatic priming of the pump. Such pumps are known as 'self priming pumps'.

24.22 CENTRIFUGAL PUMP-TROUBLES AND REMEDIES

Some of the troubles commonly experienced during the operation of the centrifugal pumps and the remedial measures to be taken for the same are as listed below:

1. *Pump fails to start pumping*. (i) Pump is probably not properly primed. Reprime the pump, opening the air vent until a steady and unbroken stream of liquid is obtained.

(ii) Total static head is probably much higher than that for which the pump is designed. Check the same with accurate vacuum and pressure gages, or may be determined by actual measurement the difference in elevation between pump and suction liquid level, and also between pump and point of discharge. Add to this static head the loss of head due to friction in pipes and that in the other fittings used in the installation.

(iii) Wrong direction of rotation of the impeller. Arrow on the pump case shows the proper direction of rotation.

(iv) Impeller may be clogged. Examine carefully for solids or foreign matter lodged in the impeller.

(v) Suction lift too high. Check with vacuum gage or by actual measurement where possible, adding to the static suction lift the loss of head in the pipe and fittings.

(vi) Strainer or suction line may be clogged. This would cause an excessive suction lift.

(vii) Speed may be too low. Check the speed with a teacheometer and compare it with the one given on the name plate of the pump. When the pump is being driven by electric motor, check up to see whether the voltage may not be too low.

2. *Pump working but not upto capacity and pressure*. (i) Air may be leaking into pump through suction line or through stuffing boxes.

(ii) Speed may be too low.

(iii) Discharge head may be higher than anticipated.

(iv) Suction lift may be too high.

(v) Foot valve or end of suction pipe may not have sufficient submergence, or it may be partly clogged or entirely too small.

(vi) Impeller may be partly clogged or too small in diameter.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(vii) Rotation may be in the wrong direction.

(viii) Wearing rings may be worn, impeller may be damaged, shaft may be loose, stuffing box packing may be defective.

(ix) If hot or volatile liquids are being pumped, there may not be sufficient positive head on the suction.

3. *Pump starts and then stops pumping.* (i) Improperly primed or leaky suction line.

(ii) Air pockets in suction line.

(iii) Air may be entering suction pipe because of liquid being delivered in suction tank or sump too near the pump suction line.

(iv) Suction lift may be too high.

4. *Pumps take too much power*:

(i) Speed may be too high.

(ii) Head may be too low and pump delivers too much liquid.

(iii) Liquid may have too high a specific gravity.

(iv) Pump may be operating in wrong direction.

(v) Shaft may be bent, impeller may be rubbing on casing, stuffing boxes may be too light, wearing rings may be worn.

ILLUSTRATIVE EXAMPLES

Example 24.1. *A centrifugal pump has an impeller 0.5 m outer diameter and when running at 600 r.p.m. discharges water at the rate of 8000 litres/minute against a head of 8.5 m. The water enters the impeller without whirl and shock. The inner diameter is 0.25 m, and the vanes are set back at outlet at an angle of 45° and the area of flow which is constant from inlet to outlet of the impeller is 0.06 m^2. Determine (a) the manometric efficiency of the pump, (b) the vane angle at inlet, and (c) the least speed at which the pump commences to work*:

Solution

$$V_f = V_{f_1}$$

$$= \frac{8000 \times 10^{-3}}{60 \times 0.06} = 2.22 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 0.25 \times 600}{60} = 7.85 \text{ m/s}$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$= \frac{\pi \times 0.50 \times 600}{60} = 15.71 \text{ m/s}$$

From the inlet velocity triangle of Fig. 24.5

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\tan\theta = \left(\frac{V_f}{u}\right)$$

$$= \left(\frac{2.22}{7.85}\right) = 0.2828$$

$$\therefore \quad \theta = 15° 47'.$$

From the outlet velocity triangle

$$V_{w1} = \left[u_1 - \left(\frac{V_{f_1}}{\tan 45°}\right)\right]$$

$$= (15.71 - 2.22)$$

$$= 13.49 \text{ m/s}$$

From Eq. 24.18

$$\eta_{\text{mano}} = \frac{gH_m}{v_{w_1} u_1}$$

$$= \frac{9.81 \times 8.5}{13.49 \times 15.71} = 0.3935 \text{ or } 39.35\%$$

For the least speed, from Eq. 24.27

$$\left[\left(\frac{\pi N}{60}\right)^2 (D_1^2 - D^2)\right] = 2gH_m$$

or $$\left(\frac{\pi N}{60}\right)^2 [(0.50)^2 - (0.25)^2] = (2 \times 9.81 \times 8.5)$$

$$\therefore \quad N = 570 \text{ r.p.m.}$$

Example 24.2. *A centrifugal pump has to discharge 225 litres of water per second against a head of 25 m when the impeller rotates at a speed of 1 500 r.p.m. Determine (a) the impeller diameter, and (b) the vane angle at the outlet edge of the impeller. Assume that $h_{mano} = 0.75$; the loss of head in pump in metres due to fluid resistance is 0.03 V_1^2, where V_1 m/s is the absolute velocity of water leaving the impeller, the area of the impeller outlet surface is (1.2 D_1^2) m^2, where D_1 is the impeller diameter in m, and water enters the impeller without whirl.*

Solution

$\eta_{\text{mano}} = (gH/V_{w1}u_1)$; or $0.75 = [25 \times g)/V_{w1}u_1]$

$$\therefore \quad [(V_{w1}u_1)/g] = 33.33 \text{ m} \qquad \text{...(i)}$$

From Eq. 24.6, the manometric head is $H_m = (V_{w1}u_1/g)$ – losses of head in the pump.

$\therefore$ Losses in the pump

$$= (33.33 - 25) = 8.33 \text{ m}$$

Thus $$0.03\ V_1^2 = 8.33$$

$$\therefore \quad V_1 = 16.66 \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_{f1} = \frac{Q}{A} = \frac{225\times10^{-3}}{1.2\times D_1^2} = \frac{0.188}{D_1^2}\ \text{m/s}$$

$$u_1 = [(\pi D_1 N)/60] = 78.54\, D_1\ \text{m/s}$$

Substituting the above value of u_1 in (i)

$$V_{w1} = \frac{33.33\times9.81}{78.54 D_1} = \frac{4.16}{D_1}\ \text{m/s}$$

Now from the outlet velocity triangle of Fig. 24.5

$$V_{f1} = \left(V_1^2 - V_{w_1}^2\right)^{1/2}$$

or $(0.188/D_1^2) = [(16.66)^2 - (4.16/D_1)^2]^{1/2}$

or $278 D_1^4 - 17.31\, D_1^2 - 0.035 = 0$

$\therefore$ $D_1 = 0.253\ 4$ m or 253.4 mm.

Thus $V_{f1} = 2.93$ m/s; $u_1 = 19.90$ m/s

and $V_{w1} = 16.42$ m/s

$\tan\phi = [V_{f_1}/(u_1 - V_{w_1})] = [2.93/(19.90 - 16.42)]$

$= 0.8420$

$\therefore$ $\phi = 40°6'$.

Example 24.3. *Find the power required to drive a centrifugal pump which delivers 40 litres of water per second to a height of 20 m through a 150 mm diameter and 100 m long pipeline. The overall efficiency of pump is 70% and Darcy's f = 0.06 for the pipeline. Assume inlet losses in suction pipe equal to 0.33 m.*

Solution

$Q = 40$ l/s $= 0.04$ m^3/s ; $H_s = (h_s + h_d) = 20$ m;

$d_s = d_d = 150$ mm $= 0.15$ m ; $h_{in} = 0.33$ m ; $\eta_0 = 70\%$

$\therefore$ $V_s = V_d = \dfrac{0.04}{(\pi/4)(0.15)^2} = 2.264$ m/s

and $\dfrac{V_d^2}{2g} = 0.26$ m

$$(h_{fs} + h_{fd}) = \frac{fLV^2}{2gd}$$

or $(h_{fs} + h_{fd}) = \dfrac{0.06\times100\times(2.264)^2}{2\times9.81\times0.15} = 10.45$ m

The manometric head H_m is given by

$$H_m = h_s + h_d + h_{fs} + h_{fd} + h_{in} + \frac{V_d^2}{2g}$$

$\therefore$ $H_m = (20 + 10.45 + 0.33 + 0.26)$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$= 31.04$ m

Power required to drive the pump is given by

$$P = \frac{wQH_m}{\eta_0}$$

$$\therefore \quad P = \frac{9810 \times 0.04 \times 31.04}{0.70}$$

$= 17400$ W $= 17.4$ kW

Example 24.4. *Find the rise in pressure in the impeller of a centrifugal pump through which water is flowing at the rate of 15 litres per second. The internal and external diameters of the impeller are 200 mm and 400 mm respectively. The widths of the impeller at inlet and outlet are 16 mm and 8 mm respectively. The pump is running at 1200 r.p.m. The water enters the impeller radially at inlet and impeller vane angle at outlet is 30°. Loss of head through the impeller may be assumed as 1.15 m*

Solution

$Q = 15$ l/s $= 0.015$ m^3/s

$D = 200$ mm $= 0.2$ m; $D_1 = 400$ mm $= 0.4$ m

$B = 16$ mm $= 0.016$ m; $B_1 = 8$ mm $= 0.008$ m

$N = 1200$ r.p.m; $\phi = 30°$; $h_{Li} = 1.15$ m

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 0.2 \times 1200}{60} = 12.57 \text{ m/s.}$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$= \frac{\pi \times 0.4 \times 1200}{60} = 25.13 \text{ m/s}$$

$$V_f = \frac{0.015}{(\pi \times 0.2 \times 0.016)} = 1.49 \text{ m/s}$$

$$V_{f1} = \frac{0.015}{(\pi \times 0.4 \times 0.008)} = 1.49 \text{ m/s}$$

i.e., $V_f = V_{f1}$

As indicated in Example 24.6, the pressure rise in the impeller is given by

$$\frac{p_1 - p}{w} = \left[\frac{1}{2g}\left(V_f^2 + u_1^2 - V_{f_1}^2 \text{cosec}^2\phi\right)\right] - h_{Li}$$

or

$$\frac{p_1 - p}{w} = \frac{1}{2 \times 9.81}\left[(1.49)^2 + (25.13)^2 - (1.49)^2(\text{cosec } 30°)^2\right] - 1.15$$

$= 30.7$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 24.5. *A centrifugal pump has the following characteristics: outer diameter of impeller = 800 mm; width of impeller vanes at outlet = 100 mm; angle of impeller vanes at outlet = 40°. The impeller runs at 550 r.p.m and delivers 0.98 cubic metres of water per second under an effective head of 35 m. A 500 kW motor is used to drive the pump. Determine the manometric, mechanical and overall efficiencies of the pump. Assume water enters the impeller vanes radially at inlet.*

Solution

$Q = 0.98\ m^3/s$; $D_1 = 800\ mm = 0.8\ m$; $N = 550$ r.p.m;
$B_1 = 100\ mm = 0.1\ m$; $\phi = 40°$; $H_m = 35\ m$
$P = 500\ kW$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$= \frac{\pi \times 0.8 \times 550}{60} = 23.04\ m/s$$

$$V_{f1} = \frac{Q}{\pi D_1 B_1}$$

$$= \frac{0.98}{\pi \times 0.8 \times 0.1} = 3.90\ m/s$$

$$V_{w1} = (u_1 - V_{f1} \cot \phi)$$
$$= (23.04 - 3.90 \cot 40°)$$
$$= 18.39\ m/s$$

Manometric efficiency is given by

$$\eta_{mano} = \frac{gH_m}{V_{w_1} u_1}$$

$$= \frac{9.81 \times 35}{18.39 \times 23.04} = 0.81 \text{ or } 81\%$$

Overall efficiency is given by

$$\eta_0 = \frac{wQH_m}{P}$$

$$= \frac{9810 \times 0.98 \times 35}{500 \times 10^3} = 0.67 \text{ or } 67\%$$

Mechanical efficiency is given by

$$\eta_{mech} = \frac{\eta_0}{\eta_{mano}}$$

$$= \frac{0.67}{0.81} = 0.83 \text{ or } 83\%.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 24.6. *Show that the pressure rise in the impeller of a centrifugal pump is given by* $[V_f^2 + u_1^2 - V_{f1}^2 \text{ cosec}^2 \phi/2g]$ *provided the frictional and other losses in the impeller are neglected.*

Solution

Applying Bernoulli's equation between inlet and outlet tips of the impeller

$$\frac{p}{w} + \frac{V^2}{2g} = \frac{p_1}{w} + \frac{V_1^2}{2g} - \frac{V_{w_1} u_1}{g}$$

For radial entry $V = V_f$, thus

$$\frac{p_1 - p}{w} = \frac{V_f^2}{2g} + \frac{V_{w_1} u_1}{g} - \frac{V_1^2}{2g}$$

From the outlet velocity triangle

$$V_{w1} = (u_1 - V_{f1} \cot \phi)$$

$$V_1^2 = V_{f_1}^2 + V_{w_1}^2 = \left[V_{f_1}^2 + \left(u_1 - V_{f_1} \cot \phi \right)^2 \right]$$

since $(1 + \cot^2 \phi) = \text{cosec}^2 \phi$

$$V_1^2 = V_{f_1}^2 \text{cosec}^2 \phi + u_1^2 - 2u_1 V_{f1} \cot \phi$$

Hence

$$\frac{p_1 - p}{w} = \frac{V_f^2}{2g} + \frac{u_1 (u_1 - V_{f_1} \cot \phi)}{g} - \frac{V_{f_1}^2 \text{cosec}^2 \phi + u_1^2 - 2u_1 V_{f_1} \cot \phi}{2g}$$

$$\therefore \quad \frac{p_1 - p}{w} = \left[\frac{V_f^2 + u_1^2 - V_{f_1}^2 \text{cosec}^2 \phi}{2g} \right]$$

Example 24.7. *Prove that, in general, for a centrifugal pump running at speed N and giving a discharge Q, the manometric head is expressible in the form,* $H_m = [AN^2 + BNQ + CQ^2]$ *where A, B and C are constants.*

Solution

If the loss of head in the pump is neglected, the manometric head is given by the pressure rise through the impeller, together with a certain percentage of the kinetic head at the exit of the impeller which is recovered in the volute chamber or the diffuser ring.

As derived in Example 24.6.

$$\text{Pressure rise} = \frac{1}{2g} [V_f^2 + u_1^2 - V_{f_1}^2 \text{ cosec}^2 \phi]$$

Let the percentage of kinetic head recovered in diffuser ring = $(k V_1^2)$ m. Then it can be shown that

$$k V_1^2 = \frac{c}{2g} \left[V_{f_1}^2 + u_1^2 - 2u_1 V_{f_1} \cot \phi + V_{f_1}^2 \cot^2 \phi \right]$$

where c is a constant

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\therefore \quad H_m = \frac{1}{2g}\left[u_1^2(1+c) - 2cu_1V_{f_1}\cot\phi + V_{f_1}^2 \times (c + c\cot^2\phi - \operatorname{cosec}^2\phi) + V_f^2\right]$$

Assuming $V_f = V_{f1}$,

$$H_m = \frac{1}{2g}\left[Au_1^2 + Bu_1V_{f_1} + CV_{f_1}^2\right]$$

Since $u_1 = (\pi D_1 N/60).\ \therefore u_1 \sim N$

and $V_{f1} = (Q/A_1);\quad \therefore V_{f_1}^2 \sim Q^2$

$$\therefore \quad H_m = \frac{1}{2g}\left[AN^2 + BNQ + CQ^2\right]$$

Example 24.8. *A centrifugal pump has an impeller 0.29 m diameter running at 960 r.p.m. with an effective outlet vane angle of 28°. The velocity of flow (assumed uniform throughout the system) is 2 m/s. The static suction lift is 2.8 m. The energy losses in metres of water are: in suction pipe, 0.61 m; in impeller, 0.49 m; in volute casing, 0.88 m. From these particulars calculate the readings of vacuum or pressure gages placed (i) at inlet to the pump, (ii) at impeller outlet (in clearance space between impeller and volute), (iii) at pump outlet or delivery flange, 0.2 m above the centerline of the pump.*

Solution

From Eq. 24.7 the pressure head at the inlet to the pump is given by

$$\frac{p_s}{w} = -\left[\frac{V_s^2}{2g} + h_s + h_{f_s}\right]$$

or

$$\frac{p_s}{w} = -\left[\left(\frac{2\times 2}{(2\times 9.81)}\right) + 2.8 + 0.61\right]$$

$$= -3.614 \text{ m}$$

From Eq. 24.8

$$\frac{p_s}{w} + \frac{V_s^2}{2g} + \frac{v_{w1}u_1}{g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_{Li}$$

Now $u_1 = [(\pi D_1 N)/60] = [(\pi \times 0.29 \times 960)/60]$

$= 14.58$ m/s

$V_{w1} = (u_1 - V_{f1}\cot\phi) = (14.58 - 2\cot 28°)$

$= 10.82$ m/s

$V_1 = \left(V_{w_1}^2 + V_{f1}^2\right)^{1/2} = [(10.82)^2 + (2)^2]^{1/2}$

$= 11.00$ m/s

Thus by substitution in the above eqution

$$-3.614 + \left(\frac{2\times 2}{2\times 9.81}\right) + \left(\frac{10.82\times 14.58}{9.81}\right) = \frac{p_2}{w} + \left(\frac{11\times 11}{2\times 9.81}\right) + 0.49$$

$\therefore$ Pressure head at impeller outlet is

$$(p_2/w) = 6.01 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From Eq. 24.9

$$\frac{p_2}{w} + \frac{V_1^2}{2g} = \frac{p_d}{w} + \frac{V_d^2}{2g} + h + h_{LC}$$

or $$6.01 + \left(\frac{11\times 11}{2\times 9.81}\right) = \frac{p_d}{w} + \left(\frac{2\times 2}{2\times 9.81}\right) + 0.20 + 0.88$$

or $$\frac{p_d}{w} = 10.89 \text{ m}$$

i.e., the pressure head at pump outlet is 10.89 m.

Example 24.9. *A centrifugal pump lifts water against a static head of 40 m, of which 4 m is suction lift. The suction and delivery pipes are both 150 mm diameter; the head loss in the suction pipe is 2.3 m and in the delivery pipe, 7.4 m. The impeller is 420 mm diameter and 25 mm wide at the mouth; it revolves at 1200 r.p.m and its effective vane angle at exit is 35°. If* $\eta_{mano} = 82\%$ *and* $\eta_0 = 72\%$*, determine the discharge delivered by the pump and power required to drive the pump. Also find the pressure head indicated at the suction and delivery branches of the pump.*

Solution

If V_p = Velocity in pipes,

then discharge $Q = [\pi(0.15)^2/4]\, V_p = 0.0177\, V_p$

Also $Q = \pi D_1 B_1 \times V_{f1}$ (neglecting the vane thickness)

or $Q = (\pi \times 0.42 \times 0.025)\, V_{f1} = 0.0177\, V_p$

or $V_{f1} = 0.537\, V_p \approx 0.54\, V_p$

$u_1 = (\pi D_1 N/60) = (\pi \times 0.42 \times 1200/60)$

$= 26.39$ m/s

From Eq. 24.17, since $V_d = V_p$

$$H_m = (h_s + h_d) + h_{fs} + h_{fd} + \frac{V_p^2}{2g}$$

or $$H_m = 40 + 2.3 + 7.4 + \frac{V_p^2}{2g} = 49.7 + \frac{V_p^2}{2g}$$

But $$\eta_{\text{mano}} = \frac{gH_m}{V_{w_1} u_1}; \text{ or } 0.82 = \frac{9.81\times H_m}{u_1(u_1 - V_{f_1}\cot\phi)}$$

or $$H_m = \frac{(0.82\times 26.39)(26.39 - 0.54\, V_p \cot 35°)}{9.81}$$

Equating the two values of H_m, we have

$$49.7 + \left(\frac{V_p^2}{2\times 9.81}\right) = \frac{(0.82\times 26.39)(26.39 - 0.54 V_p \times 1.428)}{9.81}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solving this equation, we get

$$V_p = 4.42 \text{ m/s}$$

Hence discharge

$$Q = (0.017\,7 \times 4.42) = 0.078 \text{ m}^3/\text{s}$$

$$H_m = \left(49.7 + \frac{(4.42)^2}{2 \times 9.81}\right) = 50.70 \text{ m}$$

Shaft power $$= \frac{wQH_m}{\eta_0}$$

$$= \frac{9810 \times 0.078 \times 50.70}{0.72}$$

$$= 53\,881 \text{ W} = 53.881 \text{ kW}$$

Since velocity head

$$= \frac{(4.42)^2}{2 \times 9.81} = 0.996 \text{ m} \approx 1 \text{ m}$$

Thus at pump suction the pressure head is given by Eq. 24.7 as

$$\frac{p_s}{w} = -\left(\frac{V_s^2}{2g} + h_s + h_{f_s}\right)$$

$$= -[1.0 + 4 + 2.3] = -7.3 \text{ m}$$

Similarly at pump delivery the pressure head is given by Eq. 24.13 as

$$\frac{p_d}{w} = \left(H_m + \frac{p_s}{w}\right)$$

$$= (50.7 - 7.3) = 43.4 \text{ m}.$$

Example 24.10. *Water is pumped from a low level reservoir through a main pipe pipeline of 0.45 m diameter and 1400 m length. The pump is located at the low level reservoir. At a point along the main line at a distance of 450 m from the high level reservoir, a branch line of 0.3 m diameter and 360 m length takes off to discharge 180 l/s in the atmosphere.*

Level of water surface in high level reservoir = + 30.00 *m*

Level of water in the open end of the 0.3 m diameter branch line = 25.50 *m*

Level of water in the low level reservoir = 18.00 *m*

Darcy's frictional coefficient for both pipes = 0.032

Determine the flow rate into the high level reservoir and the theoretical power of the pump assuming the delivery valve of the pump to be at + 20.00 m.

Solution

Let Q be the total discharge delivered by the pump; Q_1 be the discharge through the branch pipe delivered in the atmosphere; and Q_2 be the discharge delivered to the high level reservoir.

Thus by continuity equation, we have

$$Q = Q_1 + Q_2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

But $Q_1 = 180\ l/s = 0.18\ m^3/s$

$$Q = 0.18 + Q_2 \quad ...(i)$$

From Eqs. 24.7, 24.8 and 24.9, we have with usual notation

$$0 = \frac{p_s}{w} + \frac{V_s^2}{2g} + h_s + h_{fs} \quad ...(ii)$$

$$\frac{p_s}{w} + \frac{V_s^2}{2g} + \frac{V_{w_1} u_1}{g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_{Li} \quad ...(iii)$$

$$\frac{p_2}{w} + \frac{V_1^2}{2g} = \frac{p_d}{w} + (20 - 18 - h_s) + h_{Lc} + \frac{V_d^2}{2g} \quad ...(iv)$$

Adding Eqs. (ii), (iii) and (iv) we get

$$\frac{V_{w_1} u_1}{g} - h_{Li} - h_{Lc} = \frac{p_d}{w} + \frac{V_d^2}{2g} + 2 + h_{fs}$$

or $$H_m = \frac{p_d}{w} + \frac{V_d^2}{2g} + 2 + h_{fs} \quad ...(v)$$

Since the pump is located close to the low level reservoir, h_{fs} may be neglected. Thus from Eq. (v) we have

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = H_m - 2 \quad ...(vi)$$

Let h_f be the head loss due to friction in the main pipeline between the delivery point of the pump and the point where the branch line takes off; h_{f1} be the head loss due to friction in the branch line; and h_{f2} be the head loss due to friction in the main pipeline between the point where the branch line takes off and the high level reservoir.

Thus applying Bernoulli's equation between the delivery point of the pump and the high level reservoir, neglecting the minor losses we get

$$\frac{p_d}{w} + \frac{V_d^2}{2g} + 20 = 30 + h_f + h_{f2}$$

or $$\frac{p_d}{w} + \frac{V_d^2}{2g} = 10 + h_f + h_{f2} \quad ...(vii)$$

Similarly applying Bernoulli's equation between the delivery point of the pump and the open end of the branch line, neglecting the minor losses, we get

$$\frac{p_d}{w} + \frac{V_d^2}{2g} + 20 = 25.5 + h_f + h_{f1}$$

or $$\frac{p_d}{w} + \frac{V_d^2}{2g} = 5.5 + h_f + h_{f1} \quad ...(viii)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

From Eqs (vii) and (viii), we have

$$h_{f_1} - h_{f_2} = 4.5 \qquad \text{...(ix)}$$

The velocity of flow in the branch line is

$$V_1 = \frac{0.18}{(\pi/4)(0.3)^2} = 2.546 \text{ m/s}$$

$$\therefore \quad h_{f_1} = \frac{0.032 \times 360 \times (2.546)^2}{2 \times 9.81 \times 0.3} = 12.687 \text{ m}$$

Thus from Eq. (ix), we get

$$h_{f_2} = (12.687 - 4.5) = 8.187 \text{ m}$$

If V_2 is the velocity of flow in the main pipeline between the branch line and the high level reservoir, then

$$h_{f_2} = \frac{0.032 \times 450 \times V_2^2}{2 \times 9.81 \times 0.45}$$

or $$8.187 = \frac{0.032 \times 450 \times V_2^2}{2 \times 9.81 \times 0.45}$$

$$\therefore \quad V_2 = 2.240 \text{ m/s}$$

Thus $$Q_2 = \frac{\pi}{4} \times (0.45)^2 \times 2.240 = 0.356 \text{ m}^3/\text{s}$$

$$\therefore \quad Q = (0.18 + 0.356) = 0.536 \text{ m}^3/\text{s}$$

If V is the velocity of flow in the main pipeline between the delivery point of the pump and the point where the branch line takes off, then

$$V = \frac{0.536}{(\pi/4)(0.45)^2} = 3.370 \text{ m/s}$$

and $$h_f = \frac{0.032 \times (1400 - 450) \times (3.370)^2}{2 \times 9.81 \times 0.45}$$

$$= 39.104 \text{ m}$$

Thus from Eq. (vii), we have

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = (10 + 39.104 + 8.187) = 57.291 \text{ m}$$

Thus from Eq. (vi), we have

$$H_m = (57.291 + 2) = 59.291 \text{ m}$$

Theoretical power of the pump is given by

$$P = wQH_m$$

in SI units

$$P = \frac{9810 \times 0.536 \times 59.291}{1000} = 311.762 \text{ kW}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In Metric units

$$P = \frac{1000 \times 0.536 \times 59.291}{75} = 423.733 \text{ h.p.}$$

Example 24.11. *A centrifugal pump running at 1400 r.p.m. has the characteristics as given below:*

Discharge Q Litres / second	*12.5*	*18.7*	*25.0*	*31.3*	*37.5*	*43.8*	*50.0*
Head H_m meters	*28.3*	*27.4*	*26.4*	*25.0*	*23.4*	*20.7*	*18.0*
η_0 *(present)*	*65*	*70*	*73*	*74*	*72*	*69*	*63*

Draw the operating characteristics of the pump and determine its specific speed. If the pump is driven by a motor, determine the power of the motor when operating at maximum efficiency.

Solution

The curves of head and efficiency are plotted on a base of discharge shown in the Fig. Ex. 24.11

From the plot, maximum $\eta_0 = 74.1\%$ and the corresponding $H_m = 25.3$ m and $Q = 30.4$ 1/s.

$$\therefore \quad N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}$$

$$= \frac{1400 \times (30.4)^{1/2}}{(25.3)^{3/4}}$$

$$= 684.3$$

$$\eta_0 = \frac{(wQH_m)}{P}$$

$$\therefore \text{ Power } \quad P = \frac{(wQH_m)}{\eta_0}$$

$$= \frac{9810 \times (30.4 \times 10^{-3}) \times 25.3}{0.741}$$

$$= 10\,182 \text{ W} = 10.182 \text{ kW}$$

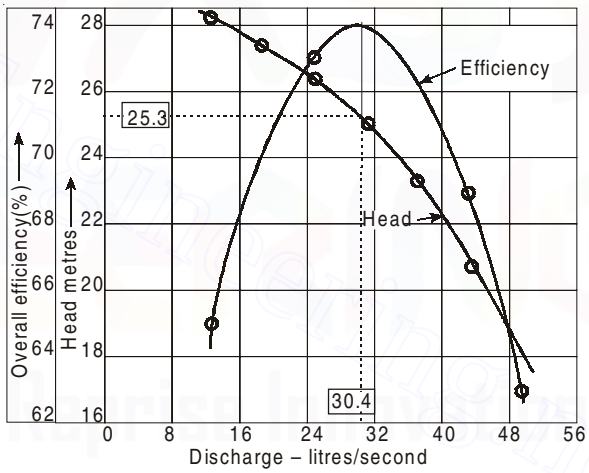


Figure Ex. 24.11

Example 24.12. *Determine the number of the impellers required for a multistage pump to lift 4 200 litres/minute against a total head of 185 m, at a speed of 750 r.p.m. The specific speed is not to exceed 700.*

Solution

The limiting N_s for each impeller = 700

Thus $$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}} \text{ ; or } 700 = \frac{750(4200/60)^{1/2}}{H_m^{3/4}}$$

$\therefore$ Head per stage $= H_m = 18.62$ m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$\therefore$ Number of stages $= \dfrac{185}{18.62} = 10.$

Example 24.13. *Water is to be pumped out of a deep well under a total head of 95 m. A number of identical pumps of design speed 1000 r.p.m. and specific speed 900 r.p.m., with a rated capacity of 150 litres/second are available. How many pumps will be needed and how should they be connected?*

Solution

If H_m is the manometric head developed by each pump then since

$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}; \quad 900 = \frac{1000(150)^{1/2}}{H_m^{3/4}}$$

$$\therefore \quad H_m = 32.5 \text{ m}$$

$\therefore$ Number of pumps needed

$$= \frac{95}{32.5} = 3$$

Since the total head required to be developed is more than the head developed by each pump, the pumps should be connected in series.

Example 24.14. *Two centrifugal pumps have the head and discharge characterstics as follows:*

Discharge Q litres/second	*0*	*4*	*8*	*12*	*16*	*20*	*24*
PUMP 1 Head H_{m_1} (metres)	*50.0*	*51.8*	*50.8*	*48.0*	*42.5*	*32.5*	*18.3*
PUMP2 Head H_{m_2} (metres)	*46.7*	*45.9*	*44.2*	*40.3*	*34.3*	*26.0*	*17.0*

Both pumps are installed together and are required to pump water through a pipe 150 mm diameter having $f = 0.02$. Calculate the heads under which pumps are working and discharge in litres per second pumped by them if:

(a) the pumps are connected in series : static lift is 65 m and the suction and the delivery pipes are 800 m long.

(b) the pumps are connected in parallel ; static lift is 15 m and the suction and the delivery pipes are 360 m long.

Solution

(a) When the pumps are connected in series then neglecting the velocity head in the delivery pipe, the total head required to be developed by both the pumps is

$$H_m = (h_s + h_d) + (h_{fs} + h_{fd})$$

or
$$H_m = 65 + \frac{0.02 \times 800 \times (Q \times 10^{-3})^2}{[\pi(0.15)^2/4]^2 \times 2 \times 9.81 \times 0.15}$$

or
$$H_m = 65 + \frac{Q^2}{57.44} \qquad \text{...(i)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where Q is the discharge in litres/second. The computed values of H_m for different values of Q in respect of Eq. (i) are as given below:

Q	0	4	8	12	16	20	24
H_m	65.0	65.3	66.1	67.5	69.5	72.0	75.0

Curves of H_{m_2} H_{m_2} and $\left(H_{m_1} + H_{m_2}\right)$ are plotted on a base of Q as shown in the Fig. Ex. 24.14 (a). On the same graph H_m v/s Q curve in respect of the above noted values is also plotted. It may be seen that the curves of H_m and $\left(H_{m_1} + H_{m_2}\right)$ intersect at a point corresponding to which $Q = 17.6$ litres/second; $H_{m_1} = 39.0$ m and $H_{m_2} = 31.5$ m.

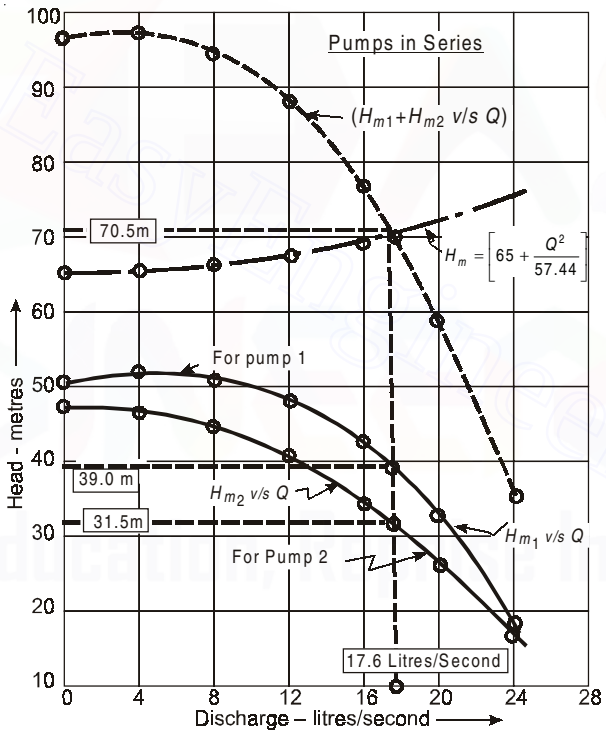


Figure Ex. 24.14 (a)

(b) Let Q_1 and Q_2 be the discharges in litres/second, flowing through the pumps 1 and 2 respectively which are connected in *parallel*. Then the total discharge flowing through the suction and delivery pipes $= (Q_1 + Q_2)$.

The total head required to be developed is

$$H_m = (h_s + h_d) + h_{f_s} + h_{f_d}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$H_m = 15 + \frac{0.02 \times 360 \times [(Q_1 + Q_2) \times 10^{-3}]^2}{[\pi(0.15)^2/4]^2 \times 2 \times 9.81 \times 0.15}$$

or $$H_m = 15 + \frac{(Q_1 + Q_2)^2}{127.64} \quad \text{...(ii)}$$

Curves of H_{m_1} and are H_{m_2} plotted on the base of Q as shown in Fig. 24.14 (b). From these curves for the same values of H_m, the values of Q_1, Q_2, and $(Q_1 + Q_2)$ are obtained which are as given below. Curve of $(Q_1 + Q_2)$ v/s H_m is also plotted on the same graph.

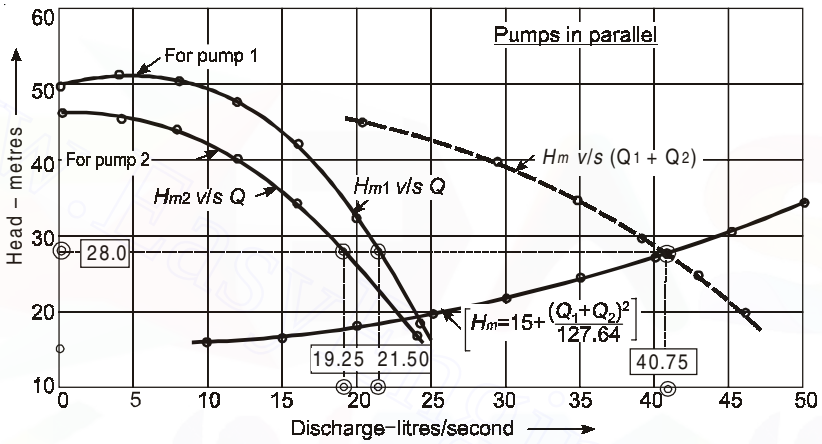


Figure Ex 24.14 (b)

H_m	20	25	30	35	40	45
Q_1	23.4	22.3	20.8	19.2	17.2	14.5
Q_2	22.7	20.5	18.2	15.6	12.2	6.0
$(Q_1 + Q_2)$	46.1	42.8	39.0	34.8	29.4	20.5

Similarly for different values of $(Q_1 + Q_2)$ the corresponding value of H_m in respect of Eq. (ii) are computed as given below; and these are also plotted on the same graph.

$(Q_1 + Q_2)$	10	15	20	25	30	35	40	45	50
H_m	15.8	16.8	18.1	19.9	22.1	24.6	27.5	30.9	34.6

It may be seen that the two $(Q_1 + Q_2)$ v/s H_m curves intersect at a point corresponding to which H_m = 28.0 and $(Q_1 + Q_2)$ = 40.75 litres/second. For H_m = 28 m, from the graph, Q_1 = 21.50 litres/second and Q_2 = 19.25 litres/second.

Example 24.15. *A three-stage centrifugal pump has impellers 375 mm diameter and 18 mm wide at outlet. The outlet vane angle is 45° and the vanes occupy 8% of the outlet area. The manometric efficiency is 84% and the overall efficiency is 75%. What head the pump will generate when running at 900 r.p.m., discharging 60 litres/second ? What is the input power?*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

$$V_{f_1} = \frac{60\times10^{-3}}{\pi\times0.375\times0.018\times0.92} = 3.08 \text{ m/s}$$

$$u_1 = [(\pi D_1 N)/60]$$

$$= \frac{\pi\times0.375\times900}{60} = 17.67 \text{ m/s}$$

Now from the outlet velocity triangle of Fig. 24.5,

$$V_{w1} = (u_1 - V_{f1} \cot 45°) = (17.67 - 3.08)$$

$$= 14.59 \text{ m/s}$$

$$\eta_{\text{mano}} = \frac{gH_m}{V_{w_1} u_1}; \text{ or } 0.84 = \frac{9.81\times H_m}{14.59\times17.66}$$

Thus $H_m = 22.06$ m

$\therefore$ Total head $H = 3H_m = 3(22.06) = 66.18$ m

$\eta_0 = (wQH)/P$

or $$P = \frac{9810\times60\times10^{-3}\times66.18}{0.75}$$

$$= 51938 \text{ W} = 51.938 \text{ kW}$$

Example 24.16. *A pump impeller is 375 mm diameter and it discharges water with component velocities of 2 and 12 m/s in the radial and tangential directions respectively. The impeller is surrounded by a concentric cylindrical chamber with parallel sides the outer diameter being 450 mm. If the flow in this chamber is a free spiral vortex, find the component velocities of water on leaving and increase in pressure if there is no loss.*

Solution

For free cylindrical vortex from Eq. 7.57

or $12 \times (187.5) = V_2 \times (225)$

or $$V_2 = \frac{12\times187.5}{225} = 10 \text{ m/s}$$

For radial flow from Eq. 7.65

$$V_1 R_1 = V_2 R_2$$

$$2 \times (187.5) = V_2 \times (225)$$

or $$V_2 = \frac{2\times187.5}{225} = 1.67 \text{ m/s}$$

Hence the component velocities of water on leaving are 1.67 m/s and 10 m/s in the radial and tangential directions respectively.

For spiral vortex motion the increase in pressure is given by Eq. 7.71 as

$$\frac{p_2}{w} - \frac{p_1}{w} = \left[\frac{(12)^2 - (10)^2}{2\times9.81} + \frac{(2)^2 - (1.67)^2}{2\times9.81}\right] = 2.304 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 24.17. *Pressures on the suction side and the discharge side of a pump are read by the pressure gage as*

(i) vacuum of 250 mm of mercury and (ii) 137.34 kN/m² (1.4 kg/cm²) respectively.

The gage on the suction side is placed 600 mm below the centreline of the pump where as the gage on the discharge side is placed 170 mm above the centreline. If the diameters of the pipes on suction and discharge sides are 150 mm and 100 mm respectively and the quantity of water to be pumped is 30 litres per second, calculate the power required to drive the pump assuming its efficiency as 60%.

Solution

$$\frac{p_s}{w} = -\frac{250}{1000} \times 13.6$$

$$= -3.4 \text{ m of water}$$

$$\frac{p_d}{w} = \frac{137.34 \times 10^3}{9810}$$

$$= 14 \text{ m of water}$$

$$V_s = \frac{30 \times 10^{-3}}{(\pi/4)(0.15)^2} = 1.698 \text{ m/s}$$

$$\therefore \quad \frac{V_s^2}{2g} = \frac{(1.698)^2}{2 \times 9.81} = 0.147 \text{ m}$$

$$V_d = \frac{30 \times 10^{-3}}{(\pi/4)(0.10)^2} = 3.820 \text{ m/s}$$

$$\therefore \quad \frac{V_d^2}{2g} = \frac{(3.820)^2}{2 \times 9.81} = 0.744 \text{ m}$$

The manometric head is given as

$$H_m = \left(\frac{p_d}{w} + \frac{V_{d^2}}{2g} + h\right) - \left(\frac{p_s}{w} + \frac{V_{s^2}}{2g} - h_1\right)$$

where h is the height of the pressure gage on the discharge side above the centreline of the pump; and h_1 is the depth of the vacuum gage on the suction side below the centreline of the pump.

$$h = (170 \times 10^{-3}) = 0.17 \text{ m; and } h_1 = (600 \times 10^{-3}) = 0.60 \text{ m}$$

Thus by substituting the values in the above equation, we get

$$H_m = (14 + 0.744 + 0.17) - (-3.4 + 0.147 - 0.60)$$

$$= 18.767 \text{ m}$$

The power required to drive the pump is given by

$$P = \frac{wQH_m}{\eta_0}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{9810 \times 30 \times 10^{-3} \times 18.767}{0.60}$$

$$= 9205 \text{ W} = 9.205 \text{ kW}$$

Example 24.18. *At what height from water surface a centrifugal pump may be installed in the following case to avoid cavitation: atmospheric pressure 101 kPa; vapour pressure 2.34 kPa; inlet and other losses in suction pipe 1.55 m; effective head of pump 52.5 m; and cavitation parameter* $\sigma = 0.118$.

Solution

Cavitation factor is given by

$$\sigma = \frac{[(p_a - p_v)/w] - (h_s + h_{f_s})}{H_m}$$

$$p_a = 101 \text{ kPa} = 101000 \text{ N/m}^2$$

$$p_v = 2.34 \text{ kPa} = 2340 \text{ N/m}^2$$

$$H_m = 52.5 \text{ m}; h_{fs} = 1.55 \text{ m}; \sigma = 0.118$$

Thus by substitution, we get

$$0.118 \times 52.5 = \frac{(101000 - 2340)}{9810} - h_s - 1.55$$

$$\therefore \quad h_s = 2.312 \text{ m}$$

i.e., the pump may be installed at a height of 2.312 m above the water surface in the supply tank.

Example 24.19. *A pump operates at a maximum efficiency of 82% and delivers 2.25* m^3/s *under a head of 18 m while running at 3600 r.p.m speed. Compute the specific speed of the pump. Also determine the discharge, head and power input to pump at a shaft speed of 2400 r.p.m. Cite the assumption made, if any.*

Solution

$$Q = 2.25 \text{ m}^3/\text{s}; H = 18 \text{ m}; \eta_0 = 82\% = 0.82;$$

$$N = 3600 \text{ r.p.m}$$

$$\text{Power } P = \frac{wQH}{\eta_0}$$

$$= \frac{9810 \times 2.25 \times 18}{0.82}$$

$$= 484518 \text{ W} = 484.518 \text{ kW}$$

The specific speed is given by

$$N_s = \frac{N\sqrt{Q}}{H^{3/4}}$$

$$Q = 2.25 \text{ m}^3/\text{s} = 2.25 \times 10^3 \text{ l/s}$$

$$\therefore \quad N_s = \frac{3600\sqrt{2.25 \times 10^3}}{(18)^{3/4}} = 19\,541 \text{ [r.p.m; l/s; m]}$$

However, if discharge Q is taken in m^3/s,

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$N_s = \frac{3600\sqrt{2.25}}{(18)^{3/4}} = 618 \text{ [r.p.m; m}^3\text{/s; m]}$$

Since $Q \propto N$; $H \propto N^2$; and $P \propto N^3$, we have

$$\frac{Q}{N} = \frac{Q_1}{N_1}$$

$$\therefore \quad Q_1 = \left(\frac{N_1}{N}\right) Q$$

$$= \frac{2400}{3600} \times 2.25 = 1.5 \text{ m}^3\text{/s.}$$

$$\frac{H}{N^2} = \frac{H_1}{N_1^2}$$

$$\therefore \quad H_1 = \left(\frac{N_1^2}{N^2}\right) H$$

$$= \frac{(2400)^2}{(3600)^2} \times 18 = 8 \text{ m}$$

$$\frac{P}{N^3} = \frac{P_1}{N_1^3}$$

$$\therefore \quad P_1 = \left(\frac{N_1^3}{N^3}\right) P$$

$$= \frac{(2400)^3}{(3600)^3} \times 484.518 = 143.56 \text{ kW}$$

The above expressions used for determining discharge, head and power at other speed are based on the assumption that the efficiency remains constant at all the speeds.

Example 24.20. *A centrifugal pump operates against a manometric head of 30 m with a manometric efficiency of 75%. The pressure rise through the impeller is 65% of the total head developed by the pump. The radial velocity of flow which is constant is 3 m/s. The outer diameter of the impeller is 400 mm and the width at outlet is 15 mm. The blades at inlet are curved backwards at 60° to the wheel tangent. Calculate (i) the discharge in litres per minute, (ii) speed, (iii) blade angle at outlet, and (iv) diameter of impeller at inlet.*

Solution

(i) The discharge Q delivered by the pump is given by

$$Q = k\pi B_1 D_1 V_{f_1}$$

Given $B_1 = 15 \text{ mm} = 0.015 \text{ m}$

$D_1 = 400 \text{ mm} = 0.400 \text{ m}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_{f_1} = 3 \text{ m/s}$$

Assuming $k = 1$, we get

$$Q = 1 \times \pi \times 0.015 \times 0.400 \times 3$$

or $$Q = 0.0565 \text{ m}^3/\text{s} = 3390 \text{ litres/minute}$$

(ii) Manometric efficiency is given by

$$\eta_{\text{mano}} = \frac{gH_m}{V_{w_1} u_1}$$

or $$\frac{V_{w_1} u_1}{g} = \frac{H_m}{\eta_{\text{mano}}}$$

$$H_m = 30 \text{ m}; \quad \eta_{\text{mano}} = 75\% = 0.75$$

$$\therefore \quad \frac{V_{w_1} u_1}{g} = \frac{30}{0.75} = 40 \text{ m} \qquad \text{...(i)}$$

i.e., total head developed by the pump = 40 m

The pressure rise through the impeller $\left(\frac{p_1 - p}{w}\right)$ is 65% of the total head developed by the pump.

Thus

$$\left(\frac{p_1 - p}{w}\right) = (0.65 \times 40) = 26 \text{ m}$$

Applying Bernoulli's equation between the inlet and outlet tips of the impeller and neglecting the head loss in the impeller, we have

$$\frac{p}{w} + \frac{V^2}{2g} = \frac{p_1}{w} + \frac{V_1^2}{2g} - \frac{V_{w_1} u_1}{g}$$

or $$\left(\frac{p_1 - p}{w}\right) = \frac{V^2}{2g} + \frac{V_{w_1} u_1}{g} - \frac{V_1^2}{2g}$$

$$V = V_f = 3 \text{ m/s}$$

Thus by substitution, we get

$$26 = \frac{(3)^2}{2 \times 9.81} + 40 - \frac{V_1^2}{2g}$$

or $$V_1 = 16.84 \text{ m/s}$$

But $$V_1 = \sqrt{V_{w_1}^2 + V_{f_1}^2}$$

$$\therefore \quad V_{w1} = \sqrt{V_1^2 - V_{f_1}^2}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V_{f_1} = 3 \text{ m/s}$$

Thus by substitution, we get

$$V_{w1} = \sqrt{(16.84)^2 - (3)^2} = 16.57 \text{ m/s}$$

By substituting in Eq. (i), we get

$$\frac{16.57 \times u_1}{9.81} = 40$$

or $$u_1 = \frac{40 \times 9.81}{16.57} = 23.68 \text{ m/s}$$

But $$u_1 = \frac{\pi D_1 N}{60}$$

or $$23.68 = \frac{\pi \times 0.400 \times N}{60}$$

or $$N = \frac{23.68 \times 60}{\pi \times 0.400} = 1131 \text{ r.p.m.}$$

i.e., speed of the pump = 1131 r.p.m.

(iii) From outlet velocity triangle, we have

$$\tan \phi = \frac{V_{f_1}}{u_1 - V_{w_1}}$$

or $$\tan \phi = \frac{3}{23.68 - 16.57} = 0.4219$$

$\therefore$ $$\phi = 22°52'$$

i.e., blade angle at outlet = 22°52′

(iv) From inlet velocity triangle

$$\tan \phi = \frac{V_f}{u}$$

or $$u = V_f \cot \theta$$

$$\theta = 60°$$

$\therefore$ $$u = 3 \cot 60° = 1.732 \text{ m/s}$$

Also $$u = \frac{\pi D N}{60}$$

or $$1.732 = \frac{\pi \times D \times 1131}{60}$$

or $$D = \frac{1.732 \times 60}{\pi \times 1131}$$

or $$D = 0.029 \text{ m} = 29 \text{ mm}$$

i.e., the diameter of the impeller at inlet = 29 mm

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 24.21. *During an experiment a centrifugal pump operating at 1150 r.p.m. developed a head of 25 m and a discharge of 3800 litres/minute at the best efficiency regime. At this regime the power drawn by the pump is 28 h.p. The mechanical losses due to bearings and packing is 2% of the pump output and the disc friction loss is 1.6 h.p. The leakage is estimated to be 160 litres/minute.*

The impeller is of 38 cm diameter and is 2 cm wide at the exit. The blade angle at outlet is 22°. There are seven blades and their thickness is constant throughout and equal to 5 mm. Assuming that the fluid approaches the impeller with prerotation, determine the overall efficiency, head slip and actual absolute flow angle at impeller exit.

Solution

The overall efficiency of the pump is given by Eq. 24.23 as

$$\eta_0 = \frac{wQH_m}{\text{Power given to the shaft}}$$

$$w = 9810 \text{ N/m}^3;\ Q = \frac{(3800-160)}{1000\times 60} = 0.061 \text{ m}^3/\text{s};$$

$H_m = 25$ m; and Power given to the shaft = 28×746 W

Thus by substitution, we get

$$\eta_0 = \frac{9810\times 0.061\times 25}{28\times 746} = 0.716 \text{ or } 71.6\%$$

$$\text{Pump output} = wQH_m$$
$$= 9810 \times 0.061 \times 25 \text{ W}$$
$$= 14960.25 \text{ W}$$

$\therefore$ Mechanical losses due to bearings and packing

$$14960.25\times\frac{2}{100} = 299.205 \text{ W}$$

$$\text{Disc friction loss} = 1.6 \text{ h.p.}$$
$$= 1.6 \times 746 \text{ W} = 1193.6 \text{ W}$$

$\therefore$ Total mechanical losses

$$= (1193.6 + 299.205) \text{ W}$$
$$= 1492.805 \text{ W}$$

Thus the power imparted by the impeller

$$= [(28\times 746) - 1492.805] \text{ W}$$
$$= 19395.195 \text{ W}$$

Again power imparted by the impeller

$$= w(Q+\Delta Q)\times\frac{V_{w_1}u_1}{g}$$

$$= 9810\times\frac{3800}{60\times 1000}\times\frac{V_{w_1}u_1}{g}$$

$$= 63.33\ V_{w_1}u_1$$

$\therefore$ $63.33\ V_{w_1}u_1 = 19395.195$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$V_{w_1} u_1 = \frac{19395.195}{63.33} = 306.26 \quad \ldots(i)$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$D_1 = 0.38$ m; and $N = 1150$ r.p.m

Thus $$u_1 = \frac{\pi \times 0.38 \times 1150}{60} = 22.88 \text{ m/s}$$

And $$V_{w_1} = \frac{306.26}{22.88} = 13.39 \text{ m/s}$$

The discharge delivered by the pump is given by

$$Q = \pi(D_1 - zt)B_1 V_{f_1}$$

$z = 7$; $t = 0.005$ m; and $B_1 = 0.02$ m

Thus $$\frac{3800}{1000 \times 60} = \pi[0.38 - (0.005 \times 7)] \times 0.02 \times V_{f_1}$$

$$\therefore \quad V_{f_1} = \frac{3800}{1000 \times 60 \times \pi \times 0.345 \times 0.02} = 2.92 \text{ m/s}$$

From outlet velocity triangle, we have

$$\text{tam}\,\beta = \frac{V_{f_1}}{V_{w_1}}$$

$$= \frac{2.92}{13.39} = 0.2181$$

$$\therefore \quad \beta = 12°18'$$

Thus actual absolute flow angle at impeller exit is 12°18′

Again from outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{u_1 - V_{w_1}}$$

$$= \frac{2.92}{(22.88 - 13.39)} = 0.3077$$

$$\therefore \quad \phi = 17°6'$$

However, the actual blade angle provided at outlet is 22°, corresponding to which if the value of V_{w_1} is V'_{w_1} it may be obtained as

$$\tan 22° = \frac{2.92}{22.88 - V'_{w_1}}$$

or $$V'_{w_1} = 15.65 \text{ m/s}$$

The head corresponding to the value of V'_{w_1}

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{V'_{w_1} u_1}{g}$$

$$= \frac{15.65 \times 22.88}{9.81} = 36.50 \text{ m}$$

But the actual head developed is

$$\frac{V_{w_1} u_1}{g} = \frac{306.26}{9.81} = 31.22 \text{ m}$$

$\therefore$ Head slip $= (36.50 - 31.22)$

$= 5.28$ m

Example 24.22. *Two identical pumps are connected to a pipe system as shown in Fig. Ex. 24.22 to pump water into the high level reservoir. The head discharge characteristic of the pumps is given by the equation*

$$H = 40 + 0.14Q - 0.0012Q^2$$

where Q is the pump discharge in litres per second and H is the head developed by the pump in m.

The pipe AB is 100 m long and is of diameter 30 cm. The pipe BC is 45 cm in diameter and 200 m long. For both pipes the friction factor can be taken as 0.03. 'V' is a regulating valve on the pipe line BC. The static lifts against which the pumps are working are 20 m and 30 m.

For a given position of the valve 'V' the pump operating against a static lift of 30 m develops a discharge of 140 litres per second. Determine the discharge of the second pump and the loss coefficient of the valve, neglecting all other local losses.

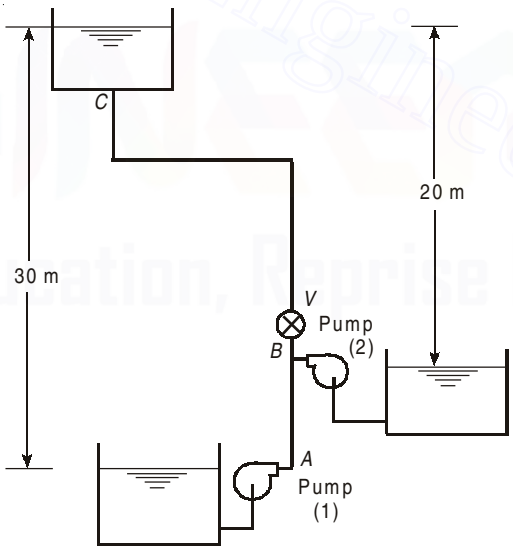


Figure Ex. 24.22

Solution

Discharge of pump No. 1 =140 l/s

$\therefore$ Head developed by pump No. 1 is

$$H_1 = 40 + 0.14 \times 140 - 0.0012 \times (140)^2 = 36.08 \text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When only Pump No.1 is operating, then

$$V_{AB} = \frac{140}{1000 \times (\pi/4) \times (0.30)^2} = 1.98 \text{ m/s}$$

$$V_{BC} = \frac{140}{1000 \times (\pi/4) \times (0.45)^2} = 0.88 \text{ m/s}$$

$$h_{f_{AB}} = \frac{0.03 \times 100 \times (1.98)^2}{2 \times 9.81 \times 0.30} = 1.998 \text{ m}$$

$$h_{f_{BC}} = \frac{0.03 \times 200 \times (0.88)^2}{2 \times 9.81 \times 0.45} = 0.526 \text{ m}$$

Let K be the loss coefficient of the valve, then head loss in the valve $= K \dfrac{V_{BC}^2}{2g}$

Thus we have

$$H_1 = H_{s_1} + h_{f_{AB}} + K\frac{V_{BC}^2}{2g} + h_{f_{BC}} + \frac{V_{BC}^2}{2g}$$

or $$36.08 = 30 + 1.998 + K\frac{(0.88)^2}{2 \times 9.81} + 0.526 + \frac{(0.88)^2}{2 \times 9.81}$$

$$\therefore \quad K = 89.11 \approx 89$$

Let the discharge of Pump No. 2 be Q l/s.

Then head developed by Pump No. 2 is

$$H_2 = 40 + 0.14\,Q - 0.0012\,Q^2$$

When both the pumps are operating the sum of the heads developed by the two pumps is

$$H_1 + H_2 = [36.08 + 40 + 0.14\,Q - 0.0012\,Q^2]$$

Discharge through pipe $AB = 140$ l/s

$$\therefore \quad V_{AB} = \frac{140}{1000 \times (\pi/4) \times (0.30)^2} = 1.98 \text{ m/s}$$

$$h_{f_{AB}} = \frac{0.03 \times 100 \times (1.98)^2}{2 \times 9.81 \times 0.30} = 1.998 \text{ m}$$

Discharge through pipe $BC = (Q + 140)$ l/s

$$\therefore \quad V_{BC} = \frac{(Q+140)}{1000 \times (\pi/4) \times (0.45)^2} = \frac{(Q+140)}{159.04} \text{ m/s}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$h_{f_{BC}} = \frac{0.03\times 200\times (Q+140)^2}{2\times 9.81\times 0.45\times (159.04)^2} = \frac{(Q+140)^2}{37219.71}\text{ m}$$

Head loss in the valve

$$= K\,\frac{V_{BC}^2}{2g}$$

$$= \frac{89\times (Q+140)^2}{2\times 9.81\times (159.04)^2} = \frac{(Q+140)^2}{5575.99}\text{ m}$$

Thus we have

$$H_1 + H_2 = H_{s_1} + H_{s_2} + h_{f_{AB}} + K\frac{V_{BC}^2}{2g} + h_{f_{BC}} + \frac{V_{BC}^2}{2g}$$

or $$[36.08 + 40 + 0.14Q\ \ 0.0012\,Q^2] = 30 + 20 + 1.998 + \frac{(Q+140)^2}{5575.99} + \frac{(Q+140)^2}{37219.71} + \frac{(Q+140)^2}{2\times 9.81\times (159.04)^2}$$

or $$Q^2 - 58.36\,Q - 14286 = 0$$

$$\therefore \quad Q = 152\text{ l/s}$$

Discharge of Pump No. 2 is 152 l/s and loss coefficient of the valve is 89.

Example 24.23. *In a pumping station, 18000 cum water is to be raised per day from an intake well to a sedimentation tank under a static head of 21 m. Lengths of suction pipe and rising main are 40 m and 150 m respectively. Diameters of pipes is 50 cm. There are two shifts of working of pumps each of 8 hours. Take coefficient of friction as 0.01 and combined efficiency of motor and pump as 80%. Recommend the number of units of pumps each having B.H.P of 30.*

Solution

Total quantity of water to be raised per day= 18,000 m^3.

Since water is to by raised by pumps working in two shifts each of 8 hours, the entire quantity of water is to be raised in (2×8) = 16 hours.

$$\therefore \text{ Discharge } \quad Q = \frac{18,000}{16\times 3600} = 0.3125\text{ m}^{3/5}$$

Diameters of suction pipe and rising main are

$$d_s = d_d = 50\text{ cm} = 0.5\text{ m}$$

$$\therefore \quad \text{Area } A = \frac{\pi}{4}\times (0.5)^2 = 0.19634\text{ m}^2$$

$$\text{Velocity } V = \frac{0.3125}{0.19634} = 1.5916\text{ m/s}$$

$$\text{Static head } H_s = 21\text{ m}$$

$$\text{Total length of pipe} = (40 + 150) = 190\text{ m}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Coefficient of friction $f = 0.01$

Head loss due to friction

$$h_f = \frac{f\,LV^2}{2gd}$$

Thus by substitution, we get

$$h_f = \frac{0.01 \times 190 \times (1.5916)^2}{2 \times 9.81 \times 0.50}$$

$$= 0.4906 \text{ m.}$$

$\therefore$ Total head $H = H_s + h_f$

$= (21.0 + 0.4906) = 21.4906$ m

$$\text{B.H.P} = \frac{WQH}{75}$$

$$= \frac{1000 \times 0.3125 \times 21.4906}{75}$$

$= 89.54$ h.p

Efficiency of motor and pump $= 80\% = 0.80$

$\therefore$ Actual B.H.P required

$$= \frac{89.54}{0.80} = 111.93 \text{ h.p.}$$

B.H.P. of each pump $= 30$ h.p.

$\therefore$ Number of pumps required

$$= \frac{111.93}{30} = 3.734 \simeq 4$$

SUMMARY OF MAIN POINTS

1. Centrifugal pumps are classified as rotodynamic type of pumps in which dynamic pressure is developed which enables the lifting of liquids from a lower level to a higher level.
2. A centrifugal pump consists of a wheel or rotor called impeller which is provided with a series of backward curved blades or vanes, an airtight casing which surrounds the impeller, a suction pipe fitted with a foot valve and strainer at its lower end and a delivery pipe provided with a delivery valve.
3. The head imparted by the impeller to the liquid is given as

$$= \frac{1}{g}\left(V_{w_1} u_1\right)$$

where

V_{w_1} = velocity of whirl at outlet; and

u_1 = tangential velocity of impeller at outlet.

4. The head of a centrifugal pump may be expressed as

(i) Static head

(ii) Manometric head

(i) The static head H_s is the vertical distance bewtween the liquid surfaces in the sump from which the liquid is being raised and the tank to which the liquid is delivered by the pump. Thus

$$H_s = h_s + h_d$$

in which h_s is the vertical height of the centre line

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the pump shaft above the liquid surface in the sump, and h_d is the vertical height of the liquid surface in the tank to which the liquid is delivered above the centre line of the pump shaft.

The term h_s is known as static suction lift, and h_d is known as static delivery lift.

(ii) The manometric head H_m is the total head that must be produced by the pump to satisfy the external requirements. Thus

$$H_m = \frac{V_{w_1} u_1}{g} - (\text{losses of head in the pump})$$

5. The manometric head is given by the following expressions.

(i) $$H_m = \left(\frac{p_d}{w} + \frac{V_d^2}{2g} + h\right) - \left(\frac{p_s}{w} + \frac{V_s^2}{2g}\right)$$

(ii) $$H_m = h_s + h_d + h_{f_s} + h_{f_d} + \frac{V_d^2}{2g}$$

where

p_d = pressure at the exit from the pump;
p_s = pressure at inlet to the pump;
V_d = velocity of flow in the delivery pipe;
V_s = velocity of flow in the suction pipe;
h = height of pressure gauge at the exit of the pump above the pressure gauge at the inlet to the pump;
h_{f_s} = head loss in the suction pipe; and
h_{f_d} = head loss in the delivery pipe.

6. The efficiencies of a centrifugal pump are

(i) Manometric efficiency

$$\eta_{\text{mano}} = \frac{gH_m}{V_{w_1} u_1}$$

or $$\eta_{\text{mano}} = \frac{H_m}{H_m + \text{losses in the pump}}$$

(ii) Volumetric efficiency

$$\eta_v = \frac{Q}{Q + \Delta Q}$$

(iii) Mechanical efficiency

$$\eta_{\text{mech}} = \frac{\left(V_{w_1} u_1 / g\right)}{\left(V_{w_1} u_1 / g\right) + \left[\begin{matrix}\text{mechanical head} \\ \text{losses in bearing}\end{matrix}\right]}$$

(iv) Overall efficiency

$$\eta_o = \frac{WQH_m}{\text{Power given to the shaft}}$$

$$\eta_o = (\eta_{\text{mano}}) \times (\eta_v) \times (\eta_{\text{mech}})$$

7. The minimum speed for starting a centrifugal pump is given by

$$N = \frac{60\sqrt{2gH_m}}{\pi\sqrt{D_1^2 - D^2}}$$

8. The specific speed N_s of a centrifugal pump may be defined as the speed in r. p.m. of a geometrically similar pump of such a size that under corresponding conditions it would deliver 1 litre of liquid per second against a head of 1 metre. The specific speed is given by

$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}$$

9. For complete similarity to exis between the model and the prototype pumps the necessary conditions are as follows:

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$$

$$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p$$

$$\left(\frac{P}{\rho gQH}\right)_m = \left(\frac{P}{\rho gQH}\right)_p$$

$$\left[\frac{NQ}{(gH)^{3/4}}\right]_m = \left[\frac{NQ}{(gH)^{3/4}}\right]_p$$

10. The efficiency of the prototype pump and that of the model pump are related as

$$\frac{0.95 - \eta_m}{0.95 - \eta_p} = \left[\frac{0.658 + \log_{10} Q_p}{0.658 + \log_{10} Q_m}\right]^2$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

11. To produce a high head the pumps are connected in *series*. The higher heads may also be produced by using multi-stage pumps. A *multistage pump* consists of two or more indentical impellers mounted on the same shaft and enclosed in the same casing.
12. The pumps are connected in *parallel* when a large quantity of liquid is required to be pumped against a relatively small head.
13. In order to predict the behaviour and performance of pump under varying conditions the following three types of characteristics curves are prepared for the centrifugal pumps.
    (i) Main and operating characteristics.
    (ii) Constant efficiency or Muschel curves
    (iii) Constant head and constant discharge curves
14. In centrifugal pumps cavitation may occur at the suction side of the pump if the pressure drops below the vapour pressure of the liquid.
    As in the case of turbines for pumps also Thoma's cavitation factor σ is used to indicate whether cavitation will occur. In order to avoid the cavitation the value of σ should not be less than the critical value σ_c given by
    $$\sigma_c = 0.103\,(N_s/1000)^{4/3}$$
    where
    N_s = specific speed of the pump.
15. Before starting a centrifugal pump it should be primed. Priming is the operation in which the suction pipe, casing of the pump and the portion of the delivery pipe upto the delivery valve are completely filled with the liquid which is to be pumped, so that all the air is driven out from this portion of the pump.

PROBLEMS

24.1 Explain with neat sketches the working of a single-stage centrifugal pump.

24.2 Under what headings the centrifugal pumps are classified?

24.3 State the difference between a closed, semi-closed and open impeller.

24.4 Discuss the various methods adopted to increase the efficiency of a centrifugal pump by altering the shape of the casing of chamber surrounding the impeller.

24.5 Why is the efficiency of a volute casing of a centrifugal pump as an energy conversion device low? How is a whirlpool or vortex chamber superior in performance?

24.6 If a centrifugal pump does not deliver any water when started, what may be the probable causes and how can they be remedied?

24.7 Define static and manometric head of a centrifugal pump. State the different types of head losses which may occur in a pump installation.

24.8 What are the different efficiencies of a centrifugal pump?

24.9 What is meant by 'Priming' of a pump? What are the different priming arrangements employed for small and big pumping units ?

24.10 Why can the suction lift of a pump not exceed a certain limit?

24.11 A centrifugal pump of the radial type delivers 5 000 litres per minute against a total head of 38 m, when running at a speed of 1450 r.p.m. If the outer diameter of the impeller is 300 mm and its width at the outer periphery is 13 mm, find the vane angle at exit. Assume manometric efficiency as 80%. [*Ans.* 71°21′]

24.12 The impeller of a centrifugal pump has 1.2 m outside diameter. It is used to lift 1800 litres of water per second against a head of 6 m. Its vanes make an angle of 150 with the direction of motion at outlet and runs at 200 r.p.m. If the radial velocity of flow at outlet is 2.5 m/s, find the manometric efficiency. Also find the lowest speed to start the pump, if the diameter of the impeller at inlet is equal to half the diameter at exit. [*Ans.* 57%; 199.4 r.p.m.]

24.13 A centrifugal pump is required to deliver 280 litres of water per second against a head of 16 m. If the vanes of the impeller are radial at outlet and the velocity of flow is constant equal to 2 m/s, find the proportions of the pump. Assume η_{mano} = 80% and the ratio of breadth to diameter at outlet as 0.1.
[*Ans.* $\beta = 8.13°$; $\alpha = 15.94°$; $D_1 = 0.67$ m; $B_1 = 0.067$ m]

24.14 A centrifugal pump is required to discharge 600 litres of water per second against a head of 12 m, when running at a speed of 750 r.p.m.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The manometric efficiency is to be 80%, the loss of head in the pump being assumed to be $0.025\,V_1^2$ of water, where V_1 is the absolute velocity of water leaving the impeller. Water enters the impeller without whirl and the velocity of flow is 3 m/s. Determine (a) the impeller diameter and the outlet area; and (b) the vane angle at the outlet edge of the impeller.

[*Ans.* (a) 0.356 m, 0.2 m^2 ; (b) 41°15′]

24.15 A centrifugal pump running at 1 450 r.p.m. has the characteristics as given below:

Discharge (Liters/sec.)	*Heat (meters)*	*Efficiency (%)*
11.3	25.8	65
16.9	25.0	70
22.6	24.1	73
28.3	23.2	74
34.0	21.4	72
39.6	18.9	69
45.2	15.8	62

Draw the operating characteristics of the pump and determine its specific speed.

The pump lifts water against a static head of 12 m through a long pipeline in which the loss of head in metres, due to friction is given by the expression, $h_f = 0.012\,Q^2$, where Q is the discharge in litres/sec. The minor losses in the pipe may be neglected. Determine the power required to drive the pump.

[*Ans.* 730 r.p.m.; 8.81 kW]

24.16 The axis of a centrifugal pump is 2.5 m above water level in the sump and the static lift from the pump centre is 33.5 m. The friction losses in the suction and the delivery pipes are 1 m and 7.5 m respectively. The suction and delivery pipes are each 75 mm diameter. The impeller is 300 mm diameter and 18 mm wide at outlet and its speed is 1 700 r.p.m. The water at inlet has radial flow and the vane angle at outlet is 32° to the tangent to the periphery. Compute the discharge of the pump and the power required to drive the pump. Assume $\eta_{mano} = 77\%$ and $\eta_0 = 72\%$. [*Ans.* 0.038 m^3/s; 25.3 kW]

24.17 A four-stage centrifugal pump has impellers 380 mm diameter and 19 mm wide at outlet. The outlet vane angle is 45° and the vanes occupy 8% of the outlet area. The manometric efficiency is 84% and the overall efficiency 72%. Determine the head generated by the pump when running at 900 r.p.m., and discharging 59 litres/second. Also determine the power required to drive the pump. [*Ans.* 92.48 m; 74.34 kW]

24.18 A two-stage centrifugal pump is required for a fire engine for a duty of 3 660 litres per minute at a head of 75 m. If the overall efficiency of the pump is 75% and the specific speed per stage is about 1 300, find (a) the running speed in r.p.m. and (b) the power of the driving engine. If the actual manometric head developed is 65% of the theoretical head, there is no slip, the outlet vane angle is 30°, and the radial flow velocity at exit is 0.15 times the tip speed at exit, find the diameter of the impellers.

[*Ans.* (a) 2 522 r.p.m; (b) 59.84 kW, 0.21 m]

24.19 A multi-stage centrifugal pump is to be designed to deliver 700 litres of water per second against a manometric head of 60 m. There are to be four equal impellers keyed to the same shaft, which has speed of 350 r.p.m. The vanes are curved back so that the direction of the relative velocity at exit makes an angle of 120° with the direction of the corresponding peripheral velocity and the impeller is surrounded by guides. Assume that the water enters radially and the velocity of flow through the impeller is 0.27 of the peripheral velocity and that the losses in the pump amount to one-third of the velocity head at discharge from the impeller. Find: (a) the outer dimensions of the impeller; (b) the manometric efficiency; (c) the angle of the guides.

[*Ans.* (a) D_1 = 0.78 m, B_1 = 74 mm; (b) 84.5%; (c) 17°44′]

24.20 Two centrifugal pumps operating in parallel are employed for lifting water against a static head of 18 m, the delivery pipe being 0.5 m in diameter and 15 000 m long having $f = 0.032$. Determine the head at the pipe inlet and the discharge; given the following data for the two pumps:

Q Litres / second	0	15	30	45	60
PUMP 1 H_{m_1} *meters*	45.0	43.5	40.0	32.7	23.3
PUMPS 2 H_{m_2} *metres*	40.7	39.0	33.7	27.3	20.7

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

24.21 Explain (i) double suction impeller, (ii) shut-off head, (iii) manometric efficiency, (iv) net positive suction head as applied to a suction pump.

24.22 Assuming that the radial component of flow of fluid through a centrifugal pump remains constant and that the fluid enters radially, prove that the ratio of pressure head H_p to velocity head H_v created by an impeller neglecting the losses is given by an equation

$$\frac{H_p}{H_v} = \left(\frac{u_1 + V_{f_1} \cot\beta}{u_1 - V_{f_1} \cot\beta}\right)$$

where u_1 is rim speed of impeller at outlet, V_{f_1} is velocity of flow and β is the exit angle of impeller blades. Hence show that ideal efficiency of the impeller is given by

$$\eta_0 = \frac{u_1 + V_{f_1} \cot\beta}{2u_1}$$

24.23 Show that the pressure rise in the impeller of a centrifugal pump, when friction and other losses in the impeller are neglected, is given by

$$(p_2 - p_1)/w = \left(u_1^2 - V_{f_1}^2\right)/2g$$

provided flow is radial at inlet, velocity of flow is constant throughout and outlet angle of impeller vane = 45°. u_1 and V_{f_1} are peripheral and flow velocities at outlet end of impeller

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Miscellaneous Hydraulic Machines

Chapter **25**

25.1 INTRODUCTION

There are several hydraulic machines which are employed for either storing the hydraulic energy and then transmitting it when required or magnifying the hydraulic energy (mostly the pressure energy) several times and then transmitting the same. In general these machines are based on the principles of hydrostatics and hydrokinetics. In these machines the hydraulic energy is transmitted through the liquid medium for which water or oil is used. Some of these machines are described below.

25.2 HYDRAULIC ACCUMULATOR—SIMPLE AND DIFFERENTIAL TYPES

The hydraulic accumulator is a device which is used for temporarily storing or accumulating the liquid under pressure supplied by the pump when it is not required by the machines. Several hydraulic machines such as lifts, cranes, presses etc., are required to do a large amount of work during a small interval of time which is followed by an idle period. The demand of the liquid under pressure (or hydraulic energy) by these machines will however not be uniform throughout the period of their operation. For example, a hydraulic crane or lift requires the liquid under pressure to be supplied during the upward motion of the load only and practically no energy is used during the downward motion. But the pump normally supplies the liquid under pressure at a more or less uniform rate throughout its supply period. As such by introducing an accumulator in between the pump and the machines (i.e., lift, crane, press, etc.) the liquid under pressure can be stored in the accumulator during the idle period of the machine (i.e., when the energy is not required by the machine), and this stored liquid will be supplied to the machine along with the uniform supply from the pump when during the working stroke of the machine it is in need of large quantity of liquid pressure. Thus by such an arrangement the

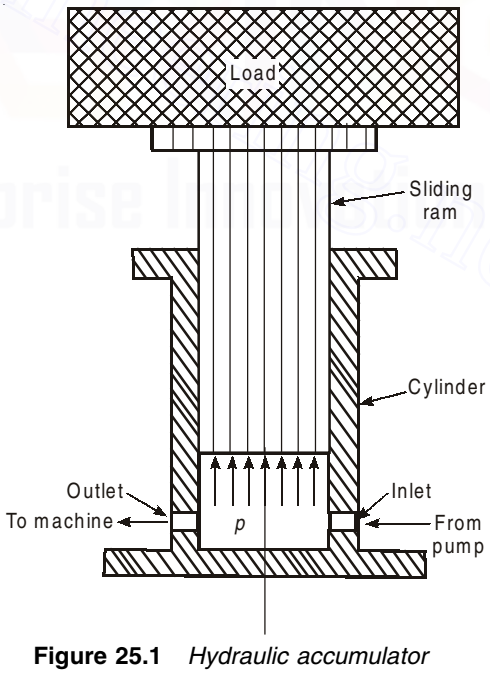


Figure 25.1 *Hydraulic accumulator*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

capacity of the pump need not be as large as that required by the machine when it is doing maximum work, since the machine will then receive a part of the required supply from the accumulator.

As shown in Fig. 25.1 the hydraulic accumulator consists of a fixed vertical cylinder containing a sliding plunger or ram. The ram is loaded with weights. One side of the cylinder is connected to the pump and the other side to the machine. In the beginning the ram is at its lowermost position. Now when the liquid is not required by the machine, then the pump will deliver the liquid under pressure to the cylinder, which will raise the loaded ram till the cylinder is full. This constitutes the upward stroke of the ram, during which the liquid under pressure (or hydraulic energy) is stored in the accumulator. Later when the machine requires liquid it will draw the same from the accumulator, and the ram will gradually move down. This is the downward stroke of the ram, during which the liquid under pressure (or the hydraulic energy) is delivered to the machine.

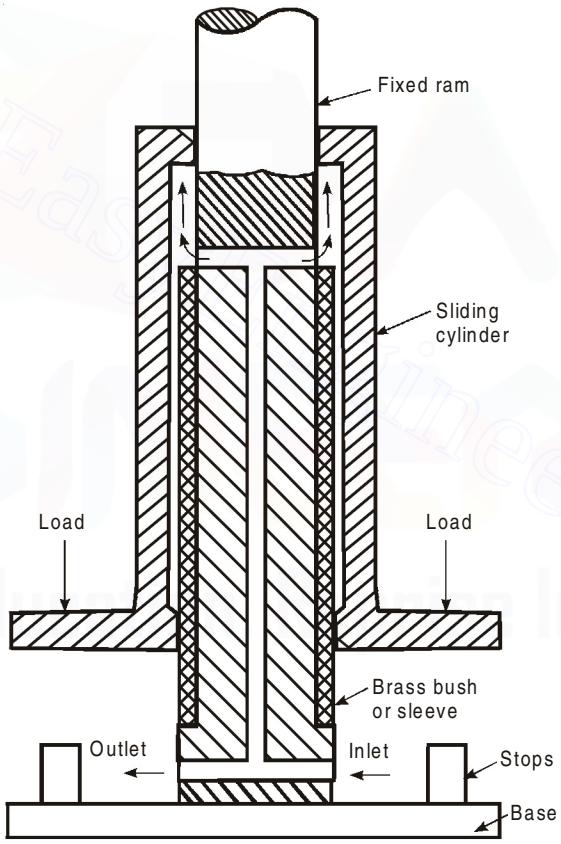


Figure 25.2 *Differential accumulator*

The maximum amount of energy that the accumulator can store is known as the *capacity* of the accumulator.

Let D be the diameter of the sliding ram, L be the stroke or lift of the ram, p be the pressure intensity of the liquid supplied by the pump and W be the total weight of the ram (including the weight of the load on the ram), then

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$W = \frac{\pi}{4} D^2 \times p \qquad ...(25.1)$$

Also the work done in lifting the ram or the capacity of the accumulator

$$= WL = \left(\frac{\pi}{4}\right) D^2 \times p \times L \qquad ...(25.2)$$

But volume of the accumulator

$$= \left(\frac{\pi}{4}\right) D^2 L$$

∴ Capacity of accumulator

$$= (p \times \text{volume}) \qquad ...(25.3)$$

Another form of accumulator, known as *Tweddell's differential accumulator* is as shown in Fig. 25.2. The advantage of this accumulator is that liquid can be stored at a high pressure by a comparatively small load on the ram. It consists of a fixed ram of which the lower portion is made larger than the upper portion by surrounding it with a closely fitting brass bush or sleeve. The fixed ram is surrounded by a sliding cylinder having a circular collar projecting outwards at the base, on which the required weights may be placed in order to load the cylinder. The fixed ram is provided with central vertical hole throughout its length, through which the liquid supplied from the pump enters the cylinder. This causes the loaded cylinder to move upwards, thus storing the hydraulic energy. When the machine draws liquid from the accumulator, the liquid leaves the cylinder through the same central hole. The liquid entering the cylinder exerts the pressure on the internal annular area of the cylinder which is equal to the horizontal cross-sectional area of the brass bush or sleeve.

Now if a is the cross-section area of the brass bush and p is the intensity of pressure of the liquid supplied by the pump, then

Load on cylinder = $(p \times a)$

Therefore by making the area of the bush small, it is possible to store liquid at a high pressure with a small load.

Again if L is the vertical lift of the cylinder, then

Capacity of accumulator

$$= paL = p \times \text{volume} \qquad ...(25.4)$$

25.3 HYDRAULIC INTENSIFIER

The hydraulic intensifier is a device which is used for increasing the intensity of pressure of the liquid, by utilizing the energy of a larger quantity of liquid at low pressure. Often hydraulic machines such as press etc., require liquid at high pressure which may not be directly available from a pump. It can however be provided by introducing an intensifier between the pump and the machine.

As shown in Fig. 25.3 an intensifier consists of a fixed ram surrounded by a sliding cylinder or ram which contains high pressure liquid, which is supplied to the machine through the fixed ram. The sliding cylinder is encased by a fixed cylinder which contains the low pressure liquid from the main supply.

As shown in Fig. 25.3 valves A and D allow low pressure liquid from the supply, valve C is for exhaust and valve B allows high pressure liquid to be supplied to the machine. In the beginning when the sliding cylinder is at its bottom-most position, the fixed cylinder is full of low pressure liquid.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The valves *B* and *D* are then closed, the valve *A* is opened thus admitting the low pressure liquid into the sliding cylinder, and the valve *C* is also opened which permits the low pressure liquid from the fixed cylinder to be discharged to exhaust and the sliding cylinder to move upwards. When the sliding cylinder reaches its topmost position, the inside of the sliding cylinder is full of low pressure liquid. Now the valves *A* and *C* are closed and the valves *D* and *B* are opened. The low pressure liquid from the supply then enters the fixed cylinder, which forces the sliding cylinder to move downwards, thereby producing high pressure liquid in the sliding cylinder, which is supplied to the machine. The same cycle of operation is repeated. The intensifier described here is thus single-acting since it supplies high pressure liquid during the downward stroke only. However, double acting intensifiers are also made which give a continuous supply of high pressure liquid. It is possible to raise the pressure intensity of liquid to as high as 157 MN/m^2 [1600 kg (f)/cm^2], by means of an intensifier.

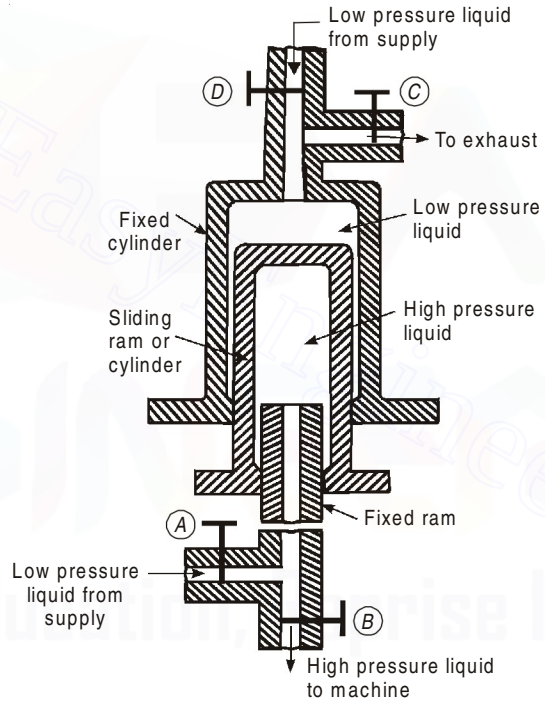


Figure 25.3 *Hydraulic intensifier*

The magnitude of the pressure intensity developed by an intensifier and the quantity of high pressure liquid supplied by it may be computed as follows:

Let D_1 be the external diameter of the sliding cylinder and D_2 be the diameter of the fixed ram; and A_1 and A_2 be their respective cross-sectional areas. Also let p_1 be the intensity of pressure of low pressure liquid in the fixed cylinder and p_2 be the intensity of pressure of high pressure liquid inside the sliding cylinder. Then as the total upward force equals total downward force,

$$p_1 A_1 = p_2 A_2$$

or
$$p_2 = p_1 (A_1/A_2) = p_1 (D_1/D_2)^2 \quad ...(25.5)$$

Further if *L* is the stroke length or lift of the sliding cylinder then in one stroke of the sliding cylinder,

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the volume of low pressure liquid entering the fixed cylinder equals (A_1L) and the volume of the high pressure liquid supplied to the machine from the sliding cylinder equals (A_2L). Thus if Q_1 is the rate of discharge of low pressure liquid entering the fixed cylinder and Q_2 is the rate of discharge of high pressure liquid supplied to machine, then

$$\frac{Q_1}{Q_2} = \frac{A_1L}{A_2L} = \frac{A_1}{A_2}$$

or $$Q_2 = Q_1(A_2/A_1) = Q_1(D_2/D_1)^2 \quad ...(25.6)$$

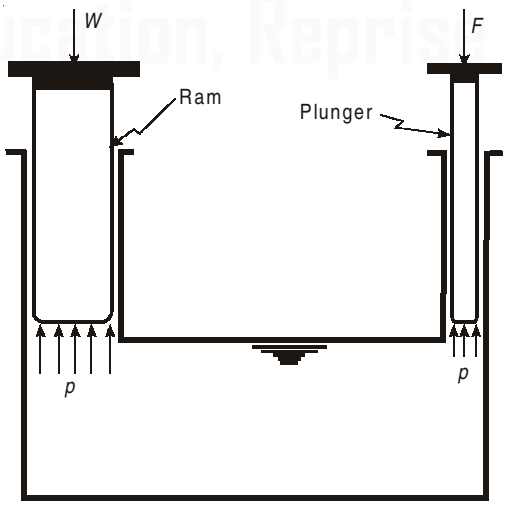
Sometimes compressed air is supplied to the fixed cylinder instead of low pressure liquid, in which case it is known as *Hydro-Pneumatic Intensifier*. Steam under pressure may also be supplied to the fixed cylinder instead of low pressure liquid or compressed air. It is then called *Steam intensifier*.

25.4 HYDRAULIC PRESS

Hydraulic press was first built by Joseph Bramah in 1795, and is still in use since it was first developed. It is a machine which is based on Pascal's law of transmission of fluid pressure and in this machine by the application of a small force a large force may be developed. The working principle of a hydraulic press may be explained with the help of Fig. 25.4. Consider a ram and a plunger operating in two cylinders of different diameters which are inter connected at the bottom through a chamber which is filled with some liquid. Let A be the area of the ram and a be the area of the plunger. Now if F is the force applied to the plunger then the corresponding pressure intensity developed is $p = (F/a)$. But according to Pascal's law, the same pressure intensity will be transmitted throughout the liquid, and therefore the ram will also be subjected to the same pressure intensity. Accordingly if W is the total weight lifted by the ram then $W = (pA)$ and hence

$$p = \frac{F}{a} = \frac{W}{A} \quad ...(25.7)$$

or $$W = F\left(\frac{A}{a}\right) \quad ...(25.8)$$



Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Figure 25.4 *Working principle of hydraulic press*

From Eq. 25.8 it may be seen that by applying a small force F on the plunger a large force W may be developed at the ram. Also the mechanical advantage for this machine is equal to (A/a), the ratio of the areas of the ram and the plunger. As such by suitably adjusting the area of the plunger and the ram even a small force may be multiplied many times.

Figure 25.5 shows the elements of a hydraulic press. In its simplest form it consists of a fixed table and a moving table mounted on sliding rams. It is usually preferred to have multiple rams i.e., a number of rams of smaller size instead of one single ram of bigger size. The advantage of having multiple rams is that the total thrust on the moving table can be controlled to some extent by operating one or all the rams. The rams are operated by liquid under pressure which is supplied by pumps. Usually a hydraulic accumulator is provided in between the press and the pump, which permits the high pressure liquid to be stored in the accumulator cylinder while the press is at rest. In some large hydraulic presses a maximum total thrust ranging from about 50 MN to 100 MN can be produced.

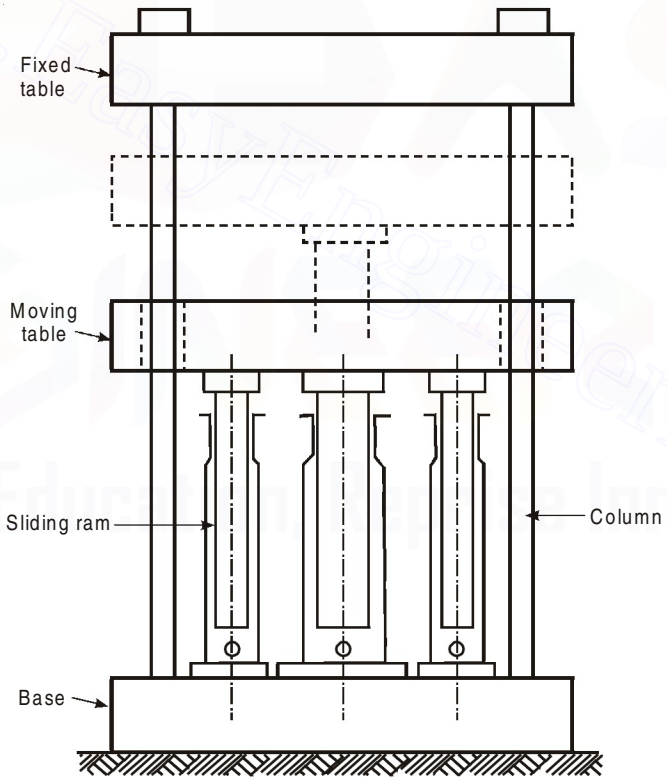


Figure 25.5 *Elements of hydraulic press*

25.5 HYDRAULIC CRANE

The hydraulic crane is a device which is used for lifting heavy loads. It can lift loads upto about 25

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

MN, and it is widely used on docks, slidings, warehouses and workshops. It consists of a central pedestal supporting a mast from which is suspended a jib or arm. The jib can be raised or lowered in order to reduce or increase the radius of action. The pedestal along with the mast can revolve about a vertical axis and the jib swings with the mast. By revolving the pedestal and lowering the jib the suspended load may be moved to any place within the crane's area of action. To the mast of the crane is attached a jigger, which consists of a ram sliding in a fixed cylinder, both having a set of pulleys (or a pulley sheave) at their ends as shown in Fig. 25.6. When liquid under pressure is admitted into the cylinder the ram will move vertically up and it will descend when the outlet valve (not shown in the figure) is opened and the liquid is removed from the cylinder. As such the set of pulleys attached to the ram will move with it, while the one attached to the cylinder will remain fixed. A wire rope with one of its ends fixed to a movable pulley attached to the ram of the jigger, is taken round all the pulleys of the two sets of pulleys and then it is stretched to pass over a guide pulley attached to the jib, leaving the other end of the rope free. The load to be lifted by the crane is suspended at the free end of the wire rope which is usually provided with a hook for suspending the load.

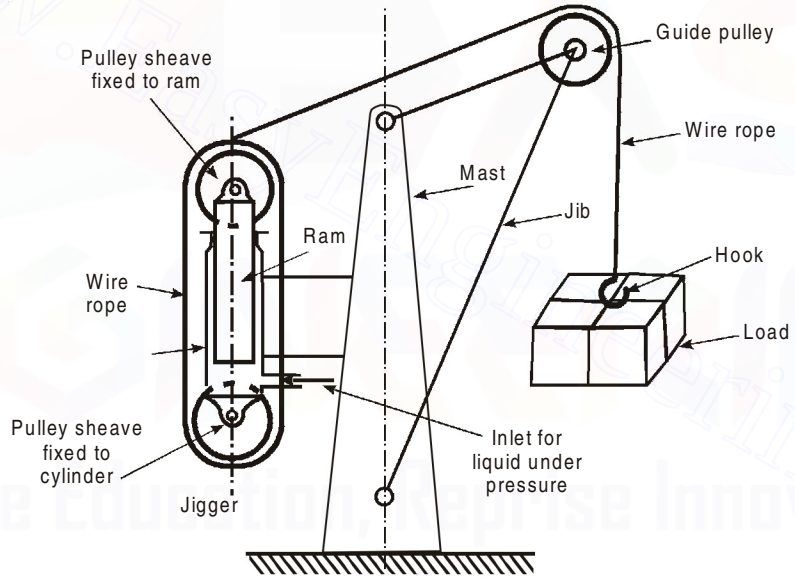


Figure 25.6 *Hydraulic crane*

When the load is to be lifted by the crane, the liquid from the cylinder of the jigger is removed through the outlet valve, which causes the ram to move down. The distance between the two sets of pulleys is then reduced, which results in releasing more length of the wire rope and the hook provided at the free end of the wire rope is lowered down to the load. The load to be lifted is attached to the hook. Now the liquid under pressure is admitted into the cylinder of the jigger, which will push the ram vertically up. The distance between the two sets of pulleys is then increased, which results in winding of the wire rope and the load is thus lifted up. The suspended load may be moved to any desired place within the crane's area of action by revolving the mast and lowering or raising the jib. Again by removing the liquid from the cylinder of the jigger, the hook along with the load is lowered as indicated above and the load is then placed at the desired place. The velocity ratio of the crane hook to the ram

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of the jigger depends on the number of pulleys in each set. Thus if there are six pulleys in a set then the velocity ratio will be six to one which means that the load suspended to the hook of the crane will move at six times the speed of the ram of the jigger. A modern hydraulic crane may have a lifting speed of about 75 m per minute. However these days hydraulic cranes have been replaced by the electric cranes.

25.6 HYDRAULIC LIFT

The hydraulic lift is a device which is used for carrying the goods as well as the persons from one floor to another in a multi-storeyed building. A simple form of a hydraulic lift consists of a ram sliding in a cylinder. At the top of the ram a platform or a cage is fitted on which the goods may be placed or the persons may stand. The liquid under pressure is admitted into the cylinder, which pushes the ram vertically upwards, thus raising the platform or the cage to the required height. The platform or the cage can be made to stay in level with each floor, so that the goods can be transferred to that floor or the persons can walk over to that floor. Again by removing the liquid from the cylinder, the ram and hence the platform or the cage can be made to move downwards. This type of lift is known as *direct acting lift* which is shown in Fig. 25.7 (a).

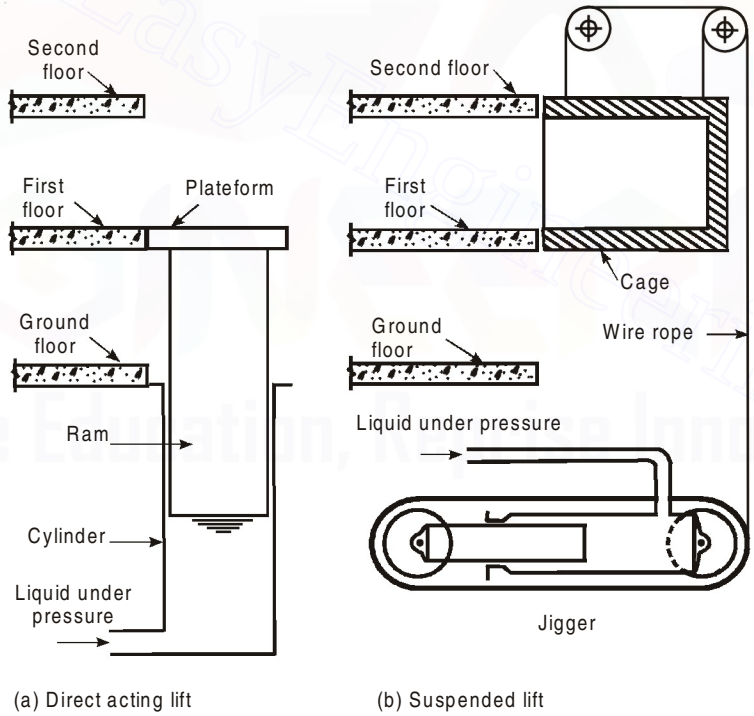


Figure 25.7 *Hydraulic lifts*

Another hydraulic lift called *suspended hydraulic lift* is a modified form of a hydraulic lift which is shown in Fig. 25.7 (b). It is fitted with a jigger which is exactly same as in the case of a hydraulic crane. Further the raising or lowering of the cage of the lift is facilitated by the jigger in the same way as in the case of the hydraulic crane. The lift cage runs between guides of hard wood or round steel and is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

usually suspended by four lifting ropes, each one being of sufficient strength to support the load. In order to balance the weight of the cage, sliding balance weights are provided. Modern hydraulic lifts have the lifting speeds ranging from 100 to 120 m per minute. However, the hydraulic lifts have been replaced by the electric lifts which have become quite common these days.

25.7 HYDRAULIC RAM

The hydraulic ram or (hydram) is a type of pump in which the energy of large quantity of water falling through small height is utilised to lift a small quantity of this water to greater height. No external power is, therefore, required to operate this pump. It can be used for water supply to countryside and remote areas where a source having large quantity of water at some height is available and where the power is scarce so that the other types of pumps cannot be employed.

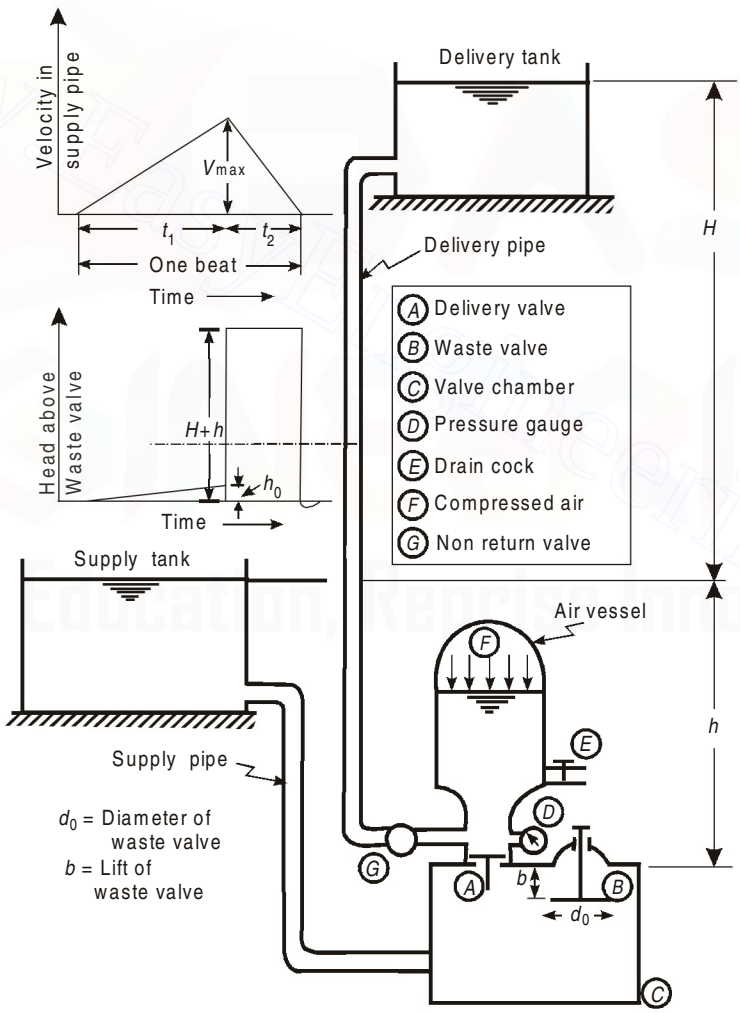


Figure 25.8 *Hydraulic ram*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As shown in Fig. 25.8, a hydraulic ram consists of a valve chamber *C* having a waste valve *B* and a delivery valve *A*. The waste valve opens into a waste water channel and the delivery valve opens into an air vessel at the base of which a delivery pipe is connected which carries water to the delivery tank above. The valve chamber is connected to a low level supply tank through a supply pipe.

The working of a hydraulic ram is based on the principle of water hammer or inertia pressure developed in the supply pipe. Initially as the water flows down the supply pipe into the valve chamber, the waste valve being open, the water flows through it to the waste water channel. As the rate of discharge past the waste valve increases, the flow of water in the supply pipe accelerates (i.e., the velocity of flow of water in the supply pipe increases). Due to the acceleration of the water column in the supply pipe, an appreciable increase of pressure in the valve chamber takes place, in order to develop the necessary velocity of flow of water through the waste valve. The pressure in the valve chamber rapidly increases to such a value at which the static thrust (due to the static column of water in the supply tank) together with the dynamic thrust (due to the accelerating flow) acting on the lower face of the waste valve is greater than the downward force due to the weight of the valve. The waste valve then almost instantaneously closes. The instantaneous closing of the waste valve brings the water in the supply pipe suddenly to rest, causing a further increase of pressure in the valve chamber due to the development of inertia pressure. Due to this increase of pressure in the valve chamber (which is almost immediately after the closure of the waste valve) the delivery valve is forced open. The water then flows from the supply tank through the delivery valve into the air vessel and the delivery pipe. Thus some of the water flowing through the delivery valve is directly supplied to the delivery tank and some of it is stored in the air vessel. The water flowing in the air vessel compresses the air inside it, which pushes a part of the water in the delivery pipe even when the delivery valve is closed. Thus an air vessel of a hydraulic ram assists in providing a continuous delivery of water at a more or less uniform rate. The flow of water through the delivery valve continues until the pressure in the valve chamber is reduced, the delivery valve then closes and the waste valve opens, thus again causing the water to flow from the supply tank to the waste water channel. This constitutes one cycle of operation or one beat of the hydraulic ram. The same cycle is then repeated.

Let d be the diameter of the supply pipe; d_0, b and W be respectively the diameter, lift and weight of the waste valve. Further let V_0 be the velocity of flow of water past the waste valve just before its closure, then the dynamic pressure head (p_i/w) or h_0 acting on the waste valve is given as

$$h_0 = \frac{p_i}{w} = \frac{V_0^2}{2g} \qquad ...(25.9)$$

Also just before the closure of the waste valve the force acting vertically upwards on the waste valve is equal to the weight of the waste valve, thus

$$\frac{\pi}{4} d_0^2 \times p_i = W$$

or

$$\frac{p_i}{w} = h_0 = \frac{W}{(\pi/4)\, d_0^2 w} \qquad ...(25.10)$$

Again if V_m is the maximum velocity of flow of water in the supply pipe just before the closure of the waste valve, then by continuity equation

$$(\pi d_0 b)\, V_0 = \left(\frac{\pi}{4} \times d^2\right) V_m \qquad ...(25.11)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Also if t_1 is the time required to build up the velocity of flow of water in the supply pipe from 0 to the maximum velocity V_m or it is the time for which the waste valve remains open during each beat, then

$$h = \frac{l_s}{g}\frac{V_m}{t_1} \qquad ...(25.12)$$

where h is the level of water in the supply tank above the waste valve and l_s is the length of the supply pipe. Equation 25.12 is based on the principle of water hammer.

Similarly if t_2 is the time during each beat for which the waste valve remains closed or the delivery valve remains open and H is the level of water in the delivery tank above that in the supply tank then

$$H = \frac{l_s}{g}\frac{V_m}{t_2} \qquad ...(25.13)$$

Thus the time taken for one complete cycle or one beat is t which is given by

$$t = t_1 + t_2 = \frac{l_s V_m}{g}\left(\frac{1}{h} + \frac{1}{H}\right) \qquad ...(25.14)$$

The number of beats per minute N, is equal to $(60/t)$ where t is time for one beat in seconds.

Further if q is the rate of discharge of water actually lifted by the ram and Q is the rate of discharge of water flowing past the waste valve, then since mean velocity of flow in the supply pipe is $(V_m/2)$, we have

$$q = \left(\frac{\pi}{4}d^2\right) \times \frac{V_m}{2} \times \frac{t_2}{t} \qquad ...(25.15)$$

and

$$Q = \left(\frac{\pi d^2}{4}\right) \times \frac{V_m}{2} \times \frac{t_1}{t} \qquad ...(25.16)$$

The working cycle of a hydraulic ram represented graphically is also shown in Fig. 25.8, where in the upper diagram the velocity in the supply pipe and in the lower diagram the pressure head in the valve chamber are plotted against the time for one beat only.

In Eqs. 25.12 and 25.13 the loss of head due to friction in the supply and the delivery pipe has been neglected. However if loss of head due to friction in supply pipe $(h_f)_s$ and that in delivery pipe $(h_f)_d$ are taken into account, then Eqs. 25.12 and 25.13 are modified as follows

$$h - (h_f)_s = \frac{l_s}{g} \times \frac{V_m}{t_1} \qquad ...(25.17)$$

$$H + (h_f)_d = \frac{l_s}{g} \times \frac{V_m}{t_2} \qquad ...(25.18)$$

There are two ways of assessing the overall efficiency of a hydraulic ram. In the Rankine system the water surface in the supply reservoir is taken as datum and hence the energy input during one cycle is (wQh), and the useful output during the same cycle is (wqH), where w is the specific weight of water. Thus

$$\text{Rankine's efficiency} = \eta_r = \frac{(wqH)}{(wQh)} = \frac{qH}{Qh} \qquad ...(25.19)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In another system the datum may be taken to be passing through the waste valve, and hence the energy input during one cycle is $[w(Q+q)h]$ and the useful output during the same cycle is $[wq(H+h]$. The corresponding overall efficiency then becomes

$$\eta_d = \frac{wq(H+h)}{w(Q+q)h} = \frac{(H+h)q}{h(Q+q)} \qquad ...(25.20)$$

which is known as D'Aubuisson's efficiency, and its value is always more than the Rankine's efficiency.

On account of several energy losses the maximum efficiency of hydraulic ram is usually limited to only about 75%. Some of the main causes of energy loss are friction and secondary losses in the supply and delivery pipes and in the valves; and the velocity energy carried away by the water leaving the waste valve.

25.8 HYDRAULIC COUPLINGS AND TORQUE CONVERTERS

These are the devices used for transmission of power through a liquid medium. A *hydraulic* (or *fluid*) *coupling* consists of a pump impeller which is attached to a driving shaft and a turbine runner attached to a driven shaft. These two units are enclosed in a single housing which contains a liquid, usually oil, because of its lubricating power, availability and stability. No direct contact exists between the driving parts and the driven parts. The oil in the housing transmits the moment of momentum or torque from the pump impeller to the turbine runner.

A typical fluid coupling is shown in Fig. 25.9. As the driving shaft is started, the pump impeller causes the fluid to flow from the eye of the pump impeller to the outer periphery of the pump impeller, where it is discharged inwardly through the turbine runner back to the pump. As the speed of the driving shaft is increased, the liquid torque on the turbine runner increases until it overcomes the inertia of the driven unit, and the turbine runner and shaft begin to rotate.

The *power input* to the coupling can be defined as

$$P_i = T\omega_p \qquad ...(25.21)$$

in which T is the torque and ω_p is the angular speed of the pump or drive shaft. The *power output* from the coupling is

$$P_0 = T\omega_t \qquad ...(25.22)$$

in which ω_t is the angular speed of the turbine shaft.

The efficiency η of the fluid coupling is defined as the ratio of power output to power input. Thus

$$\eta = \frac{T\omega_t}{T\omega_p} = \frac{\omega_t}{\omega_p}$$

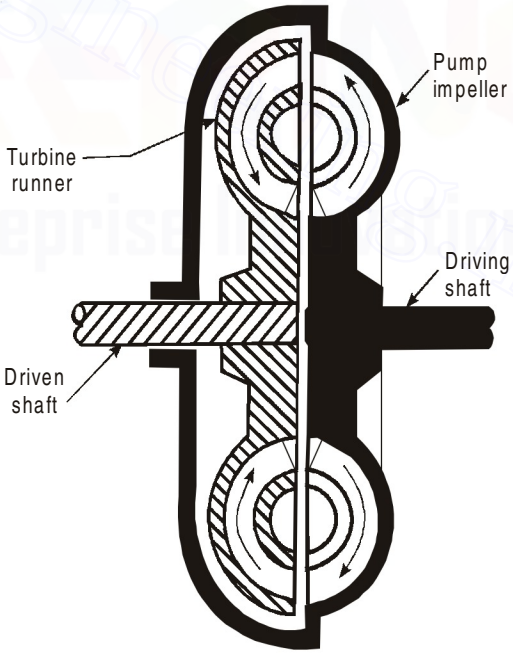


Figure 25.9 *Typical hydraulic coupling*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Further the slip is defined as

$$s = \frac{\omega_p - \omega_t}{\omega_p} = 1 - \eta \qquad \ldots(25.24)$$

The efficiency of fluid coupling is large, usually in excess of 94 percent. A typical efficiency versus *speed ratio* (ω_t/ω_p) curve for a fluid coupling is shown in Fig. 25.10. The efficiency of the fluid coupling starts at zero and increases uniformly with the speed ratio until $\eta > 95$ percent and then reduces to zero.

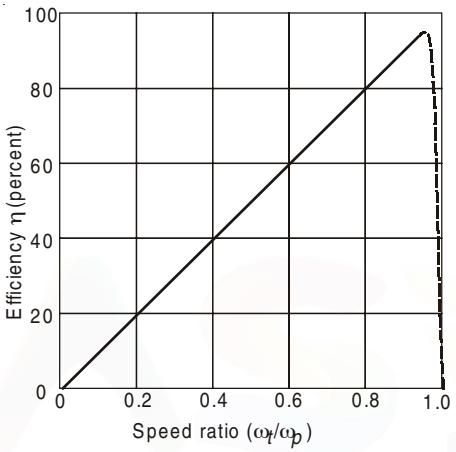


Figure 25.10 *Efficiency-speed ratio curve for a fluid coupling*

Fluid couplings are widely used with both electrical and internal combustion prime movers (ranging in size from 0.7 kW to 26.5 MW or 1 to 36 000 h.p.) to transmit torque. The fluid couplings are particularly useful where smooth shock-free operations are required and where large initial loads are involved.

A *hydraulic torque converter* is similar to the hydraulic coupling, except for a series of fixed guide vanes which are provided between the impeller and the turbine runner. In this case the liquid exerts a torque on the driven unit through these stationary guide vanes as it flows from pump to turbine. As such by suitably designing the stationary guide vanes the torque transmitted to the driven unit can be either increased or decreased. The torque-relationship is

$$T_t = T_p + T_v \qquad \ldots(25.25)$$

in which torque T_t is the torque transmitted to the turbine runner, T_p is the torque of the pump impeller, and T_v is the variation in the torque caused by the fixed guide vanes. In Eq. 25.25, T_v can be positive or negative. That is, if the stationary guide vanes are designed to receive a torque from the fluid which is in the opposite direction to that exerted on the driven shaft, an increased torque output results. Conversely, if the vanes are designed to receive a torque which is in the same sense as that of the driven shaft, a torque reduction results. As such whereas in the case of hydraulic coupling the torque output is equal to the torque input, in the case of hydraulic torque converter the torque output can be increased or reduced, and therefore the later is comparable to an electric transformer.

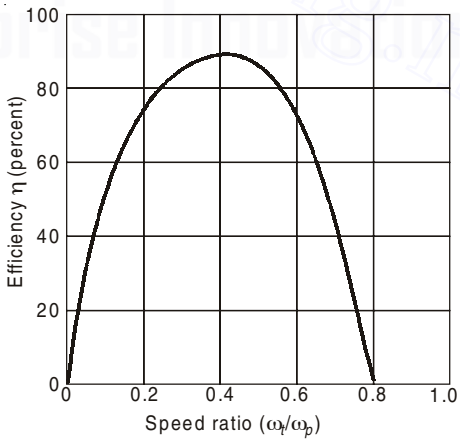


Figure 25.11 *Variation of torque converter efficiency*

To attain a large reduction in speed, $\omega_t << \omega_p$, and a large torque magnification, hydraulic torque converters are designed which utilize two or more sets of turbine runners and fixed guide vanes, the fixed vanes being located between the turbine runners.

Figure 25.11 shows the variation of torque converter efficiency with speed ratio (ω_t/ω_p). From the efficiency

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

curve, it is apparent that the efficiency of the torque converter is better at smaller speed ratios than that of the hydraulic coupling.

25.9 AIR LIFT PUMP

The air lift pump is a device which is used for lifting water from a well by utilizing the compressed air. As shown in Fig. 25.12 the compressed air is introduced through one or more nozzles at the bottom of the air supply pipe, into water at the foot of the rising main fixed in the well. Thus a mixture of air and water is formed within the rising main. Since the density of air water mixture is much less than that of pure water, a very long column of air water mixture will be required to balance even a very small column of pure water. As such the flow of air water mixture will begin through the rising main and it will be issued continuously at the top of the rising main so long as the supply of air is maintained.

If H is the submergence which is the height of static water level in the well above the tip of the nozzle and L is the height of the point to which the water is lifted above the tip of the nozzle, then $(L - H)$ is known as the 'useful lift'. It has been observed that the best results are obtained if the useful lift $(L - H)$ is less than the submergence H. In general the ratio $\left(\frac{H}{L-H}\right)$ varies from 4 when H is about 0.3 m to 1 when H is about 90 m.

The main advantage of the air lift pump is that it has no moving parts below water level and therefore cannot be damaged by solids in suspension in the water. Also it can raise more water

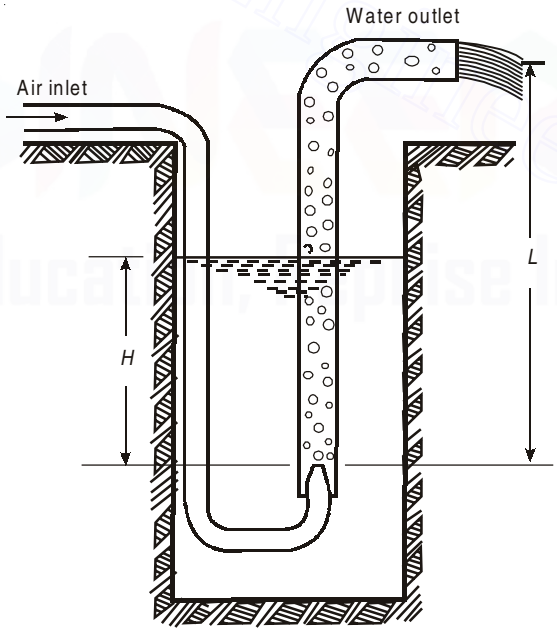


Figure 25.12 *Air lift pump*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

through a bore hole of given diameter than any other pump. However, the efficiency of the pump is low, since out of the energy expended in compressing the air, only between 20 to 40 percent is available in the form of useful power.

ILLUSTRATIVE EXAMPLES

Example 25.1. *An accumulator has a ram of 0.2 m diameter and lift of 6 m. Water is supplied at a pressure of 5900 kN/m^2. Find the necessary load on the ram and the capacity of the accumulator in kW hours.*

Solution

$$\text{Load on ram} = pA = 5900 \times \frac{\pi}{4} \times 0.2^2$$

$$= 185.35 \text{ kN.}$$

Capacity of the accumulator

$$pAH = (185.35 \times 6) = 1112.1 \text{ kN-m}$$

Since 1 kW hour = $(1000 \times 60 \times 60)$ N-m

$\therefore$ Capacity of accumulator

$$= \frac{1112.1 \times 10^3}{1000 \times 60 \times 60} = 0.309 \text{ kW hour}$$

Example 25.2. *An accumulator has a ram 0.3 m diameter and 6 m lift and is loaded with 800 kN total weight. If packing friction is equivalent to 5 percent of the load on the ram, determine the power being delivered to the machine, if the ram falls steadily through its full range in 90 seconds and if at the same time the pumps are delivering 30 litres per second through the accumulator.*

Solution

The pressure of water produced by the falling ram of the accumulator is equal to the pressure of the water supplied to the machine. Thus the pressure head H when the loaded ram is falling is

$$H = \frac{p}{w} = \frac{800 \times 10^3 \times (1 - 0.05)}{(\pi/4)(0.3)^2 \times 9810}$$

$$= 1096 \text{ m of water}$$

Power supplied by the pump

$$= wQH$$

$$= 9810 \times 30 \times 10^{-3} \times 1096$$

$$= 322\,553 \text{ W} = 322.553 \text{ kW}$$

Power supplied by the accumulator

$$= \text{weight} \times \text{distance moved/second}$$

$$= \frac{(800 \times 10^3 \times 0.95) \times 6}{90}$$

$$= 50\,667 \text{ W} = 50.667 \text{ kW}$$

$\therefore$ Total power delivered to the machine

$$= (322.553 + 50.667) = 373.22 \text{ kW}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Example 25.3. *A power of 30 kW is required to be applied to the ram of a press from an accumulator through a 100 mm diameter pipe 1500 m long. If the loss in transmission is to be 2 percent find the diameter of the ram of the press which is loaded with 1200 kN. Take Darcy's f for pipe as 0.04.*

Solution

Let H be the head supplied by the accumulator and V be the velocity of flow in the pipe.

Then $$h_f = \frac{2H}{100} = \frac{flV^2}{2gd} = \frac{0.04 \times 1500 \times V^2}{2 \times 9.81 \times 0.10}$$

or $$V_2 = (6.54 \times 10^{-4})H$$

or $$V = (2.56 \times 10^{-2}) \sqrt{H}$$

$\therefore$ Discharge $$Q = aV$$

$$= (\pi/4)\ (0.10)^2\ (2.56 \times 10^{-2})\ \sqrt{H}$$

$$= 2.01 \times 10^{-4} \sqrt{(H)}$$

$\therefore$ $$\text{Power} = 30 \times 10^3 = wQH\ (1 - 0.02)$$

$$= 9810 \times 2.01 \times 10^{-4} \sqrt{(H)} \times H \times 0.98$$

or $$H = \left(\frac{30 \times 10^3}{9810 \times 2.01 \times 10^{-4} \times 0.98} \right)^{2/3}$$

$$= 622.33 \text{ m}$$

Now if D is the diameter of the ram of the press, then

$$\frac{1200 \times 10^3}{(\pi/4)D^2 \times 9810} = 622.33$$

$$D^2 = 0.25$$

$\therefore$ $$D = 0.5 \text{ m}$$

i.e., the diameter of the ram of the press = 0.5 m.

Example 25.4. *The diameter of the two parts of the ram of a differential accumulator are 150 mm and 130 mm, and stroke length is 1.2 m. If the pressure of water is 9810 kN/m^2 when the load is either at rest at the upper end of the stroke or the load is moving with uniform velocity, what will be the weight of the loaded cylinder? How much energy can be stored in this accumulator? What will be the diameter of the ram of an ordinary accumulator to move the same load with the help of the same water pressure?*

Solution

Area of the brass bush or sleeve

$$a = \frac{\pi}{4}[(0.15)^2 - (0.13)^2] = 4.4 \times 10^{-3} \text{ m}^2$$

$\therefore$ Weight of the loaded cylinder is

$$W = (p \times a)$$

$$= (9810 \times 4.4 \times 10^{-3}) = 43.164 \text{ kN}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Energy stored in the accumulator

$$= WH$$
$$= (43.164 \times 1.2)$$
$$= 51.8 \text{ kN.m} = 51.8 \times 10^3 \text{ J}$$

For an ordinary accumulator

$$W = p \times A$$

or $$43.164 = 9810 \times \frac{\pi}{4} \times D^2$$

$$\therefore \quad D = 7.48 \times 10^{-2} \text{ m}$$
$$= 74.8 \text{ mm} \approx 75 \text{ mm}$$

Example 25.5. *A hydraulic accumulator has sliding ram of 500 mm diameter which slides through 10 m in 4 minutes during its working stroke, while weight on the ram including its self weight is equivalent to 400 kN. The pump supplies water at 0.01 m^3/s rate and packing friction amounts to 5% of total load. Determine (i) pressure intensity of water; (ii) power delivered to machine supplied by accumulator; (iii) power required to drive the pump having efficiency 76%.*

Solution

Diameter of sliding ram

$$D = 500 \text{ mm} = 0.5 \text{ m}$$

$$\therefore \quad \text{Area } A = \frac{\pi D^2}{4}$$

$$= \frac{\pi \times (0.5)^2}{4} = 0.196 \text{ m}^2$$

Weight on ram including self weight

$$W = 400 \text{ kN}$$

Packing friction $= 5\%$ of total load

$$= \frac{5}{100} \times 400 = 20 \text{ kN}$$

$\therefore$ Net load on the sliding ram

$$= (400 - 20) = 380 \text{ kN}$$

(i) Pressure intensity of water

$$p = \frac{380}{0.196} = 1938.78 \text{ kN/m}^2$$

(ii) If Q is the total discharge of water supplied to the machine, q is the discharge of water supplied by the pump and h is the stroke length of the accumulator ram, then

$$Ah = (Q - q)\, t$$

or $$0.196 \times 10 = (Q - 0.01) \times (4 \times 60)$$

$$\therefore \quad Q = 0.0182 \text{ m}^3/\text{s}$$

Therefore power delivered to the machine

$$= p \times Q$$
$$= 1938.78 \times 10^3 \times 0.0182$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 35.286 \times 10^3 \text{ W}$$
$$= 35.286 \text{ kW}$$

(iii) Power required to drive the pump

$$= \frac{pq}{\eta}$$

$$= \frac{1938.78 \times 10^3 \times 0.01}{0.76}$$
$$= 25.51 \times 10^3 \text{ W}$$
$$= 25.51 \text{ kW}$$

Example 25.6. *An hydraulic intensifier supplying water to a hydraulic press gets water from supply main at a pressure of 500 kPa. It has to intensify the pressure to 40 000 kPa. The stroke of the intensifier is 1.2 m and its capacity is 24 litres. Find the diameter of the ram and the external diameter of the sliding cylinder of the intensifier.*

Solution

If a is the cross-sectional area of the fixed ram and H is the stroke then the capacity of the intensifier

$$= (a \times H) = 24 \times 10^{-3} \text{ m}^3$$

or $$\frac{\pi}{4} \times d^2 \times 1.2 = 24 \times 10^{-3}$$

$$\therefore \quad d = 0.16 \text{ m} = 160 \text{ mm}$$

Also from Eq. 25.5, we have

$$p_1 A_1 = p_2 A_2$$

or $$5000 \times \left(\frac{\pi}{4} \times D^2\right) = 40\,000 \times \left(\frac{\pi}{4} \times 0.16^2\right)$$

$$D = 0.453 \text{ m} = 453 \text{ mm}$$

Example 25.7. *An intensifier has a ram of diameter 0.2 m and sliding cylinder of diameter 1 m. Calculate the pressure of water on low pressure side if the pressure of water on high pressure side is to be 24 500 kN/m^2. The loss due to friction at each of the packing of the intensifier is 5% of total pressure on each piston.*

Solution

Force on the end of the sliding cylinder

$$= pA$$

5% of it is lost due to friction at the packing.

Therefore net force on the sliding cylinder

$$= (p \times A)\, 0.95 = F$$

Again 5% of this force F is lost due to friction at the packing between the fixed ram and the sliding cylinder.

Thus net force on the fixed ram

$$= F \times 0.95$$

Also if p_1 is the pressure intensity on the fixed ram and a is its cross-sectional area, then

$$F \times 0.95 = p_1 \times a$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $(p \times A \times 0.95) \times 0.95 = p_1 \times a$

or $p \times \frac{\pi}{4} \times (1)^2 \times (0.95)^2 = 24\,500 \times \frac{\pi}{4} \times (0.2)^2$

$\therefore \quad p = 1086 \text{ kN/m}^2$

Example 25.8. *Water, to a hydraulic intensifier with diameter of sliding ram and fixed ram being 250 mm and 100 mm respectively, is supplied through a pipeline 80 m long, 50 mm diameter and with Darcy's f = 0.0225 under a head of 12.5 m to fixed cylinder. During working stroke the sliding ram travels 0.8 m in 30 seconds. Determine (i) pressure head delivered at outlet of fixed ram; (ii) power delivered by intensifier during working stroke. (iii) Revise these figures if each packing is assumed to offer friction of 5% of imposed load.*

Solution

Diameter of sliding ram

$D_1 = 250 \text{ mm} = 0.25 \text{ m}$

$\therefore \quad \text{Area } A_1 = \frac{\pi}{4} \times (0.25)^2 = 0.0491 \text{ m}^2.$

Diameter of fixed ram

$D_2 = 100 \text{ mm} = 0.10 \text{ m}$

$\therefore \quad \text{Area } A_2 = \frac{\pi}{4} \times (0.10)^2 = 0.00785 \text{ m}^2$

Stroke length of sliding ram

$h = 0.8 \text{ m}$

The discharge of low pressure liquid entering the fixed cylinder

$$Q_1 = \frac{A_1 h}{t}$$

$$= \frac{0.0491 \times 0.8}{30} = 1.31 \times 10^{-3} \text{ m}^3/\text{s}$$

The head loss due to friction in the supply pipe

$$h_f = \frac{fLV^2}{2gd} = \frac{fLQ^2}{12.1d^5}$$

$$\therefore \quad h_f = \frac{0.0225 \times 80 \times (1.31 \times 10^{-3})^2}{12.1 \times (0.05)^5}$$

$$= 0.82 \text{ m}$$

The pressure head of the low pressure liquid in the fixed cylinder

$H_1 = (12.5 - 0.82)$

$= 11.68 \text{ m}$

(i) Therefore the pressure head delivered at the outlet of the fixed ram

$$H_2 = H_1\left(\frac{A_1}{A_2}\right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= 11.68\left(\frac{0.0491}{0.00785}\right)$$

$$= 73.06 \text{ m}$$

(ii) The discharge of the high pressure liquid delivered by the intensifier

$$Q_2 = Q_1\left(\frac{A_2}{A_1}\right)$$

$$= 1.31 \times 10^{-3} \times \left(\frac{0.00785}{0.0491}\right)$$

$$= 2.094 \times 10^{-4} \text{ m}^3/\text{s}$$

Therefore power delivered by the intensifier

$$= w Q_2 H_2$$
$$= (9810 \times 2.094 \times 10^{-4} \times 73.06)$$
$$= 150.08 \text{ W}$$

(iii) Since there are two packings and each packing offers a friction of 5% of imposed load, the pressure head delivered at the outlet of the fixed ram

$$H_2' = H_1\left(\frac{A_1}{A_2}\right)(1-0.05)^2$$

$$= 11.68 \times \left(\frac{0.0491}{0.00785}\right)(0.95)^2$$

$$= 65.93 \text{ m}$$

Power delivered by the intensifier

$$= w Q_2 H_2'$$
$$= (9810 \times 2.094 \times 10^{-4} \times 65.93)$$
$$= 135.43 \text{ W}$$

Example 25.9. *In a testing machine the force is applied to specimen by hydraulic pressure on a ram of 0.25 m diameter. The maximum force required is 1 000 kN and the frictional resistance at the ram may be taken as an additional 50 kN. The water is supplied from an accumulator consisting of a vertical cylinder with a loaded plunger 1.125 m diameter. Find the necessary load on the plunger if friction causes a resistance of 20 kN to its motion.*

Solution

The total pressure required on the ram

$$= (1000 + 50) = 1050 \text{ kN}$$

$\therefore$ Pressure intensity required to be developed at the ram of the machine

$$= \frac{1050}{(\pi/4)(0.25)^2} = 21390.4 \text{ kN/m}^2$$

Let W kN be the load on the plunger, then the net force is $(W - 20)$ kN.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The pressure intensity developed by the plunger of the accumulator

$$= \frac{(W-20)}{(\pi/4)(1.125)^2}$$

Thus equating the two, we get

$$21390.4 = \frac{(W-20)}{(\pi/4)(1.125)^2}$$

$$\therefore \quad W = 21\,282.48 \text{ kN}$$

Example 25.10. *Find the power of the motor to drive the plunger of a hydraulic press lifting 10 kN to a height of 1 m in 10 minutes. The diameter of ram and plunger are 150 mm and 25 mm respectively. Take stroke of plunger as twice of diameter of the ram. Find also the number of strokes required by the plunger.*

Solution

Work done by the press per second

$$= \frac{(10\times10^3)\times1}{10\times60} = 16.67 \text{ N-m}$$

$\therefore$ Power of the motor required to drive the plunger

$$= 16.67 \text{ N-m/s} = 16.67 \text{ W}$$

Volume of water displaced as the ram is lifted through 1 m

$$= \left[\frac{\pi}{4}(0.15)^2\times1\right] \text{m}^3$$

Similarly the volume of water displaced during each stroke of the plunger

$$= \left[\frac{\pi}{4}(0.025)^2\times2\times0.15\right] \text{m}^3$$

$$\therefore \quad \text{No. of strokes} = \frac{\frac{\pi}{4}(0.15)^2\times1}{\frac{\pi}{4}(0.025)^2\times2\times0.15} = 120$$

Example 25.11. *A hydraulic press has a ram of 150 mm diameter and plunger of 30 mm diameter, with stroke length of 250 mm. Weight exerted by press ram amounts to 5886 N and distance moved is 1.2 m in 20 minutes. Determine (i) the force applied on the plunger; (ii) number of strokes performed by the plunger; (iii) work done by the press ram; and (iv) power required to drive the plunger.*

Solution

Diameter of ram $D = 150 \text{ mm} = 0.15 \text{ m}$

Area of ram $A = \frac{\pi}{4}(0.15)^2 = 0.0177 \text{ m}^2$

Diameter of plunger $d = 30 \text{ mm} = 0.03 \text{ m}$

Area of plunger $a = \frac{\pi}{4}(0.03)^2 = 0.00707 \times 10^{-1} \text{ m}^2$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Weight exerted by press ram

$$W = 5886 \text{ N}$$

(i) Force applied on the plunger

$$F = \left(\frac{a}{A}\right) W$$

$$= \left(\frac{0.00707 \times 10^{-1}}{0.0177}\right) \times 5886$$

$$= 235.11 \text{ N}$$

(ii) Stroke length of plunger $x = 250 \text{ mm} = 0.25 \text{ m}$

Distance moved by ram $y = 1.2 \text{ m}$

Therefore number of strokes performed by the plunger

$$n = \frac{Ay}{ax}$$

$$= \frac{0.0177 \times 1.2}{0.00707 \times 10^{-1} \times 0.25}$$

$$= 120.17 \approx 120$$

(iii) Work done by the press ram

$$= (5886 \times 1.2) \text{ N.m}$$

$$= 7063.2 \text{ N.m}$$

(iv) Power required to drive the plunger is equal to the work done by the press ram per second.

Thus power required to drive the plunger

$$= \frac{7063.2}{20 \times 60}$$

$$= 5.886 \text{ W}$$

Example 25.12. *A hydraulic lift raises a load of 80 kN through a height of 12 m once every 2 minutes, the speed of lifting being 0.6 m per second. It is worked from an accumulator which is being constantly charged by a pump. The pressure of water is 3 450 kN/m^2, the efficiency of lift is 75 percent and the efficiency of the pump 85 percent. Find the power required to drive the pump and the minimum capacity of the accumulator. Friction losses in the pipes may be neglected.*

Solution

Power required by the lift in raising the load

$$= 80 \times 10^3 \times 0.6$$

$$= 48\,000 \text{ W} = 48 \text{ kW}$$

Efficiency of lift $= 0.75$

$\therefore$ Actual power required to be supplied to the lift

$$= \frac{48}{0.75} = 64 \text{ kW}$$

This power will be supplied by the pump as well as the accumulator.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Now if Q is the rate of discharge of the pump then the power supplied by the pump

$$= \frac{wQH}{1000} \text{ kW}$$

but $$H = \frac{3450 \times 10^3}{9810} = 351.682 \text{ m of water}$$

$\therefore$ The power supplied by the pump

$$= \frac{9810 \times Q \times 351.682}{1000} = 3\,450\ Q$$

Time taken by the lift to raise the load

$$= \frac{12}{0.6} = 20 \text{ seconds}$$

Thus out of 120 seconds the lift will work only for 20 seconds and for the rest 100 seconds the water will be stored in the accumulator. Hence the required capacity of the accumulator = (100 Q) m^3.

Further this quantity of water stored by the accumulator will be supplied by it to the lift in addition to the water supplied from the pump in a period of 20 seconds when the lift is working. Thus the power supplied from the accumulator

$$= 9810 \times \left(\frac{100Q}{20}\right) \times \frac{351.682}{1000} = 17\,250\ Q.$$

$\therefore$ Total power supplied to the lift by the pump and the accumulator

$$= (3\,450 + 17\,250)\ Q = (20\,700\ Q)$$

Thus equating the two, we have

$$20\,700\ Q = 64$$

$$Q = \frac{64}{20700} = 3.09 \times 10^{-3} \text{ m}^3/\text{s}$$

Thus the power required to drive the pump

$$= \frac{9810 \times 3.09 \times 10^{-3} \times 351.682}{1000 \times 0.85}$$

$$= 12.54 \text{ kW}$$

The minimum capacity of the accumulator

$$= (100 \times 3.09 \times 10^{-3}) = 0.309 \text{ m}^3$$

Example 25.13. *The following particulars refer to a hydraulic crane:*

Diameter of crane ram = 0.3 m, velocity ratio of crane hook to ram = 5 : 1, length of supply pipe from accumulator = 150 m, diameter of supply pipe = 50 mm, pressure at accumulator = 5400 kN/m^2, mechanical friction of ram, pulleys etc., are equivalent to a pressure of 490 kN/m^2 on the ram and Darcy's f for the pipe = 0.04.

Obtain a relationship between the load W kN lifted and the speed of lifting V m/s. Hence find V for W = 50 kN.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Load on the ram of the crane × velocity of ram = Load lifted × Velocity of the lifting load (assuming the efficiency of lifting to be unity).

∴ Load on the ram of the crane

$$= \text{Load lifted} \times \text{Velocity ratio}$$
$$= 5\,W$$

Thus the pressure head on the ram of the crane

$$= \left[\frac{5W}{(\pi/4)(0.3)^2} + 490\right] \times \frac{10^3}{9810}$$
$$= (7.21\,W + 49.95) \text{ m of water}$$

Pressure head in accumulator

$$= \frac{5400 \times 10^3}{9810} = 550.46 \text{ m}$$

∴ Head lost in friction in pipe

$$= 550.46 - (7.21\,W + 49.95)$$
$$= (500.51 - 7.21\,W)$$

or $$\frac{flv^2}{2gd} = (500.51 - 7.21\,W)$$

or $$\frac{0.04 \times 150 \times v^2}{2 \times 9.81 \times 0.05} = (500.51 - 7.21\,W)$$

or $$v^2 = (81.83 - 1.18\,W)$$

∴ $$v = (81.83 - 1.18\,W)^{1/2}$$

where v is the velocity of flow of water in the pipe.

Since the velocity ratio is 5, the velocity of the ram = $\frac{V}{5}$

Again the discharge through the pipe equals the discharge into the crane ram cylinder, and thus

$$v \times \frac{\pi}{4}(0.05)^2 = \frac{V}{5} \times \frac{\pi}{4} \times (0.3)^2$$

or $$V = \text{speed of lifting} = \frac{5}{36}\,v$$

or $$V = \frac{5}{36}(81.83 - 1.18W)^{1/2}$$

For $W = 0$; $V = 1.256$ m/s

For $W = 50$ kN; $V = 0.664$ m/s

Example 25.14. *A hydraulic lift situated at a distance of 750 m from the accumulator is supplied with water under pressure from the accumulator through a 100 mm main. The accumulator maintains a steady pressure of 6 870 kN/m^2. The lift ram is 200 mm in diameter and the load on it inclusive of its own weight is 120 kN.*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Assume the friction of the ram to be equivalent to an addition of 6 percent of gross load on the ram, determine the speed at which the lift will ascend. Take Darcy's f for the main as 0.03. Neglect losses due to shock at entrance to the lift cylinder.

Solution

Let v be the velocity of flow in the main, then loss of head due to friction h_f is given as

$$h_f = \frac{flv^2}{2gd} = \frac{0.03 \times 750 \times v^2}{2 \times 9.81 \times 0.10}$$

$$= 11.47\ v^2$$

Pressure head of water in the accumulator

$$= \frac{6870 \times 10^3}{9810} = 700.31 \text{ m of water}$$

Load on the lift ram + friction losses

$$= 120\ (1 + 0.06) = 127.2 \text{ kN}$$

Pressure head due to this load

$$= \frac{127.2 \times 10^3}{(\pi/4)(0.20)^2} \times \frac{1}{9810}$$

$$= 412.73 \text{ m of water}$$

Thus we have

$$11.47\ v^2 + 412.73 = 700.31$$

$$\therefore \quad v = 5.01 \text{ m/s}$$

Now if V is the speed at which the lift ascends, then since discharge from main equals discharge into the lift ram cylinder, we have

$$V \times \frac{\pi}{4} \times (0.20)^2 = 5.01 \times \frac{\pi}{4} \times (0.10)^2$$

or

$$V = 1.25 \text{ m/s}$$

Example 25.15. *Certain machinery is worked from a weight loaded accumulator through a pipe 600 m long and 0.1 m in diameter. The accumulator has a ram 0.3 m diameter and 3.5 m stroke, loaded with 320 kN, it is supplied with water by a three throw pump running at 40 r.p.m., the plungers of pump having a diameter of 0.05 m and a stroke of 0.4 m. The slip of the pump has been estimated as 4% and Darcy's friction factor f for the pipe is 0.03. If the machinery absorbs 37 kW, calculate the longest period during which it may be operated continuously.*

Solution

Pressure head in accumulator

$$= \frac{320 \times 10^3}{(\pi/4)(0.3)^2 \times 9810}$$

$$= 461.5 \text{ m of water}$$

Head loss due to friction in the pipe

$$= \frac{flv^2}{2gd}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{0.03 \times 600 \times v^2}{2 \times 9.81 \times 0.1} = 9.17\, v^2$$

$\therefore$ Effective head at machine is

$$H = (461.5 - 9.17\, v^2)$$

Thus power supplied at machine is

$$wQH = 9\,810\,(\pi/4) \times (0.1)^2\, v\ (461.5 - 9.17 v^2) = 37 \times 10^3$$

On simplification, we get

$$v^3 - 50.33\, v + 52.37 = 0$$

Solving by trial, we get

$$v = 1.064 \text{ m/s}$$

i.e., the velocity of flow in pipe is 1.064 m/s

$\therefore$ Discharge $Q = av = \frac{\pi}{4} \times (0.1)^2 \times 1.064 = 8.36 \times 10^{-3}\ \text{m}^3/\text{s}$

Now discharge Q is the same as the discharge leaving the accumulator, but the discharge entering the accumulator is equal to the discharge of the three throw pump.

$$\text{Discharge of pump} = \left[\frac{40}{60}(1 - 0.04)(\pi/4)(0.05)^2 \times 0.4\right] \times 3$$

$$= 1.51 \times 10^{-3}\ \text{m}^3/\text{s}$$

$\therefore$ Water supplied from the accumulator

$$= (8.36 - 1.51)\ 10^{-3} = 6.85 \times 10^{-3}\ \text{m}^3/\text{s}$$

Volume of the accumulator

$$= \frac{\pi}{4}(0.3)^2 \times 3.5 = 0.2474\ \text{m}^3$$

$\therefore$ Time during which machinery may be operated continuously is same as the time in which the accumulator will be emptied and it is given as

$$t = \frac{0.2474}{6.85 \times 10^{-3}} = 36.12 \text{ seconds}$$

Example 25.16. *A 600 kN hydraulic press having a ram 0.2 m diameter is operated from an overhead tank fed from the city water main through an intensifier having low pressure piston diameter 1 m and high pressure ram diameter 0.1 m. The static head on the low pressure side of the intensifier is 21 m, and the water is supplied from the overhead tank through a pipe 50 mm diameter and 100 m long. The friction of the gland packing of the press is 2.5% of the net load exerted. Determine the rate of movement of the press ram when exerting its maximum force. Take Darcy's f for 50 mm pipe as 0.02.*

Solution

Let v be the velocity of flow in 50 mm pipe, then loss of head due to friction

$$h_f = \frac{flv^2}{2gd} = \frac{0.02 \times 100 \times v^2}{2 \times 9.81 \times 0.05} = 2.04\, v^2$$

Net pressure head available on the low pressure side

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= (21 - 2.04\, v^2)$$

Pressure head developed by the intensifier

$$= (21 - 2.04\, v^2) \times (1/0.1)^2$$

Since 2.5% is lost in friction, the net pressure intensity utilised by the press

$$= \left[(21 - 2.04\, v^2) \times 100 \times \frac{97.5}{100}\right] \times 9\,810$$

Pressure intensity developed at the press ram is also equal to

$$\frac{600 \times 10^3}{(\pi/4) \times (0.2)^2} = \left(\frac{6 \times 10^7}{\pi}\right)$$

Thus equating the two, we get

$$[(21 - 2.04\, v^2) \times 97.5 \times 9\,810] = \left(\frac{6 \times 10^7}{\pi}\right)$$

or $(21 - 2.04\, v^2) = 19.97$

or $v = 0.71$ m/s

Discharge in the pipe $= \frac{\pi}{4}(0.05)^2 \times 0.71 = 1.39 \times 10^{-3}$ m^3/s

$\therefore$ Discharge of high pressure water into the press cylinder

$$= (1.39 \times 10^{-3}) \left(\frac{0.1}{1}\right)^2 = 1.39 \times 10^{-5} \text{ m}^3/\text{s}$$

Now if V is the rate of movement of the press ram then

$$V \times \frac{\pi}{4}(0.2)^2 = 1.39 \times 10^{-5}$$

$$V = 4.42 \times 10^{-4} \text{ m/s} = 2.652 \times 10^{-2} \text{ m/min.}$$

Example 25.17. *Water at a pressure of 140 kN/m^2 and at a rate of 2.4 litres per second enters a cylinder of an intensifier. Assuming 4% of superimposed load lost in friction in each packing and ratio of diameters of sliding and fixed ram to be 10, calculate the rate in litres per minute and pressure in metres of water delivered by the intensifier.*

If this intensifier supplies water to a press consisting of a ram 0.1 m^2 in area, 12 kN in weight and pressing aluminium sheets while advancing upwards; calculate the maximum work in N.m it is capable of doing, assuming that the sliding ram of the press moves through its full stroke in 3 minutes. Neglect the packing friction of the press ram.

Solution

Quantity of water delivered by the intensifier is given as

$$2.4 \times \left(\frac{1}{10}\right)^2 \times 60 = 1.44 \text{ litres/minute}$$

Also the pressure head developed is given as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\left[\frac{140\times10^3}{9810}\times\frac{100-(2\times4)}{100}\times\left(\frac{10}{1}\right)^2\right]=1313\text{ m}$$

since 4% is the friction loss in each of the two packings.

Now, the net force causing the motion of the press ram is obtained as

$[(1313 \times 9\,810 \times 0.1) - 12 \times 10^3] = 1\,276\,053$ N

Therefore, the maximum work that can be done by the press ram

$$= 1\,276\,053 \times \left(\frac{1.44\times10^{-3}}{60}\times\frac{1}{0.1}\right)\times(3\times60)$$

$$= 5.513 \times 10^4 \text{ N-m}$$

Example 25.18. *A hydram uses 4 m^3/minute of water under a head of 3 m and pumps 0.25 m^3/minute to a tank 24 m high above the ram through 150 m long and 60 mm diameter pipe. Take Darcy's f for the pipe as 0.04 and calculate the efficiency of the ram.*

Solution

Weight of water used

$$= (4 \times 9\,810) = 39\,240 \text{ N/min}$$

Weight of water pumped

$$= (0.25 \times 9\,810)\ 2\,452.5 \text{ N/min}$$

Velocity of flow in the delivery pipe

$$= \frac{0.25}{(\pi/4)(0.06)^2}$$

$$= 88.42 \text{ m/min} = 1.47 \text{ m/s}$$

Head lost due to friction in the delivery pipe is

$$h_f = \frac{flv^2}{2gd} = \frac{0.04\times150\times(1.47)^2}{2\times9.81\times0.06} = 11.01 \text{ m}$$

$\therefore$ Effective head developed by the ram

$$= (H + h_f) = (24 + 11.01) = 35.01 \text{ m}$$

Supply head $h = 3$ m

$\therefore$ Rankine's efficiency of the ram

$$\eta_r = \frac{qH}{Qh}$$

$$= \frac{0.25\times(35.01-3)}{(4-0.25)\times3}\times100 = 71.13\%$$

D'Aubuisson's efficiency of the ram

$$\eta_a = \frac{q(H+h)}{(Q+q)h}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{0.25 \times 35.01}{4 \times 3} \times 100 = 72.94\%$$

Example 25.19. *The following particulars relate to a hydram installation:*

Supply pipe 75 mm diameter and 9 m long, supply head 1.5 m above waste valve, waste valve 100 mm diameter, 6 mm travel and weight 16 N, delivery head 7.5 m above waste valve. Neglecting slip and friction through the delivery pipe, calculate the discharge in litres per minute pumped into the delivery tank. Also find the number of beats per minute.

Solution

Upward pressure head needed to close the waste valve is

$$h_0 = \frac{16}{(\pi/4)(0.1)^2} \times \frac{1}{9810} = 0.208 \text{ m of water}$$

Maximum velocity past waste valve just before closure

$$= \sqrt{2gh_0} = \sqrt{2 \times 9.81 \times 0.208} = 2.02 \text{ m/s}$$

Let V_m be the maximum velocity in supply pipe then

$$\pi\,(0.1)\,(0.006)\,(2.02) = \frac{\pi}{4} \times (0.075)^2 \times V_m$$

$$\therefore \quad V_m = 0.862 \text{ m/s}$$

The velocity of flow in the supply pipe increases gradually from 0 to 0.862 m/s and if it takes place in t_1 seconds, then

$$1.5 = \frac{l_s}{g}\frac{V_m}{t_1} = \frac{9}{9.81} \times \frac{0.862}{t_1}$$

or $$t_1 = 0.527 \text{ s}$$

Also let t_2 be the time during one beat for which the waste valve remains closed and the delivery valve is open, then

$$(7.5 - 1.5) = \frac{l_s}{g}\frac{V_m}{t_2} = \frac{9}{9.81} \times \frac{0.862}{t_2}$$

or $$t_2 = 0.132 \text{ s}$$

$\therefore$ Total time for one complete beat

$$t = (0.527 + 0.132) = 0.659 \text{ s}$$

$\therefore$ Number of beats per minute

$$= \frac{60}{0.659} = 91.05 \approx 91$$

Discharge into delivery tank is given as

$$q = \frac{\pi}{4} d^2 \times \frac{V_m}{2} \times \frac{t_2}{t}$$

$$= \left[\frac{\pi}{4} \times (0.075)^2 \times \frac{0.862}{2} \times \frac{0.132}{0.659}\right] = 0.381 \times 10^{-3} \text{ m}^3/\text{s}$$

$$= 22.86 \text{ litres/minute}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Alternatively. At the instant of closure of the waste valve the kinetic energy of water column in the supply pipe is

$$\left[(wl_s)\times\left(\frac{\pi d^2}{4}\times\frac{V_m^2}{2g}\right)\right] = 9\,810\times 9\times\frac{\pi}{4}(0.075)^2\times\frac{(0.862)^2}{2\times(9.81)}$$

$$= 14.772 \text{ N-m per beat}$$

The weight of water that can be lifted through a height (7.5 – 1.5) = 6 m is (14.772/6) N, and number of beats per minute is 91.05, hence the discharge pumped into the delivery tank

$$= \frac{14.772\times 91.05}{6\times 9810\times 60} = 0.381\times 10^{-3}\ \text{m}^3/\text{s}$$

$$= 22.86 \text{ litres/minute.}$$

SUMMARY OF MAIN POINTS

1. Several hydraulic machines are employed for either storing the hydraulic energy and then transmitting it when required or magnifying the pressure energy several times and then transmitting the same. These machines are based on the principles of hydrostatics and hydrokinetics in which the hydraulic energy is transmitted through the liquid medium for which water or oil is used.
2. Hydraulic accumulater is a device which is used for storing the energy in the form of pressure energy and supplying it when required. The capacity of the hydraulic accumulator is given as

$$p\times\frac{\pi D^2}{4}\times L = p\times\text{volume}$$

where

p = pressure of liquid supplied by the pump;
D = diameter of the sliding ram; and
L = stroke or lift of the ram.

3. Differential accumulator is a device in which liquid can be stored at a high pressure by a comparatively small load on the ram.
4. Hydraulic intensifier is a device in which the intensity of pressure of liquid is increased by utilizing the energy of a large quantity of liquid at a low pressure. The increased pressure intensity p_2 is given by

$$p_2 = p_1\left(\frac{A_1}{A_2}\right)$$

where

p_2 = low pressure intensity of liquid;
A_1 = external area of sliding ram or cylinder; and
A_2 = area of the end of the fixed ram.

5. Hydraulic press is a device in which by the application of a small pressure (or force) a large pressure (or force) may be developed.
6. Hydraulic crane is a device which is used for lifting heavy loads by utilizing the pressure energy of liquid.
7. Hydraulic lift is a device which is used for carrying the goods as well as persons from one floor to the other in a multi-storeyed building by utilizing the pressure energy of liquid.
8. Hydraulic ram is a type of pump in which the energy of large quantity of water falling through small height is utilized to lift a small quantity of this water to greater height.

The efficiency of a hydraulic ram may be assessed in the two ways as

(i) Rankines efficiency $\eta_r = \dfrac{qH}{Qh}$

(ii) D´ Aubuisson's efficiency $\eta_d = \dfrac{(H+h)q}{h(Q+q)}$

where

q = quantity of water lifted per second through height H; and
Q = quantity of water falling through height h.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

9. Hydraulic coupling is a device in which power is transmitted from a driving shaft to driven shaft without any change of torque.
10. Torque converter is a device in which the arrangement is provided for getting increased or decreased torque at the driven shaft.
11. Air lift pump is a device which is used for lifting water from a well by utilizing the compressed air.

PROBLEMS

25.1 Describe with the help of a neat sketch the working of a hydraulic accumulator.

25.2 Explain with sketch the working of a hydraulic ram and define the efficiency of the ram.

25.3 Explain with neat sketches the principle of working of the following devices (i) Hydraulic ram; (ii) Hydraulic intensifier, (iii) Hydraulic accumulator.

25.4 An accumulator has a ram 0.3 m in diameter and a lift of 7.5 m. Water is supplied at a pressure of 5900 kN/m^2. Find the necessary load on the ram and the capacity of the accumulator in kilowatt hours. [*Ans.* 417.05 kN; 0.869 kWh]

25.5 An accumulator has a ram 0.3 m diameter and 9 m lift, and is loaded with 1000 kN total weight. If packing friction accounts for 2% of the total force on the ram, determine the power being delivered to the machine if the ram falls through its full range in 3 minutes, and if at the same time the pumps are delivering 1 360 litres per minute. [*Ans.* 363.25 kW]

25.6 A power of 30 kW is required to be supplied to the ram of a press from an accumulator through a 150 mm diameter pipe 1000 m long. If the loss in transmission is to be 2.5 percent, find the area of the press ram which is loaded with 1000 kN. Take Darcy's f for the pipe as 0.04. [*Ans.* 0.396 m^2]

25.7 A hydraulic crane fitted with a ram and pulley sheave lifts 60 kN of load at 1.5 metres per second through 75 metres every 240 seconds. If the diameter of the ram piston = 0.75 m, fluid pressure inside ram = 3900 kN/m^2 and pulley efficiency = 80 percent, find the velocity ratio of the pulley sheave. If an accumulator interposed between the ram and a continuously operating pump supplies the excess fluid required by the ram in addition to the supply from the pump, find the minimum capacity of the accumulator. [*Ans.* 23; 4.45 $\times 10^3$ kN-m]

25.8 A hydraulic lift raises a load of 60 kN through a height of 10 m once every 2 min, the speed of lifting being 0.5 m per second. It is worked from an accumulator which is being constantly charged by a pump. The pressure of water is 2950 kN/m^2, the efficiency of the lift 80 percent and the efficiency of the pump 85 percent. Find the power required to drive the pump and the minimum volume of the accumulator. Frictional losses in the pipes may be neglected. [*Ans.* 7.36 kW; 0.212 m^3]

25.9 In an installation of 6 hydraulic cranes, the working cycle of each of which takes 90 seconds in hoisting and lowering, each crane is fed with water at a pressure of 5 400 kN/m^2 and is required to lift a load of 50 kN at a speed of 0.3 m/s through a total height of 9 m, the jigger system giving a velocity ratio of 6. Estimate the stroke and diameter of the ram assuming an efficiency of 60%. [*Ans.* 1.5 m; 343 mm]

25.10 A hydraulic intensifier, supplying water to a hydraulic press gets water from supply main at a pressure of 4 900 kN/m^2. It has to intensify the pressure to 19 600 kN/m^2. If the stroke of the intensifier is 1 m and the cubic capacity is 20 litres, find the diameters of two rams of the intensifier. [*Ans.* 160 mm; 320 mm]

25.11 A hydraulic lift situated at a distance of 1 000 m from the accumulator is supplied water by it through a 150 mm main under a steady pressure of 7850 kN/m^2. The lift ram is 200 mm in diameter and the load on it inclusive of its own weight is 150 kN. Assuming the friction of the ram to be equivalent to an additional 6 percent of gross load on the ram, determine the speed at which the lift will ascend. Take Darcy's f for the main as 0.03. [*Ans.* 2.97 m/s]

25.12 In a hydraulic ram installation the supply pipe is 60 mm diameter and 4 m long; the waste valve is 150 mm diameter, has an effective lift of 6 mm and weight 15 N; and it makes 120 beats per minute. Estimate the discharge

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

through the delivery pipe against a delivery head (above water level in supply tank) of 10 metres. Neglect losses.
[*Ans.* 11.7 litres/minute]

25.13 A 600 kN hydraulic press having a ram of 0.2 m diameter is operated from an overhead from the supply mains through an intensifier having $(D_1/D_2) = 10$. The static head on the low pressure side of the intensifier is 20.5 m and the water is supplied from the overhead tank through a pipe 60 mm diameter and 100 m long. The friction of the gland packing of the press is 2% of the net exerted force. Determine the rate of movement of the press ram when exerting its maximum force. Take f for 60 mm pipe as 0.02. [*Ans.* 33 mm/min]

25.14 In a hydraulic ram, supply pipe is 75 mm diameter, waste valve is 100 mm diameter, 5 mm lift and 20 N weight. Calculate the maximum velocity that can occur in supply pipe.
[*Ans.* 0.802 m/s]

25.15 Find the area of cross-section of the ram of an accumulator loaded with 1000 kN, which is required to transmit 30 kW to a machine through a 0.1 m diameter pipeline 1200 m long. Darcy's friction coefficient is 0.01 and the loss of head due to friction in pipe should be 2 percent of the head at the accumulator.
[*Ans.* 0.28 m^2]

25.16 Water at 490 kN/m^2 and at a rate of 0.5 m^3/s enters an intensifier, having ratio of diameter of sliding and fixed ram as 10. Calculate the pressure and discharge of water delivered by the intensifier.
[*Ans.* 49 000 kN/m^2; 5 litres/second]

25.17 A hydraulic press has to work once, for 30 seconds in every 3 minutes. It is to be supplied by means of a pump discharging 0.1 m^3/s under 980 kN/m^2 pressure. Calculate the power supplied to the press if a suitable hydraulic accumulator has been introduced between the press and the pump [*Ans.* 588 kW]

25.18 A load of 4 000 kN is applied in a compression testing machine by means of four hydraulic jacks each with a ram of 0.2 m diameter. The frictional resistance on each ram is equivalent to a load of 25 kN. The jacks are supplied with water under pressure from an intensifier. Find the pressure of water supplied to the jacks and the diameter ratios of intensifier rams, if the inlet pressure to intensifier is 3 900 kPa.
[*Ans.* 3.263×10^4 kPa; 2.89]

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Elements of Hydrology

Chapter 26

26.1 DEFINITION

Hydrology may be defined as the science that deals with the processes governing the depletion and replenishment of the water resources of the land areas of the earth. In its broadest sense, it is the study of water in all its phases and includes hydraulics, the physics and chemistry of water, meteorology and various other allied natural sciences.

26.2 THE HYDROLOGIC CYCLE

The hydrologic cycle is the descriptive term applied to the general circulation of water from the oceans to the atmosphere, to the ground and back to the oceans again. Figure 26.1 shows the various phases

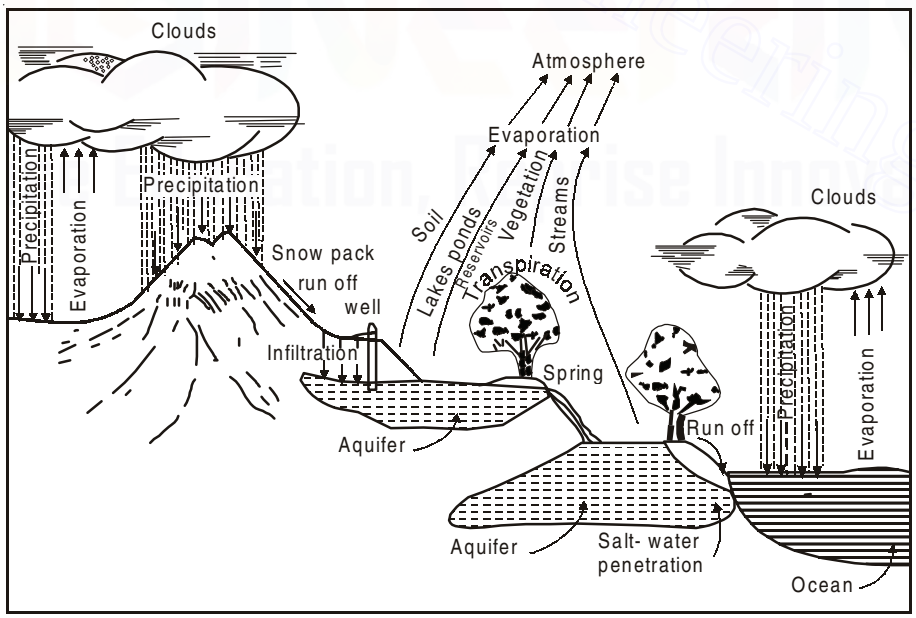


Figure 26.1 *The Hydrologic cycle*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

of hydrologic cycle. The cycle may be considered to begin with the water of the oceans. Water from the ocean surface is evaporated into atmosphere. The vapour is condensed by various processes and falls to the earth as precipitation. Some of this precipitation falls directly on the oceans, and some falls on land surfaces. A portion of that falling on the land is retained temporarily in the soil, in surface depressions and on vegetation and other objects until it is returned to the atmosphere by evaporation and transpiration. Another portion runs off from the ground surface into the streams and is returned to the ocean. Still another portion percolates into the ground and joins the ground water which also slowly finds its way to the streams as ground water flow. However, some portion of the ground water which percolates the great depths appears after long intervals as springs, artesian wells and geysers. Further the entire quantity of water that reaches the stream does not flow directly to the oceans, because throughout its travel a portion of it returns to the atmosphere by evaporation and transpiration and some portion seeps into the ground. The hydrologic cycle provides the ground work upon which the science of hydrology is based.

26.3 PRECIPITATION

Precipitation is one of the main phases of the hydrologic cycle. It includes all moisture that reaches the earth's surface in liquid or solid form due to the condensation of the atmospheric vapour. The principal forms of the precipitation are rain, snow, sleet (i.e., snow mixed with rain), hail (i.e., frozen rain drops), mist, dew and frost. Out of the various forms of precipitation, a major part of precipitation occurs in the form of rain and a minor part occurs in the form of snow. The contribution of the other forms of precipitation is very little and hence the same is generally ignored.

For almost all the hydrological designs it is necessary to have the precipitation records for a long period. Such records may be developed by the measurement of the precipitation occurring at any place from time to time. All the forms of precipitation are measured on the basis of the vertical depth of water or water equivalent which would accumulate on a level surface if all the precipitation remained where it fell. Many types of gages have been developed for the measurement of rain and snow which constitute the major part of the precipitation. Some of these gages which are commonly used are described in the following section. Since the amount of rainfall varies from place to place, it would be necessary to install the rain gages at various key points in the area. Especially in the drainage basin of a stream a network of rain gage stations evenly distributed over the entire area of the drainage basin should invariably be provided.

26.4 MEASUREMENT OF RAINFALL AND SNOWFALL

26.4.1 Measurement of Rainfall

For the measurement of rainfall rain gages are used which may be classified as follows:

(i) Non-recording rain gages.

(ii) Recording rain gages.

26.4.1.1 Non-recording rain gages

As the name indicates these rain gages do not record the rainfall directly but only collect the rain water which when measured gives the total amount of rainfall at the rain gage station during the measuring interval. The India Meteorological Department has adopted the standard Symon's rain gage for measuring the rainfall. It is a non-recording rain gage which is used at most of the rain gage stations in India. As shown in Fig. 26.2, the Symon's rain gage consists of a cylindrical metal case of internal

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

diameter 127 mm with its base enlarged to 203.2 mm diameter. At the top of the case a funnel is fixed which is provided with a brass rim measuring exactly 127 mm inside diameter. The funnel shank is inserted in a glass bottle placed inside the case. The case of the rain gage is fixed in masonry or concrete foundation such that the funnel rim is exactly 305 mm above the ground level. The rain water enters the bottle through the funnel and gets collected in the bottle. The rain water collected in the bottle is measured with the help of a standard measuring glass supplied with each rain gage which indicates the millimeters of rain that has fallen at the rain gage station. At each gage station the observations for the rainfall are taken daily two times at 8.30 a.m. and 5.30 p.m. (IST). However, if the rainfall is likely to exceed the capacity of the bottle then a few intermediate observations are also taken. The sum of the observations taken will represent the total rainfall of the past 24 hours of the day on which the observation at 8.30 a.m. is taken.

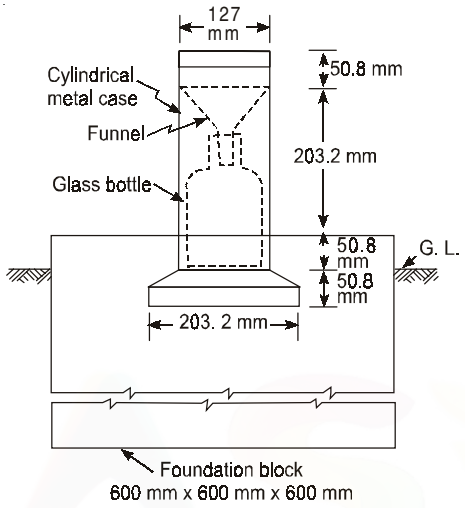


Figure 26.2 *Symon's rain gage*

26.4.1.2 Recording rain gages

These rain gages automatically record the intensity of rainfall and the time of its occurrence in the form of a pen trace on a clock driven chart, from which the total amount of rainfall for the desired duration may also be determined. These gages may be operated over extended periods without attention and the recorded observations may be collected from the gages each time after a certain fixed duration. The most widely used recording rain gages are as given below:

(a) Tipping bucket rain gage.
(b) Weighing type rain gage.
(c) Float type rain gage.

26.4.1.3 Tipping bucket rain gage

It consists of a 300 mm diameter funnel which collects rain water and conducts it to a pair of small buckets pivoted just below the funnel as shown in Fig. 26.3. The buckets are so designed that, when 0.25 mm of rainfall collects in one bucket, it tips and empties its water into the storage tank below and at the same time the other bucket is brought under the funnel. The tipping of the bucket actuates an electric circuit which causes a pen to make a mark on a record sheet mounted on a clock driven revolving drum. Since each mark on the record sheet corresponds to 0.25 mm of rainfall, by counting the same the intensity of rainfall may be determined. The total rainfall as determined from the record at the end of the day may also be checked by measuring the rain water collected in the storage tank in the same manner as in the case of a non-recording gage.

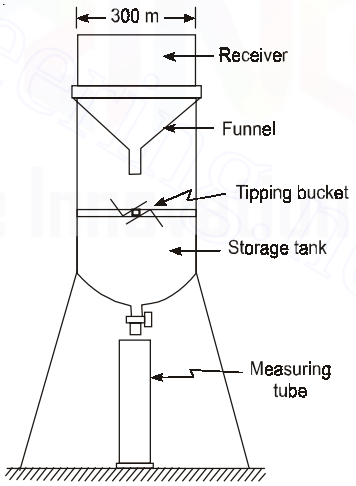


Figure 26.3 *Tipping bucket rain gage*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

26.4.1.4 Weighing type rain gage

In this rain gage the rain water passes through a funnel into a bucket, which is supported on the weighing platform of a spring or lever balance as shown in Fig. 26.4. The increase in the weight of the bucket due to the addition of the rain water causes the platform to move. The movement of the platform is transmitted through a system of links and levers to a pen which makes trace of accumulated amounts of rainfall on a suitably graduated chart wrapped round a clock driven revolving drum. The mechanism is arranged to reverse the travel of the pen after a certain amount of rainfall (say 150 mm) has accumulated and reverse again after another equal amount, so that the gage may operate unattended for a week at a time, except in regions of very intense rainfall, where the totals may exceed the capacity of the gage (usually 300 mm). The rainfall record produced by this gage is in the form of a mass curve of rainfall in which as shown in Fig. 26.5 the total rainfall is plotted with respect to time. The slope of the curve gives the intensity of the rainfall.

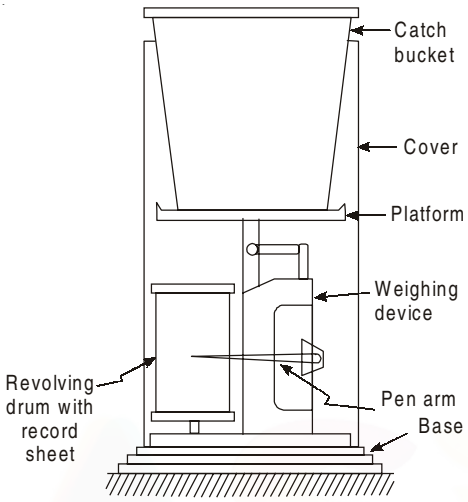


Figure 26.4 *Weighing type rain gage*

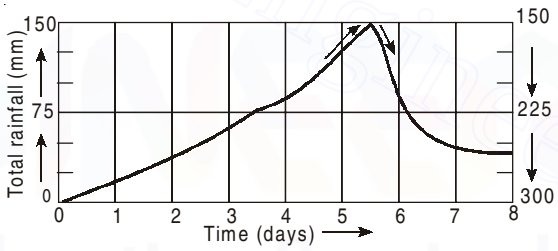


Figure 26.5 *Rainfall record sheet of a weighing type rain gage*

26.4.1.5 Float type rain gage

As shown in Fig. 26.6 in this rain gage the rain water enters a chamber which contains a float. As the level of the rain water collected in the float chamber rises, the float moves up, which actuates a pen connected to it through a connecting rod. The pen makes a trace of accumulated amounts of rainfall on a suitably graduated chart wrapped round a clock driven revolving drum. Thus in the case of this gage also the rainfall record is in the form of a mass curve of rainfall, which is same as in the case of weighing type rain gage. When the float chamber gets completely filled, it is automatically emptied by means of a siphon as shown in Fig. 26.6.

In addition to the above noted gages, in the modern times the use of radar as an aid in the measurement of rainfall is being made. However, the main use of radar is in the determination of the areal extent, orientation and movement of rain storms.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

26.4.2 Measurement of Snowfall

For the measurement of snowfall, storage gages are used. In principle these gages are same as the non-recording and recording gages described above, but they are designed to hold much greater amounts of precipitation. Recording gages designed to operate for 30 to 60 days without servicing and non-recording gages with sufficient capacity for an entire season are now in use. A non-recording snowfall gage used by US. Weather Bureau is made from 300 mm thin-walled pipe. The gage is made in 1.5 m long sections so that any height in multiples of 1.5 m may be obtained. To permit free fall of snow inside the gage, the upper portion, of the gage is in the shape of a frustrum of a cone. The equivalent depth of water contained in any snowfall may be found by melting the snow collected in the gage and measuring the quantity of water formed. Further to provide gages with capacity for large amounts of snowfall without making the container excessively large, arrangements are made to melt the snow in the gage itself as it gets collected there. This is usually accomplished by placing in the gage certain chemicals such as calcium chloride, ethylene glycol etc.

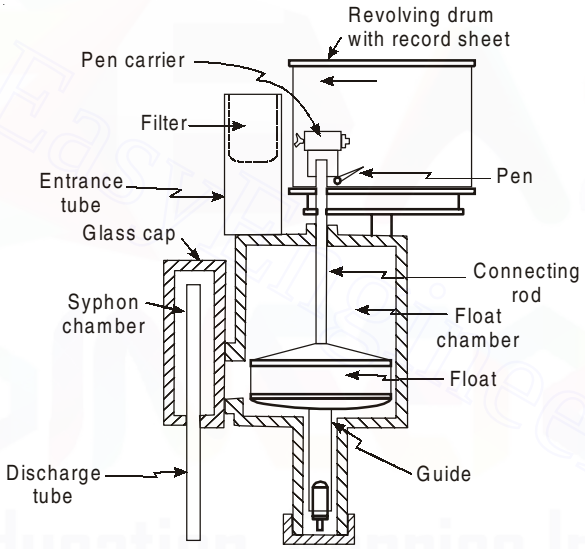


Figure 26.6 *Float type rain gage*

26.5 MEAN DEPTH OF RAINFALL OVER AN AREA

For most of the hydrologic problems, it is necessary to determine the mean or average depth of rainfall over a particular area. The following three methods are commonly used for computing the mean depth of rainfall over an area:

(i) Arithmetic mean method
(ii) Theissen polygon method
(iii) Isohyetal method.

26.5.1 Arithmetic Mean Method

In this method the average depth of rainfall over an area is obtained by dividing the sum of the depths of rainfall recorded at all the rain gage stations located in the area by the number of stations. Thus if

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

P_1, P_2, P_3..P_n are the depths of rainfall recorded at the n rain gage stations distributed over an area then the average depth of rainfall P is given as

$$P = \frac{P_1 + P_2 + P_3 + \ldots + P_n}{n} = \frac{\sum_{i=1}^{i=n} P_i}{n} \quad \ldots(26.1)$$

The results obtained by this method will be correct only if the rain gage stations are uniformly distributed over the area and the rainfall varies in a regular manner. This is so because in this method every rain gage station has equal weightage regardless of its location.

26.5.2 Theissen Polygon Method

In this method the adjacent rain gage stations are joined by straight lines thus dividing the entire area into a series of triangles as shown in Fig. 26.7. On each of these lines perpendicular bisectors are erected, thereby forming a series of polygons, each containing only one rain gage station. The entire area within any polygon is nearer to the rain gage station contained therein than to any other station, and hence it is assumed that the rainfall recorded at that station is the representative rainfall for that area. If P_1, P_2, P_3..P_n represent the depths of rainfall recorded at the rain gage stations enclosed by polygons, the areas of which are respectively A_1, A_2, A_3.....A_n, then the average depth of rainfall P for the entire area A is given as

$$P = \frac{P_1A_1 + P_2A_2 + P_3A_3 + \ldots + P_nA_n}{A} = \frac{\sum_{i=1}^{i=n} P_iA_i}{A}$$

where $A_1 + A_2 + A_3 + \ldots + A_n = A$. …(26.2)

In the Theissen polygon method each rain gage station is given weightage according to its position with respect to the boundary of the area under consideration, and hence this method is better than the arithmetic mean method.

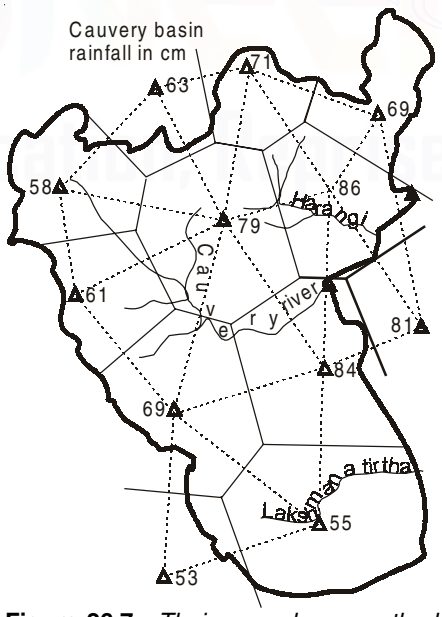


Figure 26.7 *Theissen polygon method*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

26.5.3 Isohyetal Method

Isohyets are the contours of equal rainfall. In the isohyetal method isohyets are drawn on the map of the area under consideration. For drawing the isohyets the depth of rainfall recorded at each of the rain gage stations are noted on the map of the area, at the respective stations. Then assuming a linear variation of rainfall between the two stations, the probable positions of rainfall values at some interval are interpolated between the stations. The points with equal values of rainfall are connected by smooth curves which forms the isohyetal pattern for the area as shown in Fig. 26.8. The area between the two adjacent isohyets is measured with the help of planimeter and for this area the rainfall is presumed to be equal to the mean of the two isohyet values. The average depth of rainfall P for the entire area A may be found by the following equation.

$$P = \frac{\Sigma(\text{Area between two adjacent isohyets} \times \text{mean of the two isohyet values})}{A}$$

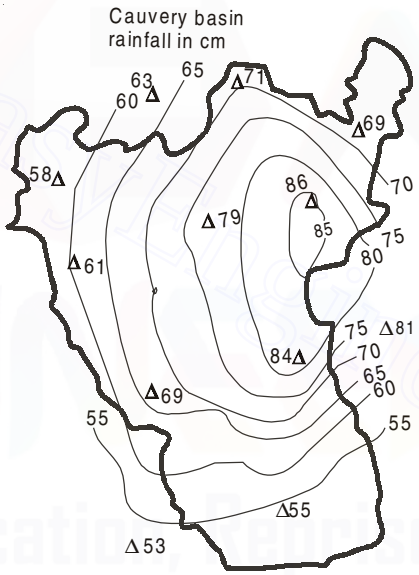


Figure 26.8 *Isohyetal method*

The isohyetal method is more accurate method for computing the average depth of rainfall. This is so because the isohyetal method takes into account the actual spatial relationship of the rain gage stations.

26.6 EVAPORATION, TRANSPIRATION AND EVAPO-TRANSPIRATION

Evaporation is the process by which water is converted into gaseous state and is returned to the atmosphere as vapour. The water is continuously evaporating from the water surfaces of natural and artificial reservoirs, lakes etc., snow and soil surfaces and also from the surfaces of vegetation. As such a part of the precipitation reaching the earth's surface is lost through evaporation. Evaporation depends upon several factors such as temperature, wind velocity, humidity, salinity of water and surface area. The rate of evaporation increases with the increase of temperature as well as with the increase of wind

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

velocity, but it decreases with the increase of humidity as well as with the increase of dissolved salts in water. Further the evaporation increases with the increase in the open surface area of water and soil.

In order to determine the quantity of water lost through evaporation from the water surfaces of natural and artifical reservoirs, lakes etc., several methods have been developed, but those commonly adopted are (i) the use of empirical formulae, and (ii) the direct measurement with the help of evaporation pans.

A large number of empirical formulae have been developed for determining the quantity of water lost through evaporation but the one which is widely used is the Lake Hefner formula which is

$$E = 6.9317\,(e_s - e_2)\,V_4$$

and

$$E = 5.5046\,(e_s - e_8)\,V_8 \qquad ...(26.4)$$

where E is evaporation in mm/day; e_s is vapour pressure at water surface in N/mm^2; e_2 and e_8 are vapour pressures in N/mm^2 at heights of 2 m and 8 m respectively above the water surface; and V_4 and V_8 are wind velocities in km/hour at heights of 4 m and 8 m respectively above the water surface.

For the direct measurement of the quantity of water lost through evaporation various types of evaporation pans have been designed, but the most commonly employed pans are (a) *USWB* Class *A* Land Pan, (b) Colorado Sunken Pan, (c) Floating Pan. The evaporation pan is installed in the vicinity of the reservoir or lake for which the loss of water through evaporation is to be determined. The quantity of water lost through evaporation from the pan is measured. Since the evaporation from pan is greater than that from the adjoining reservoir or lake, the observed values of the water lost through evaporation from the pan are multiplied by a predetermined coefficient of the pan to obtain the actual quantity of water lost through evaporation from the adjoining reservoir or lake.

Transpiration is the process by which the moisture absorbed by the roots of the plants and circulated through the plant body, returns back to the atmosphere as water vapour breathed out by the stomata of the leaves of the plants. Since transpiration is essentially the evaporation of water from leaf cells, the rate of transpiration is also influenced by all the factors which affect the evaporation. Thus high temperature, low humidity and strong wind all tend to increase transpiration. But in addition to these factors transpiration is also affected by a few other factors such as the available amount of sun-shine, available soil moisture, type and extent of plant growth etc.

It is not possible to measure the quantity of water lost through transpiration from a large area under natural conditions, and hence the measurement of the transpiration losses is restricted to studies of small samples under laboratory conditions. One such practical method for measuring the transpiration loss is with the help of instruments called potometers and phytometers. A *potometer* is a small vessel containing water and sealed so that the only escape of moisture is by transpiration from a small plant with its cut end inserted in the water. A *phytometer* is a large vessel filled with soil in which one or more plants are rooted. The soil surface is sealed to prevent evaporation so that the only escape of moisture is by transpiration. Thus in each of the above devices the loss in the weight of plant and vessel is a measure of the transpiration loss for the specimen tested.

The land surfaces which are partially covered with vegetation, loose moisture simultaneously through evaporation from the land surface and through transpiration from vegetation. As such the common practice is to estimate the combined loss of moisture from the land surface through evaporation as well as through transpiration. This combined loss of moisture from the land surface is known as evapo-transpiration or consumptive use of water. The term *evapo-transpiration* is defined as the total

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

volume of water per unit area used by the vegetal growth of a given area in transpiration and building of plant tissue and that evaporated from adjacent soil or intercepted precipitation on the area in a specified time. In the above definition the term vegetal growth includes the natural vegetation as well as the crops grown on the area. Evidently the rate of evapo-transpiration is influenced by all the factors which affect both evaporation and transpiration. Further in order to determine evapo-transpiration several methods have been developed.

26.7 INFILTRATION

Infiltration is the process by which water enters the surface strata of the soil and moves downwards to join the ground water. This water first replenishes the soil moisture deficiency and the excess water moves downwards to become a part of the ground water. The rate at which the water actually infiltrates through a soil at a given time is known as the *infiltration rate.* Further the maximum rate at which a soil in any given condition is capable of absorbing water is called its *infiltration capacity.* Initially when the soil is dry and the rainfall occurs some rain water is consumed in wetting the soil particles and hence the infiltration rate is high. But as the rain continues, the capacity of the soil to absorb the water progressively decreases. Thus if infiltration rate is plotted with respect to time then a falling curve is obtained which is known as infiltration capacity curve. Horton found that infiltration capacity curves approximate the form

$$f_p = f_c + (f_0 - f_c)\, e^{-kt}$$

where f_p is infiltration capacity in mm/hr at any time t; f_0 is initial infiltration capacity in mm/hr; f_c is final infiltration capacity in mm/hr; t is time in hours from the beginning of rainfall; and k is an empirical constant.

Infiltration capacity depends on several factors such as rainfall characteristics, surface conditions of the soil, soil characteristics, moisture content of soil, climatic condition, man's activities etc.

For determining infiltration capacity of a soil the following two methods are adopted: (i) by infiltrometer, (ii) by analysis of rainfall and runoff data. The first method is commonly used and the same is briefly discussed below.

In general there are two types of infiltrometers (a) Ring type and tube types infiltrometers, and (b) Rain simulators.

Ring type infiltrometer consists of two shallow concentric rings of sheet metal, usually ranging from 225 mm to 900 mm in diameter. The rings are placed with their lower edges a few centimetres below the ground surface and with the upper portion projecting above the ground surface. Water is then applied in the inner ring as well as in the annular space between the two rings and is always kept at the same level in both. The function of the outer ring is to prevent the water within the inner ring from spreading over a large area after penetrating below the bottom of the ring. The infiltration capacity of the soil is determined directly as the rate at which water must be added to maintain a constant level.

Tube type infiltrometer consists of a single tube about 225 mm in diameter and 450 to 600 mm long, which is placed in the ground upto a depth at least equal to that to which the water penetrates during the experiment. Thus in this case no spreading of water can occur. Again the rate at which water must be added to maintain a constant depth within the tube is then observed which directly gives the infiltration capacity of the soil.

In the case of rain simulators, on a small area of land ranging from 0.1 m^2 to 50 m^2, water is applied by sprinkling at a uniform rate that is in excess of infiltration capacity. The resulting surface runoff is measured and by subtracting it from the rate of application of water, the infiltration capacity of the soil is determined.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

As explained in section 26.10 the infiltration concept can be applied to the rational computation of surface runoff. For this purpose certain infiltration indices have been devised as given below.

ϕ-index: ϕ-index may be defined as the average rainfall intensity above which the volume of rainfall equals the volume of runoff (Fig. 26.9).

Since the difference between the rainfall and runoff volumes is equal to the sum of the volume of water intercepted, depression storage and that infiltrates into the soil, the ϕ-index represents the combined effect of interception, depression storage and infiltration, all of which are combinedly designated as basin recharge. Further the ϕ-index is based on the assumption that, for a specified storm with given initial conditions, the rate of basin recharge remains constant throughout the storm period.

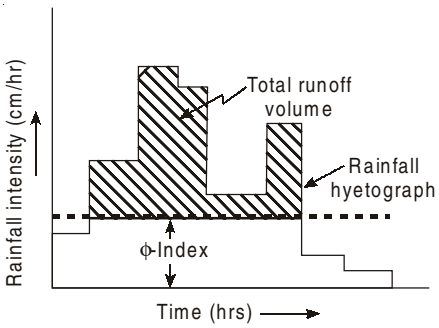


Figure 26.9 *Diagrammatic representation of ϕ-index*

W-index. *W*-index is defined as the average rate of infiltration during the time rainfall intensity exceeds the infiltration capacity. The value of *W*-index may thus be obtained from the following equation

$$W = \frac{F_i}{t_f} = \frac{1}{t_f}(P - Q_s - S_e) \qquad ...(26.6)$$

where F_i is the total infiltration, t_f is the time during which rainfall intensity exceeds infiltration capacity, P is the total precipitation, Q_s is the total surface runoff, and S_e is the retention by interception and depression storage. Thus *W*-index is equivalent to ϕ-index minus the retention by interception and depression storage.

W_{min}*-index*. With very wet conditions, when the infiltration capacity is essentially equal to the final infiltration capacity f_c and the retention rate is at a minimum, the values of *W* and ϕ-indices are almost identical. Under these conditions, *W*-index becomes the W_{min}- index by definition.

26.8 RUNOFF AND FACTORS AFFECTING RUNOFF

Runoff is that part of preciptation as well as any other flow contribution, which is transmitted through natural surface channels or streams or rivers. In the general sense runoff includes (i) surface runoff or overland flow received in the stream immediately after a heavy rain; (ii) inter-flow which is a portion of soil moisture that flows laterally through the upper soil layers and joins the stream before joining the ground water; (iii) delayed runoff or ground water flow that enters the stream after passing through deeper portions of the earth; and (iv) other delayed runoff that has been temporarily detained as snow cover or stored in natural lakes and swamps. Thus runoff is the total quantity of water received by a stream from its drainage basin or catchment. The runoff is generally classified as direct runoff and base flow (or base runoff). The direct runoff comprises the overland flow and the inter-flow which are usually grouped together, while the ground water flow that enters a stream is termed as base flow (or base runoff).

The runoff is generally considered in terms of the total flow carried by a stream during a month, season or year and accordingly it is termed as monthly, seasonal or annual runoff. The runoff is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

expressed either in millimetres or centimetres of water-depth over the entire drainage basin or, in hectare-metre or cubic metre per unit area of the drainage basin. Further the terms stream flow, discharge of a stream and rate of runoff are generally used to mean one and the same thing.

26.8.1 Factors Affecting Runoff

The runoff from the drainage basin or catchment area of a natural stream depends on several factors as described below.

26.8.1.1 Characteristics of precipitation

The pertinent characteristics of precipitation which may considerably affect the runoff are the type of precipitation, its intensity, extent and duration. The precipitation may occur either in the form of rain or snow. If the precipitation occurs in the form of rain, it will immediately produce the runoff, but if the precipitation occurs in the form of snow, it will produce the runoff much later, only when the snow will melt due to rise in the temperature. The runoff increases with the increase in the intensity of the rainfall. This is so because an intense rainfall in a short duration results in less evaporation and infiltration losses. On the other hand a mild rainfall extending over a longer duration would produce less runoff. Further a greater areal extent of the rainfall over the drainage basin and also the rainfall for a longer duration would produce greater runoff.

26.8.1.2 Characteristics of drainage basin

The runoff is considerably affected by the characteristics of the drainage basin such as size, shape, surface, orientation, altitude, topography and geology of the drainage basin.

In general larger the drainage basin greater will be the runoff, but it may not be always true. This is so because for a larger drainage basin it is very seldom that the precipitation of same intensity occurs over the entire basin. Further a larger drainage basin may have larger storage effect resulting in the reduction of the runoff. On the other hand even for a smaller drainage basin with intense rainfall, uniformly spread over the entire basin, greater runoff rate may be produced.

A drainage basin may be either fan-shaped or fern-shaped. A fan-shaped drainage basin produces a greater runoff rate than a fern-shaped drainage basin. This is so because in a fan-shaped drainage basin all the tributaries of a stream are of comparable length and hence the runoff from each tributary reaches almost simultaneously. But with a fern-shaped drainage basin reverse is the case.

The surface of the drainage basin also has substantial effect on the runoff. For example a bare surface gives more runoff as compared with a surface having vegetal covering. Similarly a paved surface also produces a greater runoff.

For the drainage basins located on the windward side of the mountains heavy precipitation occurs and hence greater runoff is produced. On the other hand if the drainage basin is located on the leeward side it receives less precipitation and hence less runoff is produced. Similarly the drainage basins located at higher altitude receive more precipitation and hence yield greater runoff.

The topography of the drainage basin also has significant effect on the runoff. A steeply slopping drainage basin produces greater runoff because a steep slope helps quicker drainage and there being virtually no depression storage, less opportunity for evaporation and infiltration losses is provided. On the contrary a drainage basin with mild slope or with undulating land will yield less runoff.

The runoff is also significantly affected by the geology of the drainage basin. A drainage basin with pervious soil will yield less runoff due to larger infiltration losses, while the one with impervious soil

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or having a hard stratum will produce large runoff due to less infiltration losses. Further if a drainage basin comprises unfissured rock it will produce greater runoff, but the runoff will be considerably reduced if the rock is fissured.

26.8.1.3 Meteorological characteristics

The runoff is significantly affected by the meteorological characteristics such as temperature, humidity, wind velocity, pressure variation etc. Higher temperature and higher wind velocity cause more losses due to evaporation and transpiration and hence reduce the runoff. On the other hand greater humidity decreases evaporation and transpiration losses, resulting in greater runoff. The variation of the pressure in the atmosphere helps the movement of storms causing the precipitation. If the direction of the storm is the same as the direction of flow of water in the drainage basin, then water will remain in the basin for lesser period and hence more runoff will be produced in a short duration, as compared to the case when the storm is moving in the opposite direction.

26.8.1.4 Storage Characteristics

The storage characteristics of a drainage basin also have significant effect on the runoff. If a drainage basin has a larger number of natural depressions, pools, lakes etc., which will store a part of the precipitation and the same will not appear as runoff, then the runoff at the out-flow point of the basin will be reduced. Similarly the creation of artificial reservoirs or tanks will also increase the storage capacity of a drainage basin, resulting in the reduction of the runoff. However, if from these surface storages of the basin, water is released subsequently, it will appear in the form of delayed runoff. Further for drainage basins having pervious deposits, large ground water storage may be created, which may also appear in the form of delayed runoff.

26.9 HYDROGRAPH

A hydrograph is a graphical plot of discharge of a natural stream or river versus time. Discharge is usually expressed in cumec, or hectare-metre per day (or per month) and time is expressed in hours, days or months. Discharge is plotted on Y-axis and the corresponding time is plotted on X-axis. Figure 26.10 shows a hydrograph of a typical stream resulting from rainfall of intensity as indicated by a hyetograph plotted at the top left corner. At the beginning there is only base flow i.e., the ground water contribution to the stream as indicated at *A* in Fig. 26.10. After the commencement of rain which lasts for some time when the initial losses like interception and infiltration are met, the surface flow begins and there is progressive increase in the stream flow. The hydrograph therefore gradually rises and reaches a peak value at *B* at time t_p, called time lag or basin lag or time to peak, measured from the centroid of the hyetograph of effective rainfall (defined as that portion of the total rainfall which enters the stream as direct runoff). Thereafter the flow in the stream recedes and hence the hydrograph declines. Thus a hydrograph consists of rising limb *AB* and recession limb *BC*. Discharge at *B* corresponds to the maximum or peak flood discharge and the area under the curve *ABC* would give the volume of the total runoff during the flood.

A single peaked hydrograph as shown in Fig. 26.10 is developed when there is a single storm causing precipitation of more or less uniform intensity. However, in actual practice the hydrograph may not be a smooth single peaked curve as shown in Fig. 26.10, but it may be a multiple peaked curve known as complex hydrograph, due to several factors such as the occurrence of precipitation of different intensities with some interval, the characteristics of the drainage basin etc.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

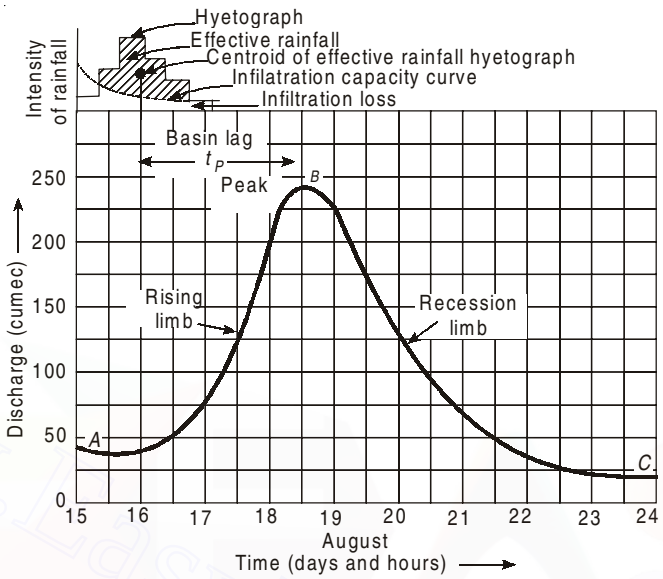


Figure 26.10 *Typical hydrograph*

26.10 METHODS OF DETERMINATION OF RUNOFF

The runoff may be determined either by the direct measurement of the stream flow or it may be estimated in an indirect manner. The stream flow may be measured either by providing a weir across the stream or by velocity area method. Out of these two methods the velocity area method is commonly adopted for measurement of stream flow, which has been described in section 14.15 of Chapter 14. Further the elevation of the water surface in the stream measured above an arbitrary datum at any section of the stream is known as stage. Corresponding to the measured discharge of the stream the stage is also recorded with the help of a gage provided at the section of the stream where the discharge is measured. By plotting the stage against the corresponding discharge a stage-discharge curve may be prepared for a site, which may be used for determining the discharge corresponding to a known stage. During high floods it is often impossible to make the discharge measurement. In such cases after the flood subsides, the stage corresponding to the flood flow may be obtained by noting the elevation of high water marks on the banks of the stream and corresponding to this value of the stage the maximum or peak value of flood discharge may be determined from the extrapolated stage-discharge curve. Alternatively the maximum flood discharge may also be determined by the slope-area method. In this method by noting the high water marks along the banks of the stream the cross-sectional area of the flow section as well as the water surface slope may be computed and using the Manning's formula, with suitably assumed value of roughness coefficient, the corresponding maximum value of the flood discharge may be determined.

The runoff may be indirectly estimated by the following methods:

(a) by empirical formulae relating rainfall and runoff;

(b) by infiltration method;

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(c) by rational method; and

(d) by unit hydrograph method.

(a) Empirical formulae relating rainfall and runoff. Several empirical formulae, curves and tables relating the rainfall and runoff have been developed which may be used for the estimation of runoff. Some of the empirical formulae are as indicated below:

(i) *Inglis and De Souza's Formula*

For ghat area

$$R = 0.85\,P - 305 \qquad ...(26.7)$$

For plains

$$R = \frac{(P-178)P}{2540} \qquad ...(26.8)$$

where R is average annual runoff in mm and P is average annual rainfall in mm over the entire drainage basin.

(ii) *Lacey's formula (for Indo-Gangetic Plain)*

$$R = \frac{P}{1+\dfrac{3048}{P}}\left(\frac{F}{S}\right) \qquad ...(26.9)$$

where R is average annual runoff in mm; P is average annual rainfall in mm over the entire drainage basin; F is a monsoon duration factor varying between 0.5 to 1.5; and S is the catchment (or drainage basin) factor which varies in the range of 0.25 for flat lands to 3.45 for hilly areas.

(iii) *A.N. Khosla's Formula*

$$R = P - \frac{T}{0.208} \qquad ...(26.10)$$

where R is average annual runoff in mm; P is average annual rainfall in mm over the entire drainage basin; and T is mean annual temperature in ℃ for the entire drainage basin.

(iv) *Formulae for some of the drainage basins in India*

Ganga basin, $R = 4.9025\,P^{0.64}$ ...(26.11)

Yamuna basin, $R = 0.1112\,P^{1.1}$ ...(26.12)

Rihand basin, $R = P - 1.6150P^{0.86}$ ...(26.13)

Chambal basin, $R = 120\,P - 49\,450$ ...(26.14)

where R is the average annual runoff in mm and P is average annual rainfall in mm over the entire drainage basin.

In addition to the empirical formulae, several investigators have presented the rainfall and runoff relationships in the form of curves and tables such as Binnie's percentage tables, Strange's curves and tables, Barlow's tables etc., which may also be used to determine the average annual runoff.

Several empirical formulae have also been developed for estimating the maximum or peak value of the flood discharge. In these formulae the maximum flood discharge Q of a river is expressed in terms of one or more of the flood producing factors, the most common being the catchment area A. Some of the important formulae used in our country are listed in Table 26.1.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

TABLE 26.1 *Empirical formulae for maximum flood discharge of rivers*

S. No.	Author	Formulae	Remarks
1	Dicken	$Q = CA^{3/4}$ C = 11.4 for Northern India; 13.9 to 19.5 for Central India; and 22.2 to 25 for Western Ghats. Max. value of C = 35.	Generally applicable for moderate size river basins in North and Central India
2	Ryves	$Q = CA^{2/3}$ C = 6.75 within 24 km of coast; 8.45 for areas between 24 and 161 km from the coast; 10.0 for limited area near the hills.	Generally applicable for the river basins in South India
3	Inglis	$Q = \frac{123A}{\sqrt{A + 10.4}}$	Derived using data of rivers of Maharashtra
4	Ali Nawaz Jung	$Q = C\,[0.386A]^{[0.925-K]}$ C = 48 to 60; Max. value of C = 85 $K = (1/14) \log 0.386\,A$ **Note:** In all the above formulae Q is flood peak discharge in cumec and A is catchment area in sq km.	Derived for the river basins of former Hyderabad State. Applicable with lower values of C for river basins of South India and with higher values of C for river basins of North India.

(*b*) *Infiltration method.* By this method the runoff may be estimated either by the use of the infiltration capacity curve, or by the use of the infiltration indices. In the first method the infiltration capacity curve for a given soil and moisture conditions is subtracted from the curve of rainfall pattern to derive the excess rainfall which represents the runoff. In the second method by subtracting the predetermined values of the infiltration indices from the rainfall intensity the excess rainfall representing the runoff may be obtained.

The infiltration capacity method is rational and would give good results if there is high degree of areal uniformity in rainfall pattern and in infiltration capacity, and if reasonably accurate infiltration curves are available. On the other hand the infiltration index methods are largely empirical and the derived values of the indices are applicable only when the rainfall characteristics and initial soil moisture conditions are indentical to those for which these are derived. However, the infiltration index methods are best suited for estimating runoff which would be produced by a rainfall occurring on a drainage basin with its soils being saturated either by a preceding rainfall, or shortly after the starting of the rainfall of higher intensity and longer duration, so that the rate of infiltration might be assumed to have reached a final constant rate prior to or shortly after the starting of the rainfall.

(*c*) *Rational method.* In the rational method the basic equation commonly known as rational formula is used which correlates runoff and rainfall and is as follows

$$Q = CIA \qquad \ldots(26.15)$$

where Q is flow in hectare-metre per hour (or cubic metre per hour); I is intensity of rainfall in metres per hour, A is area of the drainage basin in hectares (or square metres); and C is a runoff coefficient.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The value of the runoff coefficient C depends on the characteristics of the drainage basin such as soil type, vegetation, geological features etc. For different types of drainage basins the values of C are as given below.

Types of drainage basin	*Value of C*
Rocky and impermeable	0.8 – 1.0
Slightly permeable, bare	0.6 – 0.8
Cultivated or covered with vegetation	0.4 – 0.6
Cultivated absorbent soil	0.3 – 0.4
Sandy soil	0.2 – 0.3
Heavy forest	0.1 – 0.2

In this method the entire area of the drainage basin lying on the upstream of the point where the runoff is estimated, is divided into a number of sub-areas such that the time taken for the flow to reach the reference point (i) from the extreme dividing line of the nearest sub-area is one hour; (ii) from the extreme dividing line of the next sub-area is two hours and so on. Then knowing the intensity and duration of the rainfall and the values of the runoff coefficient for each sub-area at different times, and using the rational formula the runoff contributed by each sub-area at different times can be obtained. By adding the runoff contributed by each of the sub-areas during every hour, the direct runoff obtained from the entire drainage basin during every hour at the reference point is determined.

The rational method can also be used to estimate the maximum flood discharge for which the following formula is used

$$Q = CI_C A \qquad ...(26.16)$$

where Q is the maximum flood discharge in hectare-metre per hour (or cubic metre per hour); A is area of the drainage basin in hectares (or square metres); I_c is the critical intensity (or design intensity) of rainfall in metres per hour corresponding to the time of concentration t_c for the drainage basin; and C is a runoff coefficient.

The time of concentration is defined as the time which would be required for the surface runoff from the remotest part of the drainage basin to reach the point where the runoff is estimated. This time will vary, generally depending on the slope and character of the drainage basin.

The critical intensity of rainfall I_c can be obtained from the equation

$$I_C = \frac{P}{t_R}\left(\frac{t_R + 1}{t_c + 1}\right) \qquad ...(26.17)$$

where P is maximum precipitation in metres that occurs during a storm period of t_R hours. However, if the time of concentration t_c is not known then

$$I_C \approx \frac{P}{t_R} \qquad ...(26.18)$$

The value of the runoff coefficient C depends on the characteristics of the drainage basin and the same for different types of terrain are as given below.

Types of Terrain	*Value of C*
Flat residential, 30% impervious area	0.40
Moderately steep residential, 50% impervious area	0.60
Built up area, 70% impervious	0.80

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Contd.

Types of Terrain	*Value of C*
Flat cultivated land, sandy soil	0.20
Rolling cultivated land, clay-loam soil	0.50
Hilly area, forests, clay and loamy soils	0.50

(*d*) *Unit hydrograph method.* The unit hydrograph method was first proposed by L.K. Sherman in 1932. It is one of the widely used methods for the estimation of the flood discharge of a stream. By this method the entire flood hydrograph corresponding to any anticipated rainfall may be developed. A brief outline of the method is given below.

A *unit hydrograph* may be defined as a hydrograph of direct surface runoff resulting from one centimetre of effective rainfall falling uniformly over the drainage basin or catchment in space as well as in time for a specified or unit duration. The effective rainfall also called rainfall excess is that part of the total rainfall that enters the stream as direct runoff. The specified or unit duration is the period within which the effective rainfall is considered to be uniformly distributed. A unit hydrograph is designated according to the unit duration. For example if a unit hydrograph results from a rainfall of 3 hours unit duration it is known as a 3 hours unit hydrograph, meaning thereby a hydrograph produced by surface runoff from a storm lasting for 3 hours and yielding a rainfall excess of 1 centimetre.

Since the area under a hydrograph represents the total runoff, it is evident from the definition of unit hydrograph that the area under the unit hydrograph represents a direct runoff of 1 centimetre.

The theory of unit hydrograph is based on the following assumptions:

(i) The effective rainfall is uniformly distributed within its duration or specified period of time.

(ii) The effective rainfall is uniformly distributed over the entire area of the drainage basin.

(iii) For a given drainage basin, the base period of the hydrographs of direct runoff corresponding to effective rainfalls of different intensities but same unit duration is constant.

(iv) The ordinates of all the direct runoff hydrographs of a common base period are directly proportional to the total amount of direct runoff represented by each hydrograph.

(v) For a given drainage basin, the hydrograph of direct runoff corresponding to a given period of rainfall reflects all the physical characteristics of the basin.

For the application of the unit hydrograph method a unit hydrograph of suitable unit duration is derived from an observed hydrograph of the drainage basin under consideration. Then using the unit hydrograph so derived the flood hydrograph corresponding to any rainfall of the same unit duration may be obtained for the drainage basin on the basis of the above noted assumptions. In order to derive the unit hydrograph from an observed hydrograph the following steps may be adopted.

(i) Select a hydrograph resulting from an isolated, intense, short-duration rainfall of nearly uniform distribution over the entire drainage basin as well as within its duration.

(ii) Separate the base flow from the total runoff represented by the hydrograph, in order to obtain the direct runoff. For base flow separation several methods are adopted but all the methods are empirical. One of the methods to separate the base flow is by drawing a line *AC* tangential to both the limbs of the hydrograph at its lower portion as shown in Fig. 26.11 (a). The shaded portion below line *AC* represents the base flow. In another method as shown in Fig. 26.11 (b), the recession curve existing prior to the occurrence of the rainfall is extended to a point *D* on a vertical line passing through the peak *B* of the hydrograph. From point *D* a straight line *DE* is drawn to the hydrograph where *E* is a point on the hydrograph *N* days after the peak of the hydrograph. The value of *N* in days is approximately given by

$$N = 0.826\, A^{0.2} \qquad ...(26.19)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

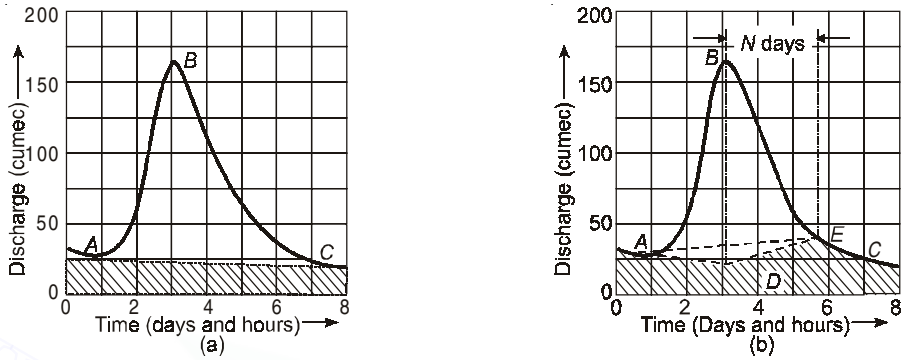


Figure 26.11 *Methods of base flow separation*

where A is the area of the drainage basin in square kilometres. The shaded portion below $ADEC$ represents the base flow. Many a time points A and E are joined by a straight line as shown in Fig. 26.11 (b) and the area below this line is taken as the base flow.

(iii) Draw a series of ordinates at some uniform time interval for the entire hydrograph.

(iv) From these ordinates of the hydrograph by deducting the corresponding ordinates of base flow obtain the ordinates of direct runoff.

(v) Find the volume of the direct runoff which is equal to $(\Sigma 0 \times t \times 3600]\ m^3$, where $\Sigma 0$ is the sum of the direct runoff ordinates in cumec and t is the time interval in hours between the successive ordinates.

(vi) Divide the volume of direct runoff by the area of the drainage basin to obtain the equivalent depth of direct runoff which is given by

$$\text{Direct runoff depth} = \frac{[\Sigma 0 \times t \times 3600] \times 100}{A \times 10^6} \text{ cm}$$

$$= \frac{0.36 \times \Sigma 0 \times t}{A} \text{ cm} \quad \text{...(26.20)}$$

where A is the area of the drainage basin in square kilometres.

(vii) By dividing each of the ordinates of direct runoff by the depth of direct runoff obtain the ordinates of the unit hydrograph. By plotting these ordinates at their respective times and drawing a smooth curve passing through these, unit hydrograph is obtained. The unit duration of the unit hydrograph so derived is same as the duration of effective rainfall corresponding to which the resulting hydrograph has been used for deriving the unit hydrograph. The use of unit hydrograph for deriving a flood hydrograph is given below.

From the above noted assumptions of the unit hydrograph theory it may be inferred that an effective rainfall of intensity R cm occurring uniformly over the entire drainage basin in a unit duration of T hours will produce a direct runoff hydrograph of the same base period as that of a unit hydrograph of T hours unit duration but having its ordinates R times those of the unit hydrograph. The principle involved in this inference is known as the principle of linearity. By applying the principle of linearity the flood hydrograph that would result from a rainfall of known intensity and its duration same as the unit duration of the unit hydrograph may be obtained as follows :

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(i) Deduct the infiltration losses from the rainfall intensity to obtain the intensity of effective rainfall.

(ii) Multiply the ordinates of the unit hydrograph by the intensity of effective rainfall to obtain the ordinates of the direct runoff hydrograph.

(iii) Add the corresponding base flow ordinates to the direct runoff ordinates to obtain the ordinates of the anticipated flood hydrograph which may be plotted.

A unit hydrograph may also be used to derive a flood hydrograph that would result from a series of rainfalls, each of different intensity but of unit duration same as that of the unit hydrograph. For this by applying the principle of linearity for each rainfall separate hydrograph of direct runoff, beginning at the same time as the rainfall, is obtained. Then applying the principle of superposition the ordinates of the individual direct runoff hydrographs at the same times are summed up to obtain the ordinates of the total direct runoff hydrograph that would result from the series of rainfalls. To the total direct runoff ordinates the corresponding base flow ordinates are added to obtain the ordinates of the anticipated flood hydrograph due to the series of rainfalls.

Further a unit hydrograph of specified duration can also be used to derive a flood hydrograph that would result from a rainfall of uniform intensity but duration longer than the specified duration of the unit hydrograph. This may be accomplished by dividing the total rainfall into a series of rainfalls, each of duration equal to the specified duration of the unit hydrograph. Then adopting the same method as indicated above the anticipated flood hydrograph for this case may also be obtained.

SUMMARY OF MAIN POINTS

1. Hydrology may be defined as the science that deals with the process governing the depletion and replenishment of the water resources of the land areas of the earth.
2. The hydrologic cycle is the descriptive term applied to the general circulation of water from the oceans to the atmosphere, to the ground and back to the oceans again.
3. Precipitation includes all moisture that reaches the earth's surface in liquid or solid form due to condensation of the atmospheric vapour.
4. For the measurement of rainfall rain gages are used which may be (i) non-recording type, and (ii) recording type.
5. Mean depth of rainfall over an area may be obtained by (i) arithmetic mean method, (ii) Theissen polygen method, and (iii) isohyetal method.
6. Evapo-transpiration is defined as the total volume of water per unit area used by the vegetal growth of a given area in transpiration and building of plant tissue and that evaporated from the adjacent soil or intercepted precipitation on the area in specified time.
7. Infiltration is the process by which water enters the surface strata of the soil and moves downwards to join the ground water.
8. ϕ-index may be defined as the average rainfall intensity above which the volume of rainfall equals the volume of runoff.
9. Runoff is that part of precipitation as well as any other flow contribution, which is transmitted through streams or rivers.
10. A hydrograph is a graphical plot of discharge of a natural stream or river versus time.
11. The runoff may be indirectly estimated by : (i) empirical formulae relating rainfall and runoff; (ii) infiltration method; (iii) rational method; and (iv) unit hydrograph method.
12. A unit hydrograph may be defined as a hydrograph of direct surface runoff resulting from one centimetre of effective rainfall falling uniformly over the catchment in space as well as in time for a specified or unit duration.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

PROBLEMS

26.1 Describe briefly with a neat sketch the hydrologic cycle.

26.2 What do you understand by the term precipitation?

26.3 Describe with neat sketches the different types of rain-gages used for measuring the rainfall.

26.4 Discuss (i) Theissen polygon method, and (ii) Isohyetal method for determining the average rainfall.

26.5 Define: evaporation, transpiration and evapo-transpiration.

26.6 Differentiate between infiltration rate and infiltration capacity.

26.7 How can infiltration capacity be determined in the field?

26.8 What are infiltration indices?

26.9 What is runoff? On what factors does runoff depend?

26.10 What are the various methods of estimating runoff from a catchment?

26.11 What is a hydrograph? Indicate the various components of hydrograph.

26.12 How can runoff be estimated by infiltration method?

26.13 What is a unit hydrograph? How can it be constructed from a flood hydrograph of a drainage basin ? Indicate the various assumptions involved in the unit hydrograph theory.

26.14 Indicate the use of unit hydrograph for developing a flood hydrograph that would result from a series of rainfalls of different intensities but duration same as the unit duration of the unit hydrograph.

26.15 What is base flow ? Describe briefly how base flow may be separated from the total runoff.

26.16 Compute and plot the ordinates of a flood hydrograph resulting from a 4 hour rainfall of intensity 10.5 cm. The ordinates of 4 hour unit hydrograph are given below:

Time (hours)	03	06	09	12	15	18	21	24
Ordinates of U.H (cumec)	0	120	370	505	397	315	255	240
Time (hours)	27	30	33	36	39	42	45	48
Ordinates of U.H (cumec)	180	135	100	70	45	25	15	0

Assume an initial loss of 5 mm and infiltration index of 1 mm/hour. Base flow is 15 cumec.

Apprise Education, Reprise Innovations
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Water Power Engineering

Chapter 27

27.1 INTRODUCTION

Water power (or hydropower) is one of the major sources of energy. The other sources of energy being the thermal power developed by the fuels such as coal, oil, etc., and the nuclear (or atomic) power. These are some of the conventional sources of energy. However, certain non-conventional sources of energy have also been developed such as tidal power, solar energy, geo-thermal energy, wind power, etc., but these are not the major sources of energy and are still in the developing stage. In this chapter various aspects of water power engineering are briefly described.

27.2 HYDROELECTRIC POWER DEVELOPMENT OF INDIA AND THE WORLD

India has a vast hydropower potential which is estimated to be about 90 million kW at 60% load factor. So far very little percentage of this vast potential has been harnessed. As such in order to develop this vast potential of power efforts are being made and several projects have been undertaken to generate more and more hydroelectric power.

The major portion of power in India is generated by hydroelectric power plants and thermal power plants, in the ratio 2 : 3. A start has, however, been made to develop large scale atomic power in the country with Tarapur Atomic Power Station having installed capacity of 0.44 million kW, Rana Pratap Sagar Atomic Power Station having installed capacity of 0.22 million kW, and a few others. But even with all this the present (upto 1982–83) total installed capacity in India is only about 35.34 million kW. This is made up of 13.04 million kW hydropower, 21.44 million kW thermal power and 0.86 million kW nuclear power. Table 27.1 gives the salient features of some of the major hydropower projects of India.

As regards the world's water power potential considerably varying estimates have been given. According to one assessment the world's water power potential at 100% efficiency and utilisation is estimated as about 5 609 million kW, while according to another assessment the world's technically exploitable water power potential is estimated as only about 2 724 million kW. It will thus be observed that the later figure of the world's water power potential is almost half of the former. The reason given for so much variation in the two assessments being that the former estimate is based on theoretical considerations while the later estimate is based on practical considerations. However, as indicated later such a vast variation in the assessment of the world's water power potential is bound to exist.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The world's hydroelectric capacity installed uptodate is about 200 million kW, which is thus about 9% of the later assessment (i.e., 2724 million kW) of the world's total hydropower potential.

The development of the hydropower in the various countries of the world varies considerably. In Europe about 50% of the available water power potential has been developed, and in U S A. only about 30% of the available water power potential has been developed. Furthermore, the share of the hydropower in the total energy developed by any country also varies considerably for different countries. For example, in the developed countries the share of the hydropower ranges from about 17% (in U.S.A., U.S.S.R.) to 98% (in Sweden, Norway) of the total energy. However, in developing countries the usage of their hydropower resources has only been started and so far relatively small percentage of it has been developed.

TABLE 27.1 *Major Hydro Power Projects of India*

Name of Power Station (1)	*River* (2)	*State* (3)	*Installed Capacity MW* (4)	*Total cost in Rs. Million* (5)	*Year of Commence ment* (6)	*Year of Compl- etion* (7)
Bhakra	Sutlej	Himachal Pradesh	1 050	1 383.33	1948	1968
Ganguwal		Punjab	77			
Kotla		Haryana	77			
Gandhi Sagar	Chambal	Madhya Pradesh	115	136.03	1954	1960
Rana Pratap Sagar	–do–	Rajasthan	172	278.40	1961	1970
Jawahar Sagar	–do–	–do–	99	184.20	1962	1973
Rihand	Rihand	Uttar Pradesh	300	378.00	1952	1966
Obra	–do–	–do–	99	242.40	1963	1971
Ukai	Tapti	Gujarat	300	1 044.00	1960	1974
Koyna Stage I & II	Koyna	Maharashtra	540	564.39	1954	1968
Koyna Stage III	–do–	–do–	320	380.00	1966	1975
Machkund	Machkund	Andhra Pradesh	114.75	182.79	1946	1961
Upper Sileru	Sileru	–do–	120	185.00	1958	1968
Lower Sileru Stage I	–do–	–do–	460	540.00	1964	—
Lower Sileru Stage II	–do–	–do–	230	—	—	—
Sharavathy	Sharavathy	Karnataka	891	1 067.73	1956	1974
Sabarigiri	Pamba & Kakki	Kerala	300	430.00	1961	1968
Balimela Stage I	Sileru	Orissa	360	447.10	1961	—
Balimela Stage II	–do–	–do–	480	—	—	1977
Ramganga	Ramganga	Uttar Pradesh	198	96.86	1961	1978
Srisailam Stage I	Krishna	Andhra Pradesh	440	747.00	1963	1984
Idukki	Periyar	Kerala	390	682.10	1963	1976
Kalinadi	Kali	Karnataka	910	—	1968	—
Nagarjuna Sagar	Krishna	Andhra Pradesh	440	1649.00	1956	1974
Nagarjuna Sagar (Pumped Storage)	–do–	–do–	110	—	1970	—
Yamuna Stage I	Tons	Uttar Pradesh	84.75	168.30	1960	1970
Yamuna Stage II	–do–	–do–	240	—	1966	1975
Kundah	Kundah	Tamil Nadu	435	—	1956	1967
Mettur (Tunnel Power House)	Cauvery	–do–	200	118.16	1960	1966
Hirakud	Mahanadi	Orissa	270	827.60	1948	1964

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 27.1 Contd.

(1)	(2)	(3)	(4)	(5)	(6)	(7)
Beas Unit I (Beas Sutlej Link)	Beas	Himachal Pradesh	990	1 468.70	1961	1977
Beas Unit II	–do–	–do–	360	1 676.70	1960	1974
Salal	Chenab	Jammu & Kashmir	540	551.50	1969	1976
Baira-Suil	Baira	Himachal Pradesh	201	204.90	1969	—
Loktak	Manipur	Manipur	105	102.50	1969	—

27.3 COMPARISON OF THERMAL AND HYDROELECTRIC POWER COSTS

The initial cost of a hydroelectric power plant of given capacity is likely to be higher than that of a thermal power plant of equal capacity. This is so because of the more expensive civil works and longer transmission lines for the former. A thermal power plant can usually by located close to its load centre. However, as a hydroelectric plant requires no fuel and it is much simpler to operate and maintain, the operating costs of a hydropower plant are much less than those of thermal power plant. Although the improvements in design have brought about a reduction in average coal consumption for a steam plant per kWh of energy generated from 1.4 kg in 1920 to 0.4 kg in 1960, yet the cost of power production in a thermal plant is usually higher. Moreover the cost of coal is largely dependent on the cost of transport from the mines to the power house. In India the ratio of the cost of water power generation to that of thermal power generations ranges from about (1/3) to (3/4). However, power system in these days incorporate both hydro (or hydel) as well as thermal power plants. This enables each of these to be used to better advantage.

27.4 ASSESSMENT OF AVAILABLE POWER

A hydropower development is essentially to utilize the hydraulic power (or energy) possessed by the water flowing in a stream and to develop from it electric power (or energy) through hydraulic turbines coupled to electric generators. The total amount of hydroelectric power that would be made available from a stream depends on continuous flow rate of the stream and the head possessed by the flowing water. Thus if Q is the rate of flow of water that may be continuously made available from any stream and it possess head H, then the amount of hydropower in kW that would be available may be expressed as

$$P = \frac{wQ(H - h_f) \times \eta}{1000}$$

or

$$P = \frac{9810Q(H - h_f) \times \eta}{1000}$$

$$= 9.81\, Q\, (H - h_f)\, \eta \qquad \ldots(27.1)$$

where $(H - h_f)$ is the net or effective head on the turbine and η is the efficiency of the hydropower station.

From Eq. 27.1 the energy in kilowatt-hours (kWh) may be expressed as

$$E = 9.81\, Q\, (H - h_f)\, \eta \times T \qquad \ldots(27.2)$$

where T is the period of flow in hours.

From the above equation for power (or energy) it is obvious that the correct assessment of the power (or energy) that would be available from a stream would depend on the correct assessment of the discharge Q, of the stream and the net head $(H - h_f)$. Since the flow rate of stream varies considerably, for the assessment of power that would be available from it, what value of discharge Q should be

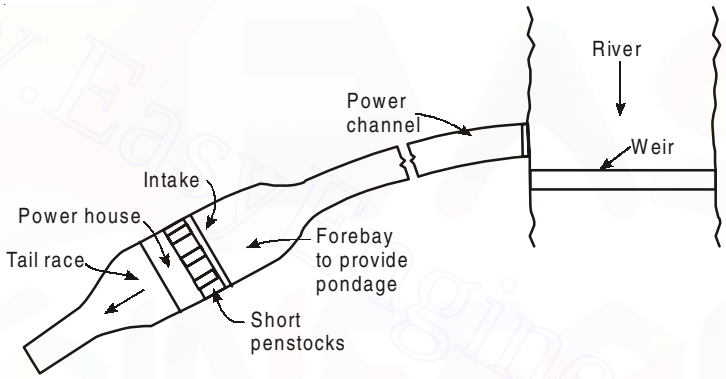
Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

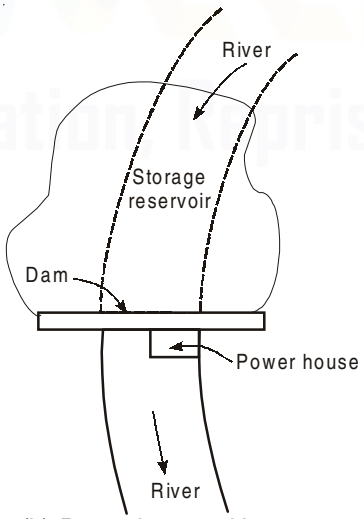
adopted is questionable. Moreover, as Q varies h_f also varies and therefore the net head available will also vary. As such the assessment of the hydropower that would be available from a stream would depend on the value of Q adopted in the above noted expressions and this would result in considerable variation in the assessment of the amount of the hydropower available from a stream. However, if a storage reservoir is developed by constructing a dam across a stream, then the variation of flow can be largely evened out and more or less constant discharge Q can be supplied to the turbines.

27.5 STORAGE AND PONDAGE

As stated earlier the flow rate of a stream varies considerably with time. For instance, during rainy season when the stream is in floods it carries a huge quantity of water as compared to other times of the year when the quantity of water carried by it is considerably less. However, the demands for the power ordinarily do not correspond to such variations of the natural flow of the stream. As such some arrangement in the form of storage and pondage of water is required to be made to regulate the flow of water so that it may be made available in requisite quantity at time when the generation of power is required.



(a) Power house with pondage



(b) Power house with storage

Figure 27.1 *Hydropower houses with pondage and storage*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Storage may be defined as impounding of a considerable amount of excess runoff during seasons of surplus flow for use during dry seasons. It is achieved by constructing a dam across a stream at suitable site, thus forming a storage reservoir on the upstream side of the dam.

Pondage may be defined as a regulating body of water in the form of a relatively small pond or reservior provided just at the hydropower plant. The pondage is necessary to regulate the flow to suit the variation in daily or weekly power (or load) demand. It is required to be provided either when the hydropower station is located at some distance away from the storage reservoir, or when the storage reservoir is not provided. When the hydropower station is located at some distance away from the storage reservoir and the water is supplied to the hydropower station through a power channel or a natural stream, then the regulation of outflow of water from the storage reservoir, to take care of the variation in daily or weekly power demand, may not be possible and a regulating body or pondage is needed directly at the hydropower station. Similarly when the storage reservoir is not provided and the hydropower station is located directly on the natural stream, then the pondage is necessary to regulate the natural flow of the stream to take care of the variation in daily or weekly power demand.

Thus in general the term *storage* is used to indicate the building up of the low natural flow of the stream to a uniform discharge and the term *pondage* is used to indicate the regulation of the resulting uniform flow, or the natural flow if there is no storage, to suit the variation in daily or weekly load demand. If the hydropower station is close to the storage reservoir then the storage reservoir itself will also provide the necessary pondage. Figure 27.1 shows the location of power houses with pondage and storage.

27.6 ESSENTIAL STREAM FLOW DATA FOR WATER POWER STUDIES

The essential stream flow data required for the assessment of the hydropower that would be available from any stream are as follows:

(i) Daily (weekly or monthly) flow over a period of years (the longer is the data better it is), to determine the plant capacity and estimated output, both of which are a function of the average flow of the stream as well as its distribution during the year.

(ii) Minimum or low-water flow to determine the amount of primary or firm or dependable power and also the size of thermal or stream auxiliary plant if used.

(iii) Maximum or flood-flow to enable proper design of spillway of the dam and to provide for the safety of the plant.

The above noted data for any stream may be developed by regularly measuring the stream flow and maintaining the records of its daily flow extending over a long period of time. The detailed past performance of a stream thus determined by measurement is the best index or measure of what may be expected in the future. The stream flow may also be estimated in an indirect manner by adopting some of the methods as indicated in section 26.10 of Chapter 26. However, by these methods only average annual stream flow and maximum or peak value of the flood discharge can be obtained.

From the daily (weekly or monthly) stream flow data a flow duration curve for the power site may be prepared which may be used to determine the plant capacity and available power at all times. In the following section the method of drawing a flow duration curve and its uses have been discussed.

27.7 FLOW DURATION CURVE

The *flow duration curve* as ordinarily constructed consists of a plot of values of stream flow (daily, weekly or monthly) in increasing order of magnitude, as ordinates and the per cent of time under consideration the flow equalled or exceeded, as abscissas. It may however be noted that instead of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

discharge, any function of discharge such as power in kW per metre of fall or for a given head, may also be used as ordinates. The curves thus obtained are termed as *power duration curves.* Moreover, in general these curves are also termed as *duration curves.*

Computations involved in the construction of flow duration curve of Fig. 27.2 are shown in Table 27.2. Rates of flow, differing by convenient amounts, are set up in Col. 1 in ascending order. In Col. 2 is indicated the number of times the daily flow of the stream equalled a value between the corresponding flow in Col. 1 and the flow next below. For example, during the year, a daily flow between 28 and 30 cumec occurred 13 times. A summation of occurrences is tabulated in Col. 3. It shows that a flow of 28 cumec was equalled or exceeded on 303 days during the year. Column 3 is then reduced to percent of time and recorded in Col. 4. Columns 1 and 4 are used to plot the flow duration curve. For a flow duration curve of 1 year period, Col. 2 will total 365 days and for a longer period than a year it will be more than 365 days.

A flow duration curve may also be constructed in the same manner with the help of mean weekly or monthly flow rates instead of daily flow rates. But the flow duration curves computed from mean monthly flows are only approximate, because in each month there is a considerable variation in flow not indicated in the duration curve. The error involved in the use of mean monthly flows for the construction of the duration curves usually ranges from 5 to 15% depending upon the characteristics of the stream and the extent of utilisation of the flow. The difference between the daily and monthly flow duration curves would be negligible for a stream with a steady flow, but much greater for very flashy streams.

The flow duration curve may either be constructed only for that year which is having the lowest flow rate out of the entire stream flow record, or it may be constructed by considering the entire stream

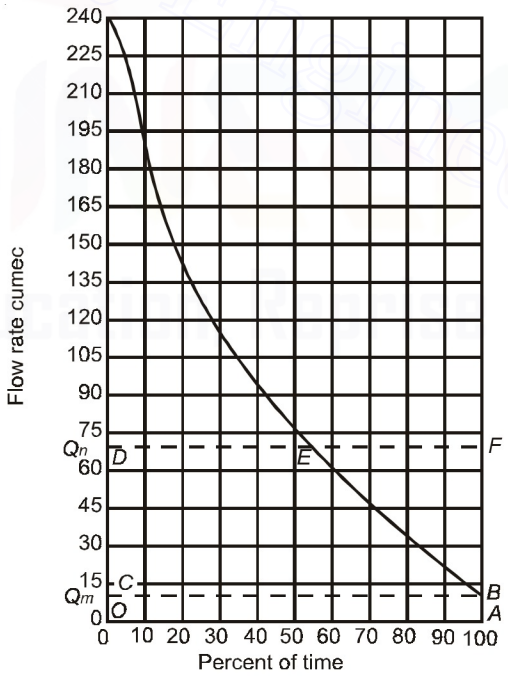


Figure 27.2 *Daily flow duration curve of typical river for the year having lowest flow of the record*

flow record. For the later case two methods are in general use for constructing the flow duration table for the plotting of the curve, which are as indicated below.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*i*) *Calender-year basis method.* In this method the mean monthly (or weekly or daily) flow rates of each year of the record are separately arranged in increasing order of magnitude and tabulated as first, second, third, etc., month (or week or day). There will thus be as many vertical columns in the table as the number of years of record considered. Then the mean value of the flow rates noted in each of the horizontal columns of the table is computed. Thus the mean of the first horizontal column of the table will give the flow rate for the driest month (or week or day) of the average year, the mean of the second horizontal column that for the next driest month (or week or day) etc. In this manner monthly (or weekly or daily) flow rates for the average year are obtained and the same are used to plot the flow duration curve as indicated earlier.

(*ii*) *Total period basis method.* In this method the mean monthly (or weekly or daily) flow rates for the entire period under consideration (irrespective of the calender year in which they occurred), are arranged in increasing order of magnitude. The flow duration curve is then prepared using the flow rates for the entire period in the same manner as indicated above. Alternatively the flow duration curve may also be constructed by adopting the following procedure. Continuous ranges of stream flow differing by convenient amount are considered and for each year separately the number of days during which the daily stream flow lies in a particular range are noted. The number of days, for all the years are considered, and those having the daily flow of the stream in the same range are then added, and the percent of time the daily stream flow occurred in a particular range is computed. The flow duration curve is then constructed by plotting the lower value of the range of flow rate against the corresponding percent of time.

TABLE 27.2 *Computations for daily flow duration curve of typical river for the year having the lowest flow of the record*

Flow rate cumec (1)	*Number of days* (2)	*Days equalled or exceeded* (3)	*Percent of time* (4)
10.00	1	365	100
12.00	12	364	99.7
14.00	8	352	96.4
16.00	6	344	94.2
18.00	9	338	92.6
20.00	10	329	90.1
22.00	6	319	87.4
24.00	7	313	85.7
26.00	3	306	83.8
28.00	9	303	83.0
30.00	13	294	80.5
35.00	6	281	77.0
40.00	14	275	75.3
45.00	21	261	71.5
50.00	14	240	63.7
55.00	10	226	61.9
60.00	10	216	59.2
65.00	5	206	56.4
70.00	5	201	55.0
75.00	18	196	53.7
80.00	11	178	48.7

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Table 27.2 Contd.

Flow rate cumec (1)	*Number of days* (2)	*Days equalled or exceeded* (3)	*Percent of time* (4)
85.00	17	167	45.7
90.00	13	150	41.1
95.00	13	137	37.5
100.00	10	124	34.0
105.00	2	114	31.2
110.00	8	112	30.7
115.00	13	104	28.5
120.00	13	91	25.0
130.00	14	78	21.4
140.00	4	64	17.5
150.00	5	60	16.5
160.00	0	55	15.1
170.00	6	55	15.1
180.00	10	49	13.4
190.00	10	39	10.7
200.00	8	29	7.9
210.00	7	21	5.7
220.00	5	14	3.8
230.00	9	9	2.4
	365		

Out of these two methods the total-period basis method gives a relatively true representation of the flow duration curve. This is so because in this method the actual flow rates of stream appear at appropriate places in the curve, but in the calendar-year basis method the stream flow rates might be averaged up with the values for the other years.

The area under the flow duration curve (Fig. 27.2) would evidently represent the average yield from the stream. As stated earlier for a hydropower project the flow duration curve can be easily converted into power duration curve by changing the ordinate, in which case the area under the curve would represent the average yield of power from the project. It can be noted that without storage, Q_m is the maximum flow rate that would be available for all the times (i.e., for 100 percent of time), and the area *OABC* would represent the firm yield of water or power. However, if a flow rate of Q_n is required to be provided which is indicated by flow demand line *DEF*, then it is evident that it would be possible to meet this uniform demand of flow rate (or power) for all the times only if storage equal to area *BEF* is provided. However, an alternative to storage, for power projects, is to instal a thermal power unit of maximum capacity equal to *BF* (on power duration curve ordinate in kW), to work as supplementary to hydropower unit. The curve also shows that natural flow sufficient to meet the flow demand Q_n is available for 53.5% of time or 195 days in this year of the lowest flow of the record.

27.8 MASS CURVE

In order to facilitate the storage computations mass curves are commonly used. A *mass curve* is a plot of accumulated flow (in hectare-metre or any other convenient unit) against time. It is usually made from the records of mean monthly flows of a stream. The mean monthly flows are summed up consecutively and each sum is plotted above the corresponding time upto which the summation has

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

been done. Figure 27.3 shows the mass curve for a typical stream. Obviously a mass curve must continuously rise as it shows accumulated flows. The slope of the curve at any point indicates the rate of flow at that particular time. If the curve is horizontal, the flow is zero, and if there is high rate of flow the curve rises steeply. Thus relatively dry periods are indicated as concave depressions on the mass curve.

As shown in Fig. 27.3 the slope of the straight line *AB* joining the end points of the mass curve represents the average discharge over the period for which the mass curve has been plotted. Further if two straight lines *A′B′* and *A″B″* are drawn parallel to *AB* and tangent to the mass curve at the lowest tangent point *C* and the highest tangent point *D* respectively, then the vertical intercept between these two straight lines represents the storage volume required to permit continuous release of water at this average discharge rate for the entire period. If the reservoir having this capacity is assumed to contain a volume of water equal to *AA′* at the beginning of the period, then the reservoir would be full at *D* and it would be empty at *C*. However, if the reservoir was empty in the very beginning, then it would be empty again at point *E* and also during the period from *F* to *B*. On the other hand if the reservoir was full in the very beginning, it would be full again at points *F* and *B*, and between points *A* and *E* there will be spill of water from the reservoir.

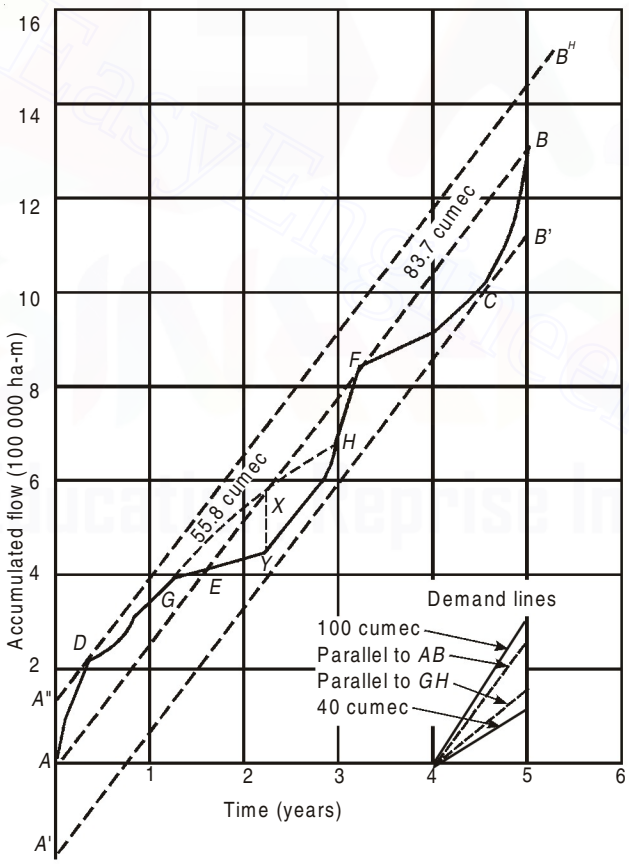


Figure 27.3 *Mass Curve*

The rate at which water is required for use in a power plant or for other requirements, is termed as rate of demand. If the rate of demand is constant, then corresponding to this rate of demand a demand

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

line may be obtained such that the slope of the demand line is equal to the rate of demand as shown in Fig. 27.3. Where the mass curve is steeper than the demand line, the natural stream-flow is at a higher rate than the rate of demand, and consequently, some of the stream-flow is available for storage or it will spill if the reservoir is already full. Where the mass curve has a flatter slope than the demand line, water will be withdrawn from storage in order to maintain the required rate of demand.

In order to obtain the storage required for a given rate of demand, a demand line corresponding to the given rate of demand is obtained. Then lines such as *GH*, shown in Fig. 27.3, are drawn parallel to the demand line and tangential to the high points of the mass curve. The maximum vertical intercept, such as *XY*, between such lines and the mass curve gives the storage required to satisfy the given rate of demand. This is so because, assuming the reservoir to be full at *G*, there is a total flow in the stream represented by the vertical distance between *G* and *Y* and there is a total demand represented by the vertical distance between *G* and *X*, leaving a gap of volume represented by *XY* which must be met from the storage.

The tangential line drawn parallel to the demand line when extended forward must intersect the mass curve, such as at *H*, so that the reservoir which was full at *G* will be filled again at *H*. However, if the line does not intersect the mass curve, the reservoir will not be filled again. Further if the reservoir is very large the time interval between the points *G* and *H* may be several years.

The rate of demand may not be always constant, in which case the demand line becomes a curve with its slope varying from point to point in accordance with the variable rate of demand at different times. Such a demand line is actually a mass curve of demand. In this case also the storage requirements can be obtained in the same way by superimposing the mass curve of demand on the mass curve of stream flow from the beginning of the dry period till the two meet again. The maximum vertical intercept between the two curves gives the requirement of storage. It is however essential that the demand line for variable demand coincide chronologically with the mass curve of streamflow, i.e., June demand must coincide with June inflow and so on.

In both the above noted cases the requirement of the storage so obtained would be the net storage which must be available for utilization and it must be increased by the amount of water lost by evaporation and seepage.

Mass curve may also be used to determine the maximum possible uniform rate of flow which may be obtained from a given storage capacity. For this slopping tangential lines are drawn at the high points of the mass curve in such a manner that the maximum intercept between these lines and mass curve is equal to the given storage capacity. The slope of these lines would give the uniform rate of flow which may be obtained from the given storage capacity. Further the slope of the flattest line would represent the firm rate of flow which may be obtained for the entire period from the given storage capacity.

27.9 TYPES OF HYDROPOWER PLANTS

Hydropower plants may be classified according to several considerations. According to the storage being provided or not hydropower plants may be classified as (i) Run-of-river plants, (ii) Reservoir or storage plants, (iii) Pumped storage plants, and (iv) Tidal plants.

(i) Run-of-river Plants. Run-of-river plants are those which utilize the flow as it comes, without any storage being provided. As such these plants would be feasible only on such rivers which have a minimum dry weather flow of such magnitude which makes the development worthwhile. A weir or barrage may be constructed across the river close to the power plant to maintain a given water level.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

These are generally low head plants and often at times of flood, tail water rises to such an extent that the plants are inoperative.

Many run-of-river plants are provided with pondage, which enables them to take care of hour-to-hour fluctuations in load on the plant throughout the period of a week. In the usual run-of-river hydroplants with pondage, installation is on a high basis in relation to minimum streamflow. In some plants, discharge is 10 to 20 times minimum streamflow.

(ii) Reservoir Plants. These are the hydropower plants which take their flow from large storage reservoirs developed by constructing dams across rivers. Depending on the storage volume, these plants can hold over surplus water from the period when stream flow exceeds demand for utilization during the period when demand exceeds the stream flow. Better utilisation of hydropower potential is thus achieved with such plants.

(iii) Pumped Storage Plants. Pumped storage hydro plants are those which pump all or portion of the water used by these plants, back to the head-water pond, to be made available again for the power generation. Essentially they consist of a tail-water pond and a head-water pond. During times of peak load, water is drawn from the head-water pond through the penstocks to operate hydroelectric generating units. This water is collected in the tail-water pond and during the off peak hours pumps are operated to pump the water back from the tail-water pond to the head-water pond. Power for operating the pumps is provided by some off peak thermal or hydropower plants.

For heads upto about 120 m special Francis turbines have been developed for the pumped storage plants, the runners of which are so shaped that they can be used both as turbine as well as pump. With such turbines which are known as *reversible turbines*, it is feasible to use the same unit for both pumping water and generating power. Alternatively pumped storage plants may have separate pumping and power generating units, especially for high heads for which multistage centrifugal pumps are used for pumping water and high head Francis turbines are installed for power generation.

(iv) Tidal Plants. Tidal plants do not involve any storage of water. These plants work on the principle that there is a rise in sea water during the high tide period and a fall during the low tide or ebb tide period. Sea water rises and falls twice a day, each full cycle occupying about 12 hours 25 minutes. The tidal range, or the difference between the high tide and the low tide levels is utilized to generate power. This is accomplished by constructing a basin separated from the sea by a wall and installing turbines in openings through this wall. During high tide water passes from the sea to the basin, thus running the turbines and generating power. During low tide water from the basin flows back to the sea which can also be utilized to generate power provided another set of turbines operating in the opposite direction is installed, or special turbines which can generate power for either direction of flow are installed. Thus depending on the type of installation either one or both directions of flow can be utilized to generate power.

Tidal plants are the recent inovations in the field of hydropower generation. However, these may be used only where the tidal range is high. Rance power plant in France is an example of this type of plants where the tidal range is about 11 m and it has 9 units of 38 000 kW each.

Hydropower plants may also be classified on a functional basis as (i) Base-load plants, and (ii) Peak-load plants.

(i) Base-load Plants. As the name indicates base-load plants are those which are capable of substantially continuous operation in the base of the load curve throughout the year. Both run-of-river

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

plants as well as reservoir plants can be used as base load hydro plants. However, when run-of-river plants without pondage are used as base load hydro plants, their full plant discharge is seldom more than the minimum flow of river.

(ii) Peak-load Plants. A peak-load plant is one designed and constructed primarily for taking care of the peak-load of a power system. Pumped storage plants are peak-load plants. Run-of-river plants with pondage can operate both as peak-load and base-load plants as river flow permits.

During season of high flow in the river, it is generally preferable to use the hydropower plants to maximum capacity as base-load plants and to use the thermal power plants when necessary, as peak-load plants, i.e., to generate the extra power needed beyond the capacity of the hydropower plants at peak-load periods. Conversely during periods of low flow, it would be better to use the thermal power plants to take the base-load and use hydropower plants with storage for taking the peak load. However, out of the hydropower and thermal power plants, the former have better peaking characteristics. Hydropower plants can be started at a notice seldom exceeding 3 to 4 minutes and there is absolutely no wastage when they are idle. But if thermal power plants are used for peak loads, the boiler fire must be kept lighted and thus there is a continuous loss of fuel, and even then it usually requires at least about 30 minutes before a steam unit can generate to full capacity.

Hydroplants may also be classified, on the basis of head under which they operate, as (i) Low head plants, (ii) Medium head plants, and (iii) High head plants.

A *low head plant* is the one which is operating under a head of less than about 30 m. Run-of-river plants are usually low head plants. A *medium head plant* is the one which is operating under a head between 30 to 250 m. A *high head plant* is the one which is operating under a head of more than about 250 m. Low heads as well as the lower ranges of medium heads may be made available by utilizing a steep slope or a fall in river or a canal. On the other hand the higher ranges of medium heads as well as high heads may be obtained by constructing dams of sufficient height and locating the power house either at the toe of the dam close to it, or in a deep depression if available at a certain distance away from the dam.

The above noted head ranges for the different types of hydropower plants are, however, arbitrary and it is customary to associate these head ranges with the types of turbines used. Thus hydropower plants for which impulse turbines are used are called high head plants. Similarly mixed flow turbines are used for medium head plants and axial flow turbines are used for low head plants. With the advances in the turbine design it has become possible to use axial and mixed flow turbines for higher heads. Consequently the ranges of head indicated above also move up.

27.10 TYPICAL HYDROELECTRIC DEVELOPMENTS OF INDIA

27.10.1 Bhakra-Nangal Hydroelectric Project

Figure 27.4 shows a schematic section of Bhakra-Nangal hydroelectric project which is one of the major hydroelectric projects of India. It consists of Bhakra dam constructed across the river Sutlej and Nangal barrage across the same river about 13 km downstream of Bhakra dam. There are two power houses one on the Left bank and the other on the right bank, located close to the dam. The left bank power house has 5 units of 90 MW each and the right bank power house has 5 units of 120 MW each. Francis turbines have been provided for these power plants which are classified as medium head plants and also they come under the category of storage or reservoir plants.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

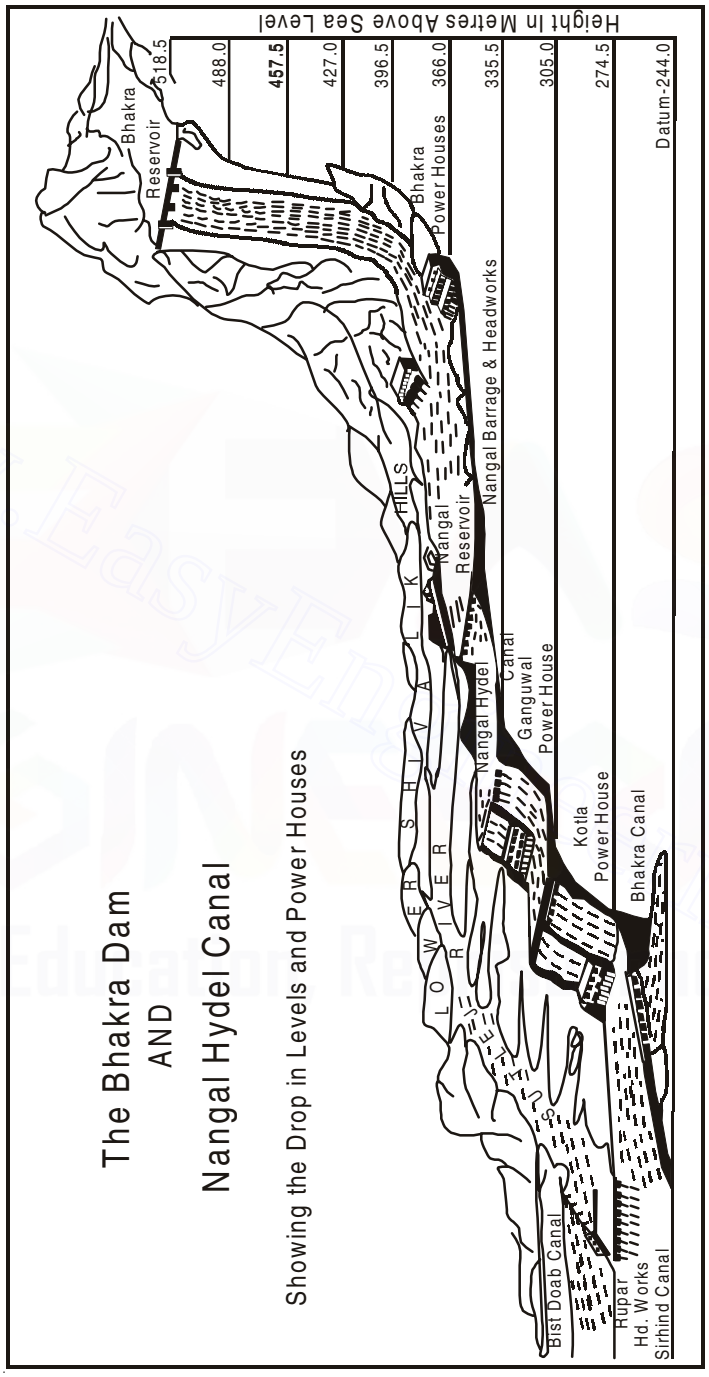


Figure 27.4 *Schematic Section of Bhakra-Nangal Hydroelectric Project*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

In addition to the above noted power houses, there are two more power houses in this project, which are located on a power channel taking off from the Nangal barrage. One of the power houses is located at Ganguwal which has in all 3 units out of which 2 units are 24 MW each and 1 unit of 29 MW. The other power house is located at Kotla on the downstream of Ganguwal and has in all 3 units of capacities exactly same as at Ganguwal. The power plants at Ganguwal and Kotla may be classified as low head plants.

27.10.2 Chambal Valley Development Scheme

The Chambal Valley Development Scheme is another major hydroelectric scheme of India. The full development consists of three dams with a power house at each dam site and a barrage. The three dams constructed in series on the Chambal river are Gandhi Sagar Dam in Madhya Pradesh and Rana Pratap Sagar Dam and Jawahar Sagar Dam in Rajasthan. At Gandhi Sagar Dam there are 5 units of 23 MW each. At Rana Pratap Sagar Dam there are 4 units of 43 MW each. At Jawahar Sagar Dam there are 3 units of 33 MW each. Francis turbines have been provided for each of these power plants which may be classified as medium head plants and also they come under the category of storage plants. A unique feature of Rana Pratap Sagar project is that since the power house is located in a pit below the river bed, the water after issuing from the draft tubes of the turbines is returned to the river at downstream through a standard horseshoe shaped tunnel 1466 m long and having 12 m internal diameter.

A barrage has been constructed across the Chambal river near Kota city in Rajasthan to divert the water released from the Storage and Power dams into irrigation canals.

27.11 FIRM (OR PRIMARY) AND SECONDARY POWER

The minimum power which can be generated throughout the year from a hydroelectric plant is called its *firm* or *primary power.* The power in addition to the firm power which would be generated for only a part of the year is known as *secondary* or *non-firm power.* Thus for a run-of-river plant without any storage the firm power would correspond to the minimum flow of the river which would be available throughout the year. However, by providing the storage the firm power can be considerably increased. Similarly, by having one or more thermal stations which can be utilised to generate extra power when hydroelectric generation is low, a part of the secondary power can be converted into firm power.

As described earlier a flow duration curve can be used to estimate the firm and secondary powers that would be generated from a stream. Thus as shown in Fig. 27.2 the flow rate Q_m would correspond to the firm power that would be available from the river without any storage being provided. But if the provision for the storage equal to area *BEF* is made then the firm power for the same river may correspond to a higher flow rate Q_n represented by line *DEF.* However, in the absence of any storage, area *BCDE* would correspond only to the secondary power that would be available from the river.

27.12 LOAD FACTOR, UTILISATION FACTOR AND CAPACITY FACTOR

Load factor is defined as the ratio of the average load during a certain period to the maximum or peak load during that period. The load factor is thus related to a certain period and therefore, there will be daily load factors, weekly load factors, load factors for a given month and yearly load factors. For instance, during a certain week a power plant generates 8 400 000 kWh and the peak load during the week is 100 000 kW. Therefore, the load factor during the week is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\frac{8400000}{24\times7\times100000} = 0.50 \text{ or } 50\%$$

The load factor of a power plant would vary greatly with the character of the load. In highly industrialised areas the load factors will be high, but in residential areas the load factors may be as low as 25 to 30%. The installed capacity of a power plant has to be equal to peak load but the total number of units (kWh) generated will be governed by the average load. Thus, if the load factor of a power plant is low, a large proportion of the generating capacity remains idle for most of the time, and the cost of generation is high. Conversely, if the load factor of a power plant is high the generating capacity is better utilised and the cost of generation is relatively less.

Utilization factor (or *plant-use factor*) is defined as the ratio of the peak load developed during a certain period to the installed capacity of the plant. It thus represents the maximum proportion of the installed capacity utilized during that period. With constant head, utilization factor would also be the ratio of water actually utilized for generating maximum power corresponding to peak load to that available in the river, and usually there will be little difference in this factor, whether expressed as a ratio of power or water. For a hydroelectric plant, utilization factor commonly varies from about 0.40 to 0.90, depending on plant capacity, load factor, available pondage and storage etc.

Capacity factor (or *plant factor*) may be defined as the ratio of the energy that the plant actually produces during any period to the energy that it might have produced if operated at full capacity throughout this period. For instance, if during a particular week the peak load on a power plant with a capacity of 100 000 kW was 65,000 kW and if the energy produced by the plant was 6 720 000 kWh, the capacity factor for the week (168 hr).

$$= \frac{6720000}{100000\times168} = 0.4 \text{ or } 40\%$$

During the same period the load factor

$$= \frac{6720000}{65000\times168} = 0.616 \text{ or } 61.6\%.$$

Capacity factor will be identical with load factor when the maximum or peak load just equals the plant capacity. For a hydroelectric plant, capacity factor commonly varies from about 0.25 to 0.70 or more, depending on load factor, plant capacity, available pondage and storage etc.

The thermal power plants may be operated at any desired capacity factor, whereas the annual capacity factor at which hydropower plants may operate is usually limited by the variation in the flow of water in the river. Theoretically a thermal power plant might operate at 100% annual capacity factor, but practically because of the necessity for an annual maintenance period the maximum annual capacity factor is much lower and usually it does not exceed 80%. Moreover, there is always a decline in the annual capacity factor of thermal power plant with its age.

27.13 COMPONENTS OF HYDROPOWER PLANTS

The various components of hydropower plants include (i) storage or diversion dam; (ii) waterways such as tunnel, power channel or penstocks with forebay and necessary appurtenances such as intake structures, air vent valve, surge tanks, etc.; (iii) power house with turbines, generators and other appurtenances; (iv) switch yard for transmission of power.

Dams. A *dam* is a structure constructed across a river at a suitable site to develop an artificial

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

reservoir for the storage of water and to create head. Dams may be of different types such as earth dam, masonry dam, concrete dam etc. Out of these concrete dams are quite common. In order to control the supply of water from the storage reservoir certain gates and valves are used. The different types of gates used are plain sliding gates, wheeled or roller gates etc. Similarly the common types of valves used are butterfly and needle valves.

Waterways. A *waterway* is a passage through which water is carried from the storage reservoir to the power house. It may consist of a tunnel, channel or penstock. If a hill intervenes the reservoir and the power house then a tunnel may be driven to provide a waterway. The tunnel may be circular or horse shoe shaped, lined or unlined, lined with concrete or reinforced concrete or steel depending upon the nature of rock through which it is driven. It may flow full as a pressure conduit or partly full as an open channel.

Penstocks. *Penstocks* are the pipes of large diameter, used as waterways for conveying water from the reservoir to the power house. These are usually made of steel or reinforced concrete. Cast iron or wooden penstocks are also used for low heads and relatively small developments. The thickness of the penstock is determined on the basis on the magnitude of stresses developed due to static and water hammer loads. Usually a penstock is supported on short piers but when it is laid along slopes or encounters change in alignment, it should be supported on anchor blocks. Air valves are provided at the sections of the penstocks where there are steep changes in the gradient. In the case of long steel penstocks, expansion joints should be provided to take care of expansion and contraction due to changes in temperature.

Alternatively an open channel to be followed by penstocks along the slopes may also be the choice as waterway. The power channel may be rectangular, trapezoidal or circular and may be lined or unlined.

Forebay. A *forebay* is an enlarged body of water provided just in front of the penstocks. The main function of the forebay is to provide a small balancing storage upstream of the power house to store temporarily the water rejected by the plant when the load is reduced and to meet the instantaneous increased demand when the load is instantaneously increased. When water is carried through a channel, the forebay may be developed by enlarging the channel just upstream of the intake for the penstocks leading water to the turbines in the power house. However, if the power house is located close to the dam then since the penstocks directly take water from the reservoir, the reservoir itself will act as forebay. The forebay is also provided with a spilling arrangement in order to store water only upto certain maximum level.

Intake Structure. The water from the reservoir or forebay is let into the penstocks through intake structure. The main components of an intake structure are trash racks and gates. A *trash rack* is a grating made of a series of steel bars of rectangular cross-section set parallel to each other and placed across the entire intake opening in an inclined position. It is provided to prevent the debris entering into the water passage of the hydropower plant which may otherwise damage the plant equipment. A debris cleaning device is usually fitted on the trash rack. Further if ice may get deposited on the trash rack, a heating element or some other ice removing equipment is also provided. The trash racks are usually located ahead of the gates. The gates along with their hoisting arrangements are provided to control the entry of water into the penstocks. Just at the back of the gates at the entrance to the penstocks air inlets are provided in order to supply air to the penstocks when the head gates are closed and the turbine gates are open. This is so because if the air inlets are not provided the sudden drainage of the penstocks (when the head gates are closed and turbine gates are opened) may lead to the collapse of the penstocks.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Surge Tanks. As explained in Chapter 21 a surge tank is cylindrical open-topped storage reservoir which is provided to protect the penstocks against water hammer pressures. It however provides protection for only that portion of the penstock which lies on the upstream of it. As such a surge tank should be provided as close to the turbine as possible. A detailed discussion about the surge tank is given in Chapter 21.

Power House. A power house of a hydroelectric scheme houses the various hydraulic and electric equipments. The various hydraulic equipments are turbines, gates or gate valves, governors etc. The various electrical equipments include generators, transformers, switching equipment, transmission lines and transmission structures, auxiliary electrical equipments etc. A switch yard for the transmission of power is usually located outdoors near the power house.

ILLUSTRATIVE EXAMPLES

Example 27.1. *Two turbo-generators each of capacity 25 000 kW have been installed at a hydel power station. During a certain period the load on the hydel plant varies from 15 000 kW to 40 000 kW. Calculate (i) the total installed capacity, (ii) the load factor, (iii) the plant factor, and (iv) the utilization factor.*

Solution

(i) The total installed capacity of the hydel plant

$$= 2 \times 25\,000 = 50\,000 = 50\,000 \text{ kW}.$$

(ii) $$\text{Load factor} = \frac{\text{Average load}}{\text{Peak load}}$$

$$\text{Average load} = \left(\frac{15000 + 40000}{2}\right)$$

$$= 27\,500 \text{ kW}$$

$$\text{Peak load} = 40\,000 \text{ kW}$$

$$\therefore \quad \text{Load factor} = \frac{27500}{40000} = 0.6875 \text{ or } 68.75\%.$$

(iii) $$\text{Plant factor} = \frac{\text{Energy actually produced}}{\text{Max. energy which can be produced}}$$

If the time under consideration is T hours then the total energy actually produced during the time

$$= \left(\frac{15000 + 40000}{2}\right) T \text{ kWh}$$

$$= 27\,500\,T \text{ kWh}$$

Further the maximum energy which would have been produced during this time

$$= 2 \times 25\,000 \times T = 50\,000\,T \text{ kWh}$$

$$\therefore \quad \text{Plant factor} = \frac{27500\,T}{50000\,T} = 0.55 \text{ or } 55\%$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iv) Utilization factor $= \dfrac{\text{Max. power developed}}{\text{Total power that can be developed}}$

$= \dfrac{40000}{50000} = 0.80$ or 80%.

Example 27.2. *A run-of-river hydroelectric power station is proposed across a river at a site where a net head of 25 m is available on the turbine. The river carries a sustained minimum flow of 30 m^3/sec in dry weather and behind the power station sufficient pondage is provided to supply daily peak load of demand with a load factor of 71%. Assuming the plant efficiency of 56.4%, determine the maximum generating capacity of the generator to be installed at the power house. If the daily load pattern indicates $21\frac{1}{2}$ hours average load and $2\frac{1}{2}$ hours of peak load, determine the volume of pondage to be provided to supply the daily demand.*

Solution

The average power developed corresponding to the dry weather flow is given by Eq. 27.1 as

$P = 9.81Q\,(H - h_f)\,\eta$

$= 9.81 \times 30 \times 25 \times 0.564 = 4\,150$ kW

Load factor $= \dfrac{\text{Average load}}{\text{Peak load}}$

$\therefore$ Peak load $= \dfrac{4150}{0.71} = 5\,845$ kW

$\therefore$ The maximum capacity of the generators to be installed

$= 5\,845$ kW

During $2\frac{1}{2}$ hours of peak load the excess power to be developed from the pondage

$= (5\,845 - 4\,150) = 1\,695$ kW

The discharge required for developing the excess power during the peak demand is

$Q = \dfrac{1695}{9.81 \times 25 \times 0.564} = 12.25\ m^3/s$

$\therefore$ Pondage $= (12.25 \times 2.5 \times 3\,600)$

$= 11 \times 10^4\ m^3 = 11$ hectare-metre

Example 27.3. *The following data gives the average monthly flow in cumec in a dry year and it is intended to use it for designing a hydropower plant : July 17.0, Aug. 18.4, Sept. 17.6, Oct. 13.6, Nov. 13.3, Dec. 11.3, Jan. 9.9, Feb. 9.6, Mar. 8.5, Apr. 11.8, May 12.7, June 14.2.*

Assuming the net head on the turbine as 91.5 m and efficiency 80% draw the power duration curve. It is intended to develop electric power at a firm rate of 9 000 kW either by providing a storage or by providing standby diesel unit and no storage. In the two cases, determine the minimum capacities of the reservoir or the diesel unit.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Solution

Table 27.3 Shows the computation involved for the plotting of the power duration curve.

TABLE 27.3

Month	*Flow (m^3/sec)*	*Flow arranged in ascending order*	*Power = 9.81Q ($H–h_f$)η (kW)*	*% of time of availability*
July	17.0	8.5	6 104	100.0
Aug.	18.4	9.6	6 894	91.7
Sept.	17.6	9.9	7 109	83.3
Oct.	13.6	11.3	8 114	75.0
Nov.	13.3	11.8	8 473	66.7
Dec.	11.3	12.7	9 120	58.3
Jan.	9.9	13.3	9 551	50.0
Feb.	9.6	13.6	9 766	41.7
March	8.5	14.2	10 197	33.3
April	11.8	17.0	12 208	25.0
May	12.7	17.6	12 638	16.7
June	14.2	18.4	13 213	8.3

The power duration curve is plotted as shown Fig. Ex. 27.3.

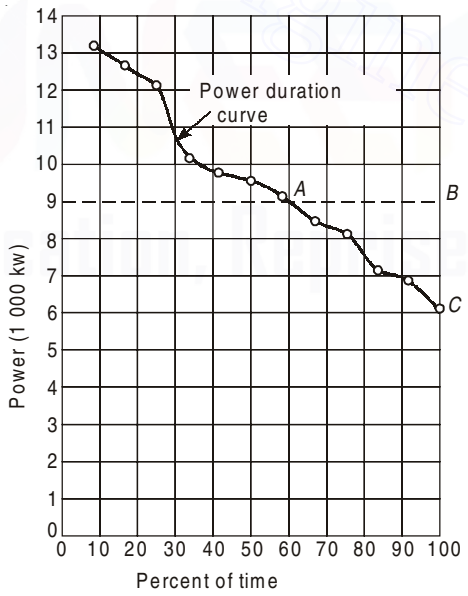


Figure Ex. 27.3

As seen from the figure, in order to obtain a firm power at the rate of 9 000 kW, the power corresponding to area *ABC* will be required to be developed from the storage. Further it is seen that for

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

60% of the time the power developed from the natural flow in the stream in an excess of the required power. But for the 40% of the time the natural flow will be required to be supplemented from the storage to develop the required power.

The area of the portion *ABC* is obtained by counting the squares as 565 mm^2 which is equivalent to

$$= \frac{565\times1000\times10\times12}{10\times10\times100} \text{ kW-month}$$

$$= 6\,780 \text{ kW-month}$$

The storage required $$= \frac{6780\times30\times24\times3600}{9.81\times91.5\times0.80}$$

$$= 2.45\times 107 \text{ m}^3$$

$$= 2450 \text{ hectare-metre}$$

With no storage the firm power available

$$= 6\,104 \text{ kW}$$

$\therefore$ The capacity of the diesel unit required

$$= (9\,000 - 6104)$$

$$= 2\,896 \text{ kW say } 2900 \text{ kW}$$

Example 27.4. *The water turbine of a hydel storage scheme has an output of 7 000 kW, when working under a head of 25 m and with an overall efficiency of 80%. The inflow in the reservoir during a year is as given below:*

Month :	*J*	*F*	*M*	*A*	*M*	*J*	*J*	*A*	*S*	*O*	*N*	*D*
Inflow :												
in 10^6 m^3	*102*	*108*	*90*	*81*	*72*	*87*	*120*	*90*	*93*	*99*	*105*	*96*

Find (i) the minimum capacity of reservoir to satisfy the uniform demand of water, (ii) the total quantity of water wasted during the year. Use graphical or analytical method. Assume that the reservoir is full at the beginning of January.

Solution

The discharge required for the turbine

$$= \frac{7000\times10^3}{9810\times25\times0.80} = 36 \text{ m}^3/\text{s}$$

$\therefore$ Flow required for the complete month

$$= (36 \times 30 \times 24 \times 3\,600) \text{ m}^3$$

$$= 93\times10^6 \text{ m}^3$$

From the following table it is seen that the greatest reservoir depletion is 42 $\times10^6$ m^3 and hence the minimum capacity of the reservoir required to satisfy the uniform demand of water

$$= 42\times 10^6 \text{ m}^3$$

The total quantity of water wasted during the year

$$= 27 \times10^6 \text{ m}^3$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The analytical solution is given in the following Table:

Month	*J*	*F*	*M*	*A*	*M*	*J*	*J*	*A*	*S*	*O*	*N*	*D*
Inflow in 10^6 m^3	102	108	90	81	72	87	120	90	93	99	105	96
Flow thro turbine in 10^6 m^3	93	93	93	93	93	93	93	93	93	93	93	93
Draft from storage in 10^6 m^3	—	—	3	12	21	6	—	3	—	—	—	—
Storage refill in 10^6 m^3	—	—	—	—	—	—	27	—	—	6	12	—
Reservoir depletion in 10^6 m^3	—	—	3	15	36	42	15	18	18	12	—	—
Water Wasted in 10^6 m^3	9	15	—	—	—	—	—	—	—	—	—	3

Example 27.5. *During a low water week a river has an average daily flow of 30 m^3/sec with a fluctuation during the day requiring a pondage capacity of approximately 20% of the daily discharge. A hydroelectric plant is to be located on the river which will operate 5 days a week, 24 hours a day, but will supply power at a varying rate such that the daily load factor is 50%, corresponding to which the pondage required is equal to 0.2 times the mean flow to the turbine. On Saturdays and Sundays all the flow is ponded for use on rest of the days.*

If the effective head on the turbines when the pond is full is to be 20 m and the maximum allowable fluctuation in pond level is 1 m, find

(a) the surface area of the pond to satisfy all the operating conditions.

(b) the weekly output at the switch board in kWh. Assume turbine efficiency 85% and generator efficiency 94%.

Solution

Average daily flow in river = 30 m^3/s

$\therefore$ Total flow per day

$$= (30 \times 24 \times 3\,600)\ \text{m}^3$$
$$= 2.592 \times 10^6\ \text{m}^3$$

Total inflow on Saturday and Sunday

$$= (2 \times 2.592 \times 10^6)\ \text{m}^3$$
$$= 5.184 \times 10^6\ \text{m}^3$$

This is to be used on the rest of the five days and hence the amount used daily from this pondage

$$= \frac{5.184 \times 10^6}{5} = 1.037 \times 10^3\ \text{m}^3$$

The corresponding additional flow to the turbine

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$= \frac{1.037\times10^6}{24\times3600} = 12 \text{ m}^3/\text{s}$$

∴ Total average flow to the turbine

$$= (30 + 12) = 42 \text{ m}^3/\text{s}$$

It is given that the fluctuations in the flow of river require a pondage capacity of approximately 20% of the daily discharge. Hence additional pondage required on this account

$$= \left(\frac{30\times20}{100}\right)\times 24 \times 3\,600 = 0.518 \times 10^6 \text{ m}^3$$

The pondage required on account of variation in the load

$$= (0.2 \times 42 \times 24 \times 3\,600) = 0.726 \times 10^6 \text{ m}^3$$

Hence the total pondage required

$$= (5.184 + 0.518 + 0.726) \times 10^6 \text{ m}^3$$
$$= 6.428 \times 10^6 \text{ m}^3$$

Since maximum allowable fluctuation in the pond level = 1 m

∴ Surface area of pond = 6.428×10^6 m^2

With a fluctuation of 1 m the average head on the turbine

$$= \left(\frac{20+19}{2}\right) = 19.5 \text{ m}$$

∴ Maximum power generated is

$$P = (9.81 \times 42 \times 19.5 \times 0.85 \times 0.94) \text{ kW}$$
$$= 6.42 \times 10^3 \text{ kW}$$

The weekly output at the switch board is

$$E = [6.42 \times 10^3 \times (5 \times 24)] \times \frac{50}{100} \text{ kWh}$$
$$= 3.852 \times 10^5 \text{ kWh}$$

Example 27.6. *A run-of-river hydel power plant with an installed capacity of 15 000 kW operates at 20% load factor when it serves as a peak load station. What should be the minimum discharge in the stream so that it may serve as the base load station ? The plant efficiency may be taken as 80% when working under a head of 15 m. Also calculate the maximum load factor of the plant when the discharge in the stream is 30 m^3/s.*

Solution

When the plant operates as a peak load station with a load factor of 20%, the average power developed

$$= \frac{15000\times20}{100} = 3 \times 10^3 \text{ kW}$$

If Q is the minimum flow necessary for the plant to run as a base load plant, the power developed is

$$P = 9.81\, Q\, (H - h_f)\eta$$
$$= (9.81 \times Q \times 15 \times 0.80) = (117.72Q) \text{ kW}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Equating the two, we get

$$117.72Q = 3 \times 10^3$$

$$\therefore \quad Q = \frac{3\times10^3}{117.72} = 25.48 \text{ m}^3/\text{s}$$

If the stream flow is 30 m^3/s then the power developed is

$$P = (9.81 \times 30 \times 15 \times 0.80)$$
$$= 3.532 \times 10^3 \text{ kW}$$

$\therefore$ Maximum load factor

$$= \frac{3.532\times10^3}{15000}$$
$$= 0.2355 \text{ or } 23.55\%$$

Example 27.7. *A run-of-river plant is installed on a river having a minimum flow of 15 m^3/s. If the plant is used as a peak load plant operating only for 6 hours a day, determine the firm capacity of the plant (a) without pondage, (b) with pondage but allowing 10% of the water to be lost in evaporation and other losses. Head at the plant is 10 m and the plant efficiency may be assumed as 85%.*

Solution

(a) Without pondage the firm capacity

$$= (9.81 \times 15 \times 10 \times 0.85) = 1.251\times 10^3 \text{ kW}$$

(b) With pondage the total volume of water stored during the 18 hours when the plant is not operating

$$= (15 \times 18 \times 3\,600) = 9.72 \times 10^5 \text{ m}^3$$

The loss of water due to evaporation etc.,

$$= (9.72 \times 10^5) \frac{10}{100} = 0.972 \times 10^5 \text{ m}^3$$

$\therefore$ Net amount of water available from the pondage for the generation of the power during 6 hours when the plant is operating

$$= (9.72 - 0.972) \times 10^5 = 8.748 \times 10^5 \text{ m}^3$$

The corresponding rate of flow available from the pondage

$$= \frac{8.748\times10^5}{6\times3600} = 40.5 \text{ m}^3/\text{s}$$

$\therefore$ Total flow available for the power generation

$$= (15 + 40.5) = 55.5 \text{ m}^3/\text{s}$$

The firm capacity of the plant

$$= (9.81 \times 55.5 \times 10 \times 0.85)$$
$$= 4.628 \times 10^3 \text{ kW}.$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

SUMMARY OF MAIN POINTS

1. Water power or hydro power is one of the major sources of energy.
2. If Q is the rate of flow of water that may be continuously made available from any stream and it possess head H, then the amount of hydropower in kW that would be available is given as

$$P = \frac{wQ\left(H - h_f\right) \times \eta}{1000}$$

or $$P = 9.81Q\,(H - h_f) \times \eta$$

where

$(H - h_f)$ = net or effective head on the turbine; and
η = efficiency of the hydropower station.

3. Storage may be defined as impounding of a considerable amount of excess runoff during seasons of surplus flow for use during dry seasons.
4. Pondage may be defind as a regulating body of water in the form of a relatively small pond or reservoir provided just at the hydropower plant.
5. The flow duration curve consists of a plot of values of stream flow in increasing order of magnitude as ordinate and the percent of time under consideration the flow equalled or exceeded, as abscissa.
6. A mass curve is a plot of accumulated flow against time. It is commonly used for the storge computations.
7. The hydropower plants may be classified according to the storage being provided or not as
   (i) Run-of-river plants
   (ii) Reservoir plants
   (iii) Pumped storage plants
   (iv) Tidal plants.
8. The hydropower plants may be classified on a functional basis as
   (i) Base-load plants
   (ii) Peak-load plants
9. The minimum power which can be generated throughout the year from a hydroelectric plant is called its *firm* or *primary power.*
   The power in addition to the firm power which would be generated for only a part of the year is known as *secondary* or *non-firm power.*
10. *Load factor* is defined as the ratio of the average load during a certain period to the maximum or peak load during that period.
11. *Utilization factor* is defined as the ratio of the peak load developed during a certain period to the installed capacity of the plant.
12. *Capacity factor* is defined as the ratio of the energy that the plant actually produces during any period to the energy that it might have produced if operated at full capacity.

PROBLEMS

27.1 What are the different types of hydropower plants ? Describe each one briefly.

27.2 Write a brief note on the relative economics of a hydel plant and a thermal plant for power generation.

27.3 What do you understand by 'flow duration curve' ? How is it prepared ?

27.4 What is a 'mass curve' ? Explain the procedure for preparing a mass curve and also its uses.

27.5 Differentiate between storage and pondage. Support your answer with a neat sketch.

27.6 Explain briefly how can the power available for a hydel project be estimated.

27.7 Explain the terms-load factor, plant factor and utilization factor. Derive a relation between these factors.

27.8 Distinguish between a base-load power plant and a peak-load power plant.

27.9 Differentiate between firm power and secondary power.

27.10 Make a neat sketch of a hydropower plant and show clearly the various elements.

27.11 Show that capacity factor is equal to the product of the load factor and the utilization factor.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Fluvial Hydraulic

Chapter 28

28.1 INTRODUCTION

The channels may be classified as *rigid boundary channels* and *mobile boundary channels.* The rigid boundary channels are those which have their boundaries (i.e., bed and banks) made up of nonerodible material and hence these channels have fixed boundary configuration. Further the resistance offered to the water flowing in a rigid boundary channel depends only on the nature of the boundary surface and it can be determined upto a fair degree of accuracy as indicated in Chapter 15. On the other hand, the moblie boundary channels include rivers and unlined alluvial canals, the boundaries of which are made up of loose soil which can be easily eroded and transported by the flowing water. As such the boundary configuration of these channels tends to change. On account of varying boundary configuration in the case of mobile boundary channels the resistance to the flowing water depends not only on the nature of the boundary surface but also on the condition of the bed and the banks of the channel. The study of flow of water in mobile boundary channels is dealt with in fluvial hydraulics.

28.2 SEDIMENT TRANSPORT IN CHANNELS

On account of erosion in the catchment, the rivers along with water receive a huge quantity of sediment which comprises soil particles of various sizes ranging from fine silt to coarse sand. A portion of this sediment is also received by the channels which draw water directly from the rivers. In the case of rigid boundary channel the quantity of sediment transported by it remains more or less same as that entering the channels at its off taking section. Moreover, in such channels relatively higher velocities of flow may be permitted, so that the sediment is not allowed to get deposited. As such for rigid boundary channels the problem of sediment transport does not exist. However, in the case of unlined alluvial channel the quantity of sediment transported may vary from section to section due to scouring of bed and banks of the channel as well as due to silting (or deposition of sediment) at any section, the variation in the quantity of sediment transported by a channel may result in modifying the cross-section of the channel. Thus for unlined alluvial channels the main problem associated with the flow of water is that of the sediment transported along with the water. The problem of sediment transport is of major practical importance in the design of unlined irrigation canals or drainage canals, in the design of which efforts are made to develop such a channel section which would be non-silting non-scouring type so that it would be a stable channel. Evidently in order to design such a stable channel section the knowledge of fundamental mechanism of sediment transportation is of prime importance. Further another important aspect of the problem of sediment transport is to determine the condition of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the bed surface of an alluvial stream or channel which might be developed as the sediment characteristics, and fluid and flow characteristics are changed. As stated earlier the condition of the bed considerably affects the resistance to the flow of water in unlined alluvial channels, the determination of the same would greatly assist the estimation of the resistance to the flow of water in such channels. Some of these aspects of the problem of sediment transport are discussed in the following sections of this chapter.

28.3 SEDIMENT PROPERTIES

The entrainment, transportation and deposition of sediment depend upon the properties of the sediment as well as the flow. The various properties of the sediment may be grouped under two headings:

(a) Individual grain properties

(b) Bulk properties

The most important properties of the individual particles of the sediment transported are (i) size, (ii) shape, and (iii) specific gravity. The bulk properties of the sediment transported include (a) the grain size distribution, (b) specific weight or bulk unit weight, and (c) porosity.

Size. The size of the sediment particles is determined either by *sieving* or by *sedimentation*. By sieving it is possible to obtain the size of the particles upto 1/16 mm, because this is the size of the finest screen which can be used for accurate size determination. Since silt sizes lie below the sieve range indicated above, size determinations must be made microscopically or indirectly by the method of sedimentation. In sedimentation method the size of a particle is determined by considering a hypothetical sphere of quartz which will have the same fall velocity as that of the sediment particle in settling down in water, and then the size of the particles is given by the diameter of the sphere defined as *sedimentation diameter*.

For perfect spheres of diameter d the velocity of fall V at low values of Reynolds number $\left(\text{Re} = \frac{Vd}{\upsilon} < 0.1\right)$ may be determined analytically from the Stokes' law as

$$V = \frac{\gamma_w}{18}\frac{d^2}{\mu}\left(\frac{\gamma_s - \gamma_w}{\gamma_w}\right)$$

where γ_s is specific weight of the solid; γ_w is specific weight of water; and μ is absolute viscosity of water.

For higher Reynolds number the following expression may be used

$$V^2 = \frac{4}{3}\frac{gd}{C_D}\left(\frac{\gamma_s - \gamma_w}{\gamma_w}\right)$$

in which C_D is drag coefficient, the value of which may be determined from C_D v/s Re curve for sphere as shown in Fig. 18.6 in Chapter 18. The values of the fall velocity given by the above expressions correspond to a single sphere falling in an infinite extent of fluid, and the same will therefore be required to be corrected for the finite extent of the fluid and also for the effect of the concentration of the particles.

Shape. In sediment analysis the most pertinent shape parameter is known as sphericity. *Sphericity* is defined as the ratio of the surface area of a sphere having the same volume as the particle to the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

surface area of the particle. Thus sphericity indicates the deviation of the shape of the grain from that of the sphere.

Because of practical difficulty of measuring sphericity, the particle is usually represented by another shape parameter known as *shape factor*. In this, the particle is usually represented by an ellipsoid of the same proportions. The shape factor is then defined by $\left(a/\sqrt{bc}\right)$, where a, b, c, represent the three principal axes of such an ellipsoid. For a sphere and a cube shape factor is equal to unity.

In addition to the above noted shape factors another factor which is also used for defining the shape of the sediment particles is *roundness*. Roundness is defined as the ratio of the average radius of curvature of the edges to the radius of the largest circle that can be inscribed within the projected area of the grain. Roundness indicates the wear to which the particle has been subjected during relative movement of the sediment and the water.

Specific Gravity. Since practically all stream sediment has its origin in the disintegration of the parent rock, essentially all constituents of the original rock are found in the sediment transported by the stream. The composition of sediment ranges from silt which is relatively inert silica to sand which is almost pure silica or quartz. The specific gravity of quartz is 2.65 and this is generally used as characteristics specific gravity of the sediment.

Grain Size Distribution. Because sediment consists of innumerable particles differing in size, shape, specific gravity, and velocity of fall it is necessary to make statistical analysis of these characteristics, as determined from an adequate number of samples, in order to define fully the representative particle characteristics of any sediment as a whole.

A sample of sediment is usually divided into certain class interval according to size, and the percentage of the total in each class (according to number, volume or usually weight) is ascertained. By plotting the percentage against the diameter, frequency distribution curve of the sample is obtained. Usually a cumulative frequency curve is prepared which is obtained by plotting the percentage of sediment (by number, volume or usually by weight) equal to or larger than the corresponding size of sediment. The cumulative frequency curve is in fact, the summation of the frequency distribution curve. The abscissa indicating particle size is plotted on logarithmic or natural scale while the percentage is plotted either on natural or probability scale. With the help of this curve, the composition of the sample may be defined in terms of the parameters such as mean diameter, median diameter, standard deviation from the mean, skewness etc. However, instead of using these parameters, many research workers have introduced other characteristic diameters from the cumulative curve to describe the characteristic of the sample. These characteristic diameters are corresponding to a certain percentage in the cumulative curve such as d_{20}, d_{50}, d_{80} etc., meaning that 20%, 50% or 80% of the particles are equal to or larger than this size.

The other bulk properties of the sediment are its specific weight and porosity. The specific weight is defined as the dry weight per unit volume of the sediment in place. The porosity is a measure of the volume of voids per unit volume of the sediment in place. Both these properties are pertinent to hydraulic engineering studies.

28.4 MODES OF SEDIMENT MOVEMENT

Depending on the shape, size and composition of the sediment particles and the velocity of the flowing water the sediment particles are transported in one or more of the following modes:

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) Sliding or Rolling along the Bed. If the particle is relatively round it will roll while if it is cubical it will overturn or slide.

(b) Saltation or Skipping or moving along in small jumps. The height of each jump being of the order of 1 or 2 grain diameters and the length of the jump being 3 to 5 grain diameters. Saltation depends on the contact of the particle with the bed, since it is the reaction from the bed that will cause the impulse necessary for the jumping of the particles.

(c) Movement in Suspension without contact with the bed. In this state of movement the grain moves freely through the liquid and is not dependent on the support from the bed. This concerns the movement of the finer particles which are supported and kept in the state of suspension by the turbulent eddies against gravity.

28.5 TYPES OF SEDIMENT LOAD

The sediment transported by a stream or channel may be divided into the following three categories:

(a) Bed load

(b) Suspended load

(c) Wash load.

Bed load is defined as that portion of the sediment which moves on or near the bed of the stream or channel. On the other hand *suspended load* is that portion of the sediment which is moving in suspension in the flowing water, being kept in suspension by the turbulent eddies. It also includes the finest particles which being colloidal would not settle even in still water. Thus a suspended grain moves at the same forward velocity as the filament of water in which it is travelling, so that there is no drag between the two. A grain of bed load, on the other hand, moves at a lower velocity than the layer in which it is travelling and on it a drawing force is exerted through fluid drag.

Wash load is used to designate the portion of the rather fine suspended particles which do not originate from the river bed but from the head water area (or catchment area) owing to soil erosion. It was distinguished from the other by Einstein and it consists of colloidal materials. It is supported by molecular movement and it does not depend on velocity or turbulence or turbulent eddies.

28.6 INITIATION OF SEDIMENT MOTION

Many attempts have been made to study the conditions of initiation of the movement of sediment. In an early approach it was considered that the sediment particles would just start moving only when the velocity of flow reached a certain minimum value. This velocity of flow is termed as 'critical or pick-up velocity', the value of which would be different for different types of materials. The velocity of flow at which the motion of the sediment particles just start has been related to the sediment size.

A. Brahms and W. Airy tried to solve analytically the problem of initiation of grain movement by water. According to their theory, the sediment particle begins to move when the drag exerted on it by the moving water is sufficient to overcome the frictional resistance between the particles and the bed. Thus following expression for the critical velocity V_{cr}, may be obtained on the basis of their theoretical consideration,

$$V_{cr}^2 = \frac{4}{3}\frac{dg}{\alpha C_D}(s-1)\tan\phi \qquad ...(28.1)$$

In which d is grain diameter, g is gravitational acceleration, α is a shape coefficient, C_D is drag coefficient, s is specific gravity of sediment, and ϕ is the angle of friction. From Eq. 28.1 it is seen that critical

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

velocity is proportional to square root of the grain diameter. Since the volume and hence the weight W of the grain is proportional to the cube of the grain diameter, it follows that

$$V_{cr} \propto W^{1/6}; \text{ or } V_{cr} = kW^{1/6} \qquad \ldots(28.2)$$

However, the above approach has several limitations as indicated. The value of C_D is not constant and it cannot be easily determined. Further the determination of the value of the critical velocity is difficult and also the ratio (V_{cr}/V_{mean}) may not be constant. From this it may be concluded that velocity is not a perfect parameter to define the initiation of sediment movement.

(*a*) **Jeffreys' lift concept.** In 1929 Jeffreys indicated that on account of difference in velocities at the top and bottom of a grain lift is exerted on it, which tends to initiate the motion of the sediment particles. Assuming a cylindrical shape for the particle the dynamic uplift force per unit length is equal to

$$\left[\pi\rho_w r V^2\left(\frac{1}{3}+\frac{1}{9}\pi^2\right)\right]$$

where ρ_w is mass density of water, r is the radius of the cylinder and V is the uniform stream velocity. If the uplift force exceeds the submerged weight of the particle then the condition for the particle to be lifted up is

$$\pi\rho_w r V^2\left(\frac{1}{3}+\frac{1}{9}\pi^2\right) > \pi r^2(\rho_s-\rho_w)g$$

or

$$\left(\frac{1}{3}+\frac{1}{9}\pi^2\right)V^2 > \left(\frac{\rho_s-\rho_w}{\rho_w}\right)gr \qquad \ldots(28.3)$$

where ρ_s is the mass density of the sediment particles.

For sand of 0.6 mm diameter with $[(\rho_s-\rho_w)/\rho_w] = 1.63$, the value of V comes to 57.92 mm/s. Thus according to this consideration the velocity of this magnitude at grain level should be able to provide enough lift to neutralize its weight. The grain could then move along with the fluid without the assistance of drag.

However, this approach also has several limitations as indicated. The above derived condition is applicable only for a two dimensional object lying in an ideal fluid. But in actual practice the sediment particles are three dimensional objects and also the fluid is not ideal. Moreover, there exists a controversy about the consideration of the lift and drag in the initiation of the motion of the sediment particles.

Another approach developed to explain the initiation of the sediment motion is the tractive force approach which is indicated below.

The tractive force is the force exerted by the flowing water on the sediment particles, thereby causing their motion. The tractive force per unit area will be equal to the boundary shear stress τ_0. The tractive force required to initiate general movement of grains is called *critical tractive force.* Several investigators have developed expressions for the critical tractive force per unit area τ_{cr}. However, an important development in the tractive force approach was made by Shields in 1936, which is discussed below.

(*b*) **Shields tractive force approach.** According to Shields, relative magnitude of tractive force and resistance to motion of uniform sediment grains, expressed by the dimensionless ratio $\left[\frac{\tau_0}{(\gamma_s-\gamma_w)d}\right]$, represented one significant parameter in the bed load movement, in which γ_s and γ_w are the specific

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

weights of the sediment and water respectively and d is the grain diameter. The dimensionless ratio is termed as the *entrainment function.* This was plotted as a function of the ratio of the grain diameter and the laminar sublayer thickness, (d/δ'), which was shown to be the only other significant parameter for initiating grain movement on a level bed, besides unknown factors for shape and grading. This is so because according to Shields the magnitude of tractive stress on a grain will depend on the fact whether the grain is outside or inside the laminar sublayer. Thus if $(d/\delta') < 1$, the grain is covered within the laminar sublayer and hence the tractive stress on the grain will depend only on the viscous or laminar flow within the sublayer. On the other hand if $(d/\delta') > 1$, the grain projects outside the layer and hence the turbulent flow outside also affects the tractive stress on the grain.

Further, since $\delta' \propto \frac{\upsilon}{V_*}$, $\frac{d}{\delta'} \propto \frac{V_* d}{\upsilon}$ instead of the parameter $\left(\frac{d}{\delta'}\right)$, $\left(\frac{V_* d}{\upsilon}\right)$ may be used which is known as *shear Reynolds number.* By plotting $\left[\frac{\tau_{cr}}{(\gamma_s - \gamma_w)d}\right]$ against $\left(\frac{V_* d}{\upsilon}\right)$ a curve of the form shown in Fig. 28.1 has been obtained. The curve indicates an inverse relationship between the two when the grain is completely enclosed in the laminar sublayer i.e., $\left(\frac{d}{\delta'}\right) < 1$ or $\frac{V_* d}{\upsilon} < 11.6$. It then rises and becomes horizontal at a large value of $\left(\frac{V_* d}{\upsilon}\right)$, i.e., when $\left(\frac{V_* d}{\upsilon}\right) > 600$ or so, when the laminar sublayer is completely disrupted. At that time the value of $\left[\frac{\tau_{cr}}{(\gamma_s - \gamma_w)d}\right]$ becomes constant at about 0.06, thereby indicating that in turbulent flow τ_{cr} is proportional to $(\gamma_s - \gamma_w)d$. However, the studies conducted by later investigators have indicated that the value of this constant is about 0.04 to 0.05.

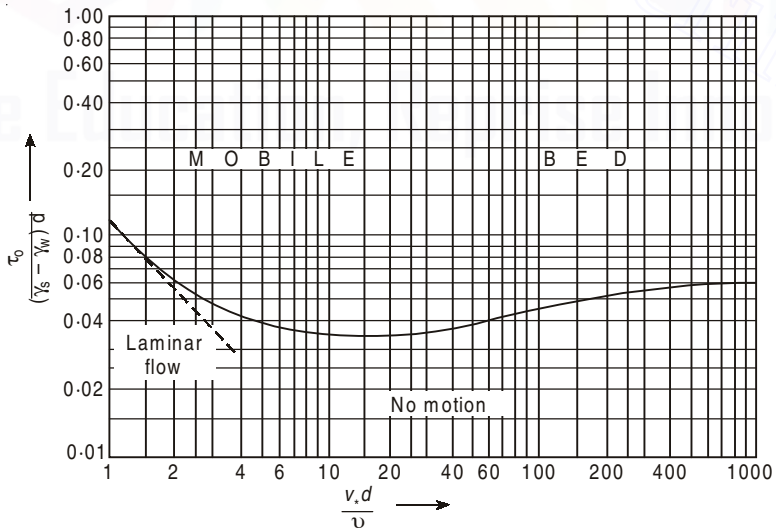


Figure 28.1 *Shields analysis of the entrainment function*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*c*) **White's Approach.** Another notable contribution in this field is that of C.M. White. According to him a distinction has been made between different types of motion viz.,

(i) Steady viscous motion.

(ii) Steady inviscid motion.

(iii) Turbulent motion.

Steady viscous motion is that which has laminar flow in the main stream as well as boundary layer. For this flow the shear Reynolds number $(V_* d/v) < 3.5$ and the critical tractive stress τ_{cr} is given by

$$\tau_{cr} = \alpha \left[\frac{\pi}{6}\eta d(\gamma_s - \gamma_w)\tan\phi\right] \quad ...(28.4)$$

where η is a packing coefficient the value of which for natural or random packing is constant equal to 0.6, α is an experimental coefficient introduced to account for the eccentricity of the drag exerted on the grains; and the other notation are same as defined earlier. On the basis of the experimental results White has shown that for $(V_* d/\upsilon) < 3.5$, the value of '$\alpha\eta$' and hence $[\tau_{cr}/(\gamma_s - \gamma_w)d]$ from Eq. 28.4 is independent of $(V_* d/\upsilon)$. The average value of '$\alpha\eta$' is 0.36 and is more or less constant.

This however contradicts the earlier results of Shields.

The steady inviscid motion as defined by White is that in which the boundary layer is turbulent but the main stream has laminar flow. While in the turbulent motion both the boundary layer and the stream flow are in turbulent state. For both these flows $(V_* d/\upsilon) > 3.5$.

For $V_* d/\upsilon > 3.5$ the critical tractive stress is given by the following expression

$$\tau_{cr} = \left[\frac{\pi}{6}\eta d(\gamma_s - \gamma_w)\tan\phi\right] \quad ...(28.5)$$

For steady inviscid motion the experimental observations indicated that the observed value of the critical tractive force was nearly half of that given by Eq. 28.5. The reasons for this discrepancy is that whereas the value of the shear stress τ_{cr} given by Eq. 28.5 is the instantaneous value, the experimentally observed value of τ_{cr} is an average value, and the instantaneous value of shear at incipient movement is twice the average value. Thus for this case the actual value of the critical tractive stress is given by Eq. 28.5 with a turbulence factor (or coefficient) = $\frac{1}{2}$.

For the third case of turbulent motion, no experiments were conducted by White. However, a comparison of the experimental data of other investigators indicated that the stress required to initiate motion of sediment particles in this case was about $\frac{1}{4}$ of that given by Eq. 28.5. It was thus shown that in this case the instantaneous stress value could reach four times the average value and Eq. 28.5 gives only the instantaneous value of the critical tractive stress. Hence the actual value of the critical tractive stress for this case is given by Eq. 28.5 with a turbulence factor = $\frac{1}{4}$.

For the usual turbulent flow conditions for quartz grains in water the critical tractive stress is approximately given by

$$\tau_{cr} = 785.8d \quad ...(28.5\text{ a})$$

where τ_{cr} is in N/m^2 and d is the grain diameter in metres.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

28.7 BED DEFORMATIONS IN ALLUVIAL STREAMS

The bed of a stream transporting sediment under usual conditions of flow does not remain plane but develops certain undulations on the bed resulting in different types of bed configurations. The various bed configurations that may be formed in an alluvial stream are *ripples, dunes, plane bed, standing wave* and *antidunes,* all of which are shown in Fig. 28.2. These bed configurations and their associated flow phenomena are known as *regimes of flows.*

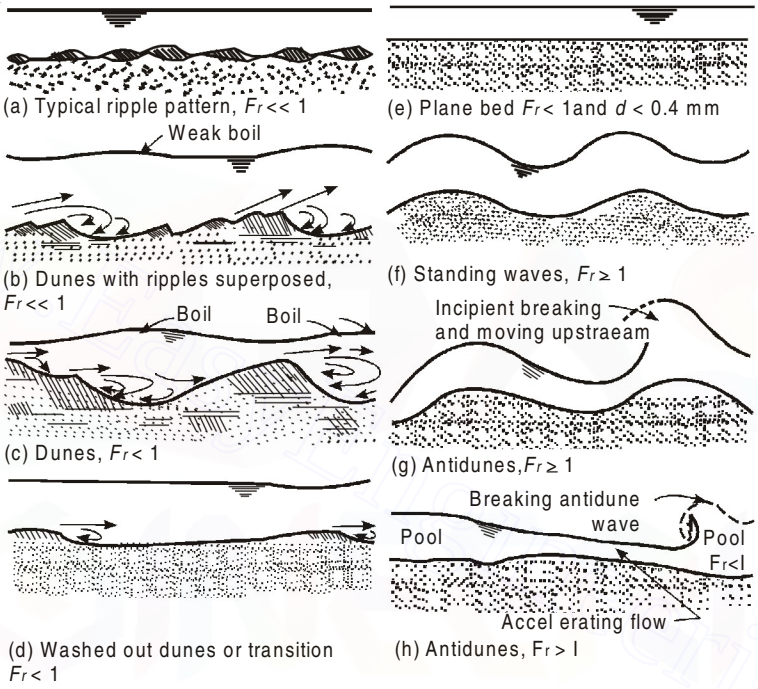


Figure 28.2 *Bed deformation in alluvial streams*

As the flow of water occurs over an initially plane bed of an alluvial stream at relatively low velocities ripples are formed as soon as the general movement of the bed material begins. *Ripples* are small unsymmetrical triangle-shaped elements having gentle upstream slopes and steep downstream slopes. Their length ranges from 120 mm to 600 mm and height ranges from 6 mm to 60 mm. It has however been reported that ripples are formed only when $d_{50} \leq 0.6$ mm and for bed material coarser than this, directly the dunes are formed instead of ripples. With the gradual increase in the velocity of flow over a bed having ripples (or if the bed material is coarser than 0.6 mm over a plane bed), larger bed formations called dunes are formed. *Dunes* are large triangle-shaped elements similar to ripples. Their lengths range from 0.6 m to several metres and their heights range from 30 mm to several centimetres. Dunes cause large separation zones in the flow. The sediment is eroded on the upstream side. This process results in the apparent downstream migration of the dunes at a speed much less than the mean stream velocity. The ripples or dunes or both offer a relatively large resistance of flow as compared with plane bed.

As the mean flow passes to higher Froude number say Fr = 0.5 and if the depth of flow $D < 0.3$ m and the median size of the bed material is less than about 0.4 mm the dunes wash out and a plane bed and

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

plane water surface are developed. However, if the median diameter is larger than 0.4 mm the dunes may be completely washed out only when Fr ≈ 1, and for Fr < 1 the complete plane bed may not be developed but there may be washed out dunes. This corresponds to the transitional regime of flow which has relatively low resistance to flow.

With further increase in shear and Froude number, *standing sand and water waves*, which are in phase and of sinusoidal form, develop and persist upto Fr = 1.2 at which *antidunes* develop. The antidunes are very much similar to standing waves except that they gradually build up from a plane bed and grow in height until they become unstable and break. Both the standing waves as well as antidunes move slowly in the upstream direction, but the sediment is transported in the direction of flow only. Moreover, in this regime of flow the resistance to flow is relatively small but sediment transport is large.

28.8 RESISTANCE TO FLOW IN ALLUVIAL STREAMS

It is obvious that the resistance to flow in alluvial streams varies considerably with the changes in the bed conditions. As such the problem of determining the correct value of the resistance coefficient for alluvial streams with highly deformable beds is quite complex. However, the problem has received the attention of several investigators who have attempted to obtain the values of the resistance coefficient for the flows in the alluvial streams with deformable beds. Some of the earlier notable contributions are those of Meyer-Peter and Muller, Einstein and Barbarossa ; Engelund, Alam and Kennedy etc. Almost all these investigators have adopted an approach in which the boundary resistance is divided into two parts viz., grain roughness and bed form roughness, and the corresponding resistance coefficients are defined either by dividing the bed slope or the hydraulic radius (see Illustrative Example 28.3).

However, apart from the above noted attempts certain other approaches have also been suggested by various other investigators. Amongst these the notable contribution is that of Garde and Ranga Raju which is discussed below : (Refer J.A.S.C.E. Hyd. div. July 1966).

(*a*) **Garde and Ranga Raju Approach.** The resistance to flow of water in alluvial stream may be expressed in terms of dimensionless parameters as

$$\frac{V}{\sqrt{\frac{\Delta\gamma_s}{\rho_w} d}} = \phi_1 \left[\frac{R}{d}, \frac{S}{(\Delta\gamma_s / \gamma_w)}, \frac{g^{1/2} d^{3/2}}{\upsilon} \right] \quad \ldots(28.6)$$

in which V = mean velocity of flow, $\Delta\gamma_s = (\gamma_s - \gamma_w)$, $d = d_{50}$ = median sediment size for which 50% of the material by weight is finer, ρ_w = mass density of water, S = bed slope of the stream, R = hydraulic radius, υ = kinematic viscosity of water, g = gravitational acceleration. It has been stated that since viscous effects will partly govern the resistance for the plane bed regime with fine bed material and the same being not very well known, it may be neglected. Thus the viscous term ($g^{1/2} d^{3/2} / \upsilon$) is omitted from the Eq. 28.6, which then becomes

$$\frac{V}{\sqrt{\frac{\Delta\gamma_s}{\rho_w} d}} = \phi_2 \left[\frac{R}{d}, \frac{S}{(\Delta\gamma_s / \gamma_w)} \right] \quad \ldots(28.7)$$

By analysing extensive flume and field data these investigators have established a criteria for

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

determining the regime of flow. It has been represented in terms of a graphical plot of $\left(\frac{S}{\Delta\gamma_s/\gamma_w}\right)$ against (R/d) as shown in Fig. 28.3.

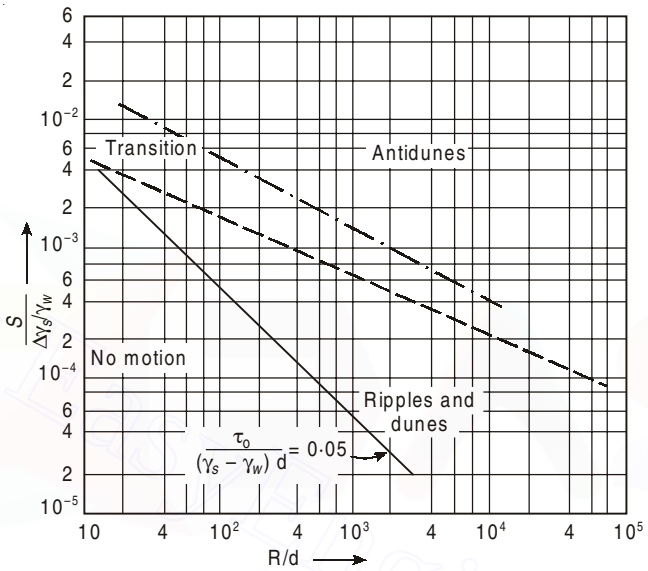


Figure 28.3 *Regime criterion –* $\left[\frac{S}{\Delta\gamma_s/\gamma_w} \textit{ versus } \left(\frac{R}{d}\right) \textit{ plot}\right]$

Further these investigators have proposed the use of the following resistance equation

$$\frac{V}{\sqrt{\frac{\Delta\gamma_s}{\rho_w}d}} = K\left(\frac{R}{d}\right)^{2/3}\left(\frac{S}{\Delta\gamma_s/\gamma_w}\right)^{1/2} \quad \ldots(28.8)$$

in which K is a constant, the value of which for the plane bed with no sediment motion is given as 7.66 and that for the antidune regime of flow it is given as 6. For the other regimes of flow viz., ripples, dunes, or transition, two graphical plots of $\frac{V}{\sqrt{\frac{\Delta\gamma_s}{\rho_w}R}}$ versus $\left[(R/d)^{1/3}\left(\frac{S}{\Delta\gamma_s/\gamma_w}\right)\right]$ and K versus $\frac{V}{\sqrt{\frac{\Delta\gamma_s}{\rho_w}R}}$ have been given which may be used to obtain the value of K in Eq. 28.8. Thus the mean velocity of flow in an alluvial stream for known values of R (or D), S, $\Delta\gamma_s$, γ_w and d may be computed

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

using Eq. 28.8, by first predicting regime of flow and then determining the appropriate values of K to be used in Eq. 28.8.

28.9 DESIGN OF UNLINED ALLUVIAL CHANNELS-KENNEDY'S AND LACEY'S THEORIES

In our country irrigation channels are usually constructed in alluvial soil without any lining in the bed and banks. In the design of such channels the main aim of the designer is to obtain stable-non-silting and non-scouring channel section. Several attempts have been made to design stable unlined alluvial channels, but the most notable contributions are those of Kennedy and Lacey which are briefly discussed below.

28.9.1 Kennedy's Theory

According the Kennedy the *critical velocity* V_0 in a channel may be defined as the mean velocity of flow which will just keep the channel free from silting or scouring. On the basis of the observations made on certain stable reaches of Upper Bari Doab canal system, Kennedy gave the following formula (converted in SI or metric units) for the critical velocity (in m/s)

$$V_0 = 0.55 \text{ m } D^{0.64} \qquad ...(28.9)$$

where D is depth of flow in the channel in metres. This equation is applicable to channels flowing in sandy silt of the same grade as that which exists on the Upper Bari Doab canal system on which the observations were made. Kennedy later recongnized that the grade of the silt played a significant role in determining the critical velocity and therefore introduced another factor in the above equation, called by him *critical velocity ratio* or C.V.R, m, to make the equation applicable to other regions also. The equation was then given as

$$V = 0.55 \text{ m } D^{0.64} \qquad ...(28.10)$$

where $m = (V/V_0)$, which may be assigned a value from 1.1 to 1.2 for sands coarser than that of Upper Bari Doab canal system and a value from 0.8 to 0.9 for the finer sands.

For the bed slope of the channel Kennedy considered Kutter's formula as noted below,

$$V = \frac{23 + \frac{1}{n} + \frac{0.00155}{S}}{1 + \left(23 + \frac{0.00155}{S}\right)\frac{n}{\sqrt{R}}}\sqrt{RS} \qquad ...(28.11)$$

Using these equations the following procedure may be adopted to design channel section according to Kennedy's theory.

The design discharge of the channel Q, the rugosity coefficient n, the bed slope S (obtained from the longitudinal section) and the C.V.R. m are known.

(i) Assume a trial value of D metres.

(ii) Compute the value of V using Eq. 28.10.

(iii) Compute area of flow section using continuity equation as $A = (Q/V)$.

(iv) Assuming the side slope of the channel as $\frac{1}{2}$: 1 compute the bed width of the channel B using the relation

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$A = \left[B + \left(\frac{1}{2}\right)D\right]D$$

(v) Compute the wetted perimeter P using the relation

$$P = \left(B + \sqrt{5}D\right)$$

and also compute the hydraulic radius R, since $R = (A/P)$.

(vi) Compute the mean velocity of flow from Kutter's Eq. 28.11. If this value of velocity is same as that found in step (ii), the assumed depth is correct. If not repeat the calculations with a changed value of D till the two values of the velocity come out to be very nearly the same.

Instead of using the above noted trial procedure for the design of a channel, in actual practice and design is carried out with the help of Garret's diagrams which provide a graphical solution of Kennedy's and Kutter's equation.

However, if in place of the slope S, the B/D ratio is assumed to be given then the design of the channel section may be carried out as follows:

(i) The area of flow section A may be expressed in terms of D as

$$A = D^2 (x + 0.5)$$

where $(B/D) = x$, which is known.

(ii) Using Eq. 28.10 and the continuity equation the following equation for the depth of flow D may be obtained

$$Q = 0.55 \text{ m } (x + 0.5)\, D^{2.64}$$

from which the value of D may be computed.

(iii) Knowing D, compute B and R and using Kennedy's Eq. 28.10 compute V.

(iv) Knowing V and R compute the bed slope S from Kutter's Eq. 28.11.

28.9.2 Lacey's Regime Theory

The regime theory postulates that a channel with erodible boundaries tends to adjust its dimensions viz., width, depth and slope in order to attain a state of equilibrium called regime state. Thus a regime channel has been defined by Lacey as a stable channel transporting the minimum bed load consistent with a fully active bed. According to Lacey a channel will be in regime if it carries a constant discharge and it flows uniformly in unlimited, incoherent alluvium of the same character as that transported and the silt grade and the silt charge are constant.

Lacey has differentiated between the initial and final regime conditions of a channel. The initial regime condition is attained for a channel shortly after it has been constructed and put into service. The channel adjusts bed slope either by silting or scouring but on account of the tenacity of the banks, the bed-width of the channel is not altered. The channel then appears to have attained stability but it is not the final state of stability and hence it represents the initial regime condition of the channel. When the continuous action of water ultimately overcomes the resistance of the banks and sets up a condition such that the channel may adjust its complete section, then the final or true regime condition is attained. On the basis of the available data, Lacey developed a number of regime relationships to facilitate the design of regime or stable channels. Some of these relationships are given below.

The first two equations originally suggested by Lacey in F.P.S. units, and converted in SI or metric units are

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$V = \sqrt{\frac{2}{5} fR} \quad \text{...(28.12)}$$

$$Af^2 = 140\, V^5 ; Qf^2 = 140\, V^6 \quad \text{...(28.13)}$$

where V is velocity of flow in m/s, A is area of cross-section of channel, f is silt factor, and R is hydraulic radius. By comparing Eq. 28.12 with Kennedy's Eq. 28.10 it may be noted that Lacey considered the hydraulic radius rather than the depth of flow as a significant variable. Further the silt factor was introduced as a measure of the silt grade, and it was related to the mean grain size in case of regime channels as indicated below,

$$f = 1.76 \sqrt{m_r} \quad \text{...(28.14)}$$

where m_r is the mean particle size in mm.

By eliminating f in Eqs 28.12 and 28.13 the following relationship between wetted perimeter P and discharge Q may be obtained,

$$P = 4.75 \sqrt{Q} \quad \text{...(28.15)}$$

This equation which shows that the wetted perimeter of a regime channel is determined by discharge alone and is independent of silt grade, is very important and useful.

For computing the bed slope for a regime channel, Lacey obtained the following relationship,

$$S = \frac{f^{5/3}}{3340\, Q^{1/6}} \quad \text{...(28.16)}$$

Using these equations a channel section may be designed according to Lacey's theory by adopting the following procedure:

The design discharge Q and the mean particle size m_r are known.

(i) Using Eq. 28.14 compute the silt factor f.

(ii) Using Eq. 28.13, compute the velocity of flow V.

(iii) Compute the area of flow section using continuity equation $A = (Q/V)$.

(iv) From Eq. 28.15 compute the wetted perimeter P.

(v) Assuming the side slope of the channel as $\frac{1}{2}$: 1, compute bed width B and the depth D using the relations

$$A = \left[B + \left(\frac{1}{2}\right) D\right] D$$

and

$$P = \left(B + \sqrt{5}\right) D$$

(vi) Computed the bed slope S from Eq. 28.16.

ILLUSTRATIVE EXAMPLES

Example 28.1. *Using White's equation, find the critical tractive stress for*
(*a*) *0.5 mm and grains in water,*

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\eta = 0.6, \frac{V_* d}{\upsilon} > 3.5$$

(*b*) *2 mm perspex grains (sp. gr. 1.4) in water,*

$$\eta = 0.5, \frac{V_* d}{\upsilon} > 3.5$$

The flow is turbulent in both cases. Take turbulent coefficient 1/4 and $\tan \phi = 0.6$.

Solution

According to White's equation for $\frac{V_* d}{\upsilon} > 3.5$, critical tractive stress is given by

$$\tau_{cr} = \frac{\pi}{6} \eta d \, (\gamma_s - \gamma_w) \tan \phi$$

or

$$\tau_{cr} = \frac{\pi}{6} \eta d \, (s - 1) \, \gamma_w \tan \phi$$

Assuming the specific gravity of sand grains as 2.65, and substituting the values, we get

(a) $$\tau_{cr} = \frac{\pi}{6} \times 0.6 \times \frac{0.5}{10^3} \times (2.65 - 1) \times 9\,810 \times 0.6$$

$$= 1.526 \text{ N/m}^2$$

Since turbulent coefficient $= \frac{1}{4}$, the average value of the critical tractive stress is

$$(\tau_{cr})_{av} = \frac{\tau_{cr}}{4} = \frac{1.526}{4} = 0.382 \text{ N/m}^2$$

(b) $$\tau_{cr} = \frac{\pi}{6} \times 0.5 \times \frac{2}{10^3} \times (1.4 - 1) \times 9\,810 \times 0.6$$

$$= 1.233 \text{ N/m}^2$$

The average value of the critical tractive stress is then

$$(\tau_{cr})_{\text{av}} = \frac{1.233}{4} = 0.308 \text{ N/m}^2$$

Example 28.2. *Using Jeffreys' equation find the velocity needed to lift (a) 0.2 mm, (b) 0.6 mm sand grains from the bottom.*

Solution

From Eq. 28.3 the velocity required to lift the sand grain is given by

$$V^2 \geq \frac{\rho_s - \rho_w}{\rho_w} gr \times \frac{1}{\left(\frac{1}{3} + \frac{1}{9}\pi^2\right)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $$V^2 \geq (s-1)\, gr \times \frac{1}{1.43}$$

(a) In this case $r = 0.1$ mm, $s = 2.65$, thus by substitution the minimum value of the velocity required to lift the grain is obtained as

$$V^2 = (2.65-1) \times 9.81 \times \frac{0.1}{10^3} \times \frac{1}{1.43} = 1.132 \times 10^{-3}$$

$\therefore$ $V = 0.034$ m/s $= 34$ mm/s

(b) $r = 0.3$ mm

$$V^2 = (2.65-1) \times 9.81 \times \frac{0.1}{10^3} \times \frac{1}{1.43} = 3.396 \times 10^{-3}$$

$\therefore$ $V = 0.058$ m/s $= 58$ mm/s

Example 28.3. *The following data are given for a trapezoidal channel with side slopes* $\frac{1}{2}$ *hor. : 1 ver.*

Discharge $Q = 140\ m^3/s$*; bed width* $B = 50$ *m; depth* $d = 2.5$ *m; bed slope* $S = 1.6 \times 10^{-4}$*; grain diameter* $d = 0.3$ *mm.*

Find the total shear stress at the bed τ_0 *and its component* τ_0' *due to sand grains and* τ_0'' *due to bed deformations.*

Solution

Considering the channel section to be wide rectangular with no side friction, we have

$$A = 50 \times 2.5 = 125\ m^2$$

$$R \approx D = 2.5\ m$$

The total bed shear stress

$$\tau_0 = \gamma RS$$

$$= 9\,810 \times 2.5 \times 1.6 \times 10^{-4} = 3.924\ N/m^2$$

Further, we have

$$\frac{V}{V_*'} = 7.66 \left(\frac{R'}{k_s}\right)^{1/6}$$

$$V = \frac{Q}{A} = \frac{140}{125} = 1.12\ m/s$$

$$k_s = d = 0.3\ mm = (0.3 \times 10^{-3})\ m$$

$$V_*' = \sqrt{\frac{\tau_0'}{\rho}} = \sqrt{\frac{wR'S}{\rho}} = \sqrt{gR'S}$$

Thus by substitution, we get

$$\frac{1.12}{\sqrt{9.81 \times R' \times 1.6 \times 10^{-4}}} = 7.66 \left(\frac{R'}{0.3 \times 10^{-3}}\right)^{1/6}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or $R' = 0.93$ m

Since $\tau_0' = \gamma R' S$

$\therefore \quad \tau_0' = 9\,810 \times 0.93 \times 1.6 \times 10^{-4} = 1.46$ N/m^2

Again $\tau_0 = \tau_0' + \tau_0''$

$\therefore \quad \tau_0'' = (\tau_0 - \tau_0') = (3.924 - 1.46) = 2.464$ N/m^2

Example 28.4. *For a wide rectangular channel the mean velocity of flow is 0.6 m/s; the depth of flow 0.6 m; size of bed material 0.915 mm; specific gravity of material 2.65; bed slope 1 in 6 900; υ for water 1×10^{-6} m^2/s. Examine the possibility of motion of bed.*

Solution

Boundary shear stress is given as

$$\tau_0 = \gamma RS = \gamma DS = 9\,810 \times 0.6 \times \frac{1}{6900} = 0.853 \text{ N/m}^2$$

Shields' entrainment function is given as

$$\frac{\tau_0}{(s-1)\gamma d} = \frac{0.853}{(2.65-1)\times 9810 \times 0.915 \times 10^{-3}} = 0.058$$

Further shear Reynolds number is given as

$$\frac{V_* d}{\upsilon} = \frac{\sqrt{gRS} \times d}{\upsilon} = \frac{\left(\sqrt{\frac{9.81 \times 0.6}{6900}}\right) \times 0.915 \times 10^{-3}}{1 \times 10^{-6}} = 26.72$$

Now by plotting this point on the Shields' curve shown in Fig. 28.1 it is observed that it lies above the Shields' curve and hence it is concluded that the bed material will be in motion.

Example 28.5. *A very wide channel passing through very coarse material of representative sediment size 25 mm has a depth of flow 1.07 m. If the bed is under condition of incipient motion, find the slope and discharge per metre width. Take specific gravity of material 2.68.*

Solution

According to Shield's consideration at the condition of incipient motion, we have

$$\frac{\tau_0}{(s-1)\,\gamma d} = 0.06$$

or $$\frac{\gamma DS}{(s-1)\gamma d} = 0.06$$

or $$\frac{1.07 \times S}{(2.68-1)\times 25 \times 10^{-3}} = 0.06$$

$\therefore \quad S = 2.36 \times 10^{-3} = 1$ in 425

According to Manning's formula the discharge per metre width is given as

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$q = \frac{1}{n} \times R^{2/3} \times S^{1/2} \times D$$

The value of n is given by Strickler's equation as

$$n = \frac{(d_{50})^{1/6}}{21}$$

in which $d_{50} = (25 \times 10^{-3}) = 0.025$ m

Thus $$n = \frac{(0.025)^{1/6}}{21} = 0.026$$

$\therefore$ By substitution, since $R = D = 1.07$ m, we get

$$q = \frac{1}{0.026} \times (1.07)^{5/3} \times \left(\frac{1}{425}\right)^{1/2}$$

$$= 2.09 \text{ cumec/m}$$

Example 28.6. *An alluvial river rectangular in cross-section has average width 107 m, representative sediment size 0.33 mm and average water surface slope of 5.3 × 10^{-4}. Prepare a stage-discharge table for the depths of flow varying from 0.30 m to 4.5 m.*

Solution

Assuming the specific gravity of the material as 2.65, we have

$$\frac{S}{(\Delta\gamma_s / \gamma_w)} = \frac{5.3 \times 10^{-4}}{1.65} = 3.21 \times 10^{-4}$$

From the plot between $\left(\frac{S}{\Delta\gamma_s / \gamma_w}\right)$ and $\left(\frac{R}{d}\right)$ as shown in Fig. 28.3, it will be observed that for $\left(\frac{S}{\Delta\gamma_s / \gamma_w}\right) = 3.21 \times 10^{-4}$, at $\left(\frac{R}{d}\right) = 5\,000$ the bed changes from dunes to plane bed and upto $\left(\frac{R}{d}\right) = 180$ (approximately) there is a state of no motion.

Now considering a depth of 0.30 m, $R \approx D = 0.30$ m.

So $$\frac{R}{d} = \frac{0.30}{0.33 \times 10^{-3}} = 909$$

It is thus indicated that in this case the bed will be in motion for the depth of flow varying from 0.30 to 4.5 m.

In Eq. 28.8 for dune bed condition the value of K may be taken as 4 and for plane bed condition the value of K may be taken as 6. The following expressions may be obtained from Eq. 28.8 for calculating the mean velocity of flow in an alluvial stream for different bed conditions.

(i) $\frac{V}{\sqrt{gdS}} = 4\left(\frac{R}{d}\right)^{2/3}$ for dune bed condition

(ii) $\frac{V}{\sqrt{gdS}} = 6\left(\frac{R}{d}\right)^{2/3}$ for plane bed condition

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$\sqrt{gdS} = \sqrt{(9.81 \times 0.33 \times 10^{-3} \times 5.3 \times 10^{-4})} = 1.31 \times 10^{-3}$$

The required stage discharge table is as given below:

Depth	$\frac{R}{d}\left(=\frac{D}{d}\right)$	*V*(*m*/*s*) *from* (*i*)	*V*(*m*/*s*) *from* (*ii*)	*Q* (m^3/s) (= *V*× *B* ×*D*)
0.30	909	0.492	—	15.793
0.60	1 818	0.781	—	50.140
0.90	2 727	1.023	—	98.515
1.20	3 636	1.239	—	159.088
1.50	4 545	1.438	—	239.799
*1.80	5 455	—	2.436	469.174
2.10	6 364	—	2.699	606.465

* As stated earlier for $\left(\frac{R}{d}\right) \geq 5000$ the dune bed changes to plane bed.

Depth	$\frac{R}{d}\left(=\frac{D}{d}\right)$	*V*(*m*/*s*) *from* (*i*)	*V*(*m*/*s*) *from* (*ii*)	*Q*(m^3/s) (= *V*× *B* ×*D*)
2.40	7 273	—	2.951	757.817
2.70	8 182	—	3.192	922.169
3.00	9 091	—	3.424	1099.104
3.30	10 000	—	3.648	1288.109
3.60	10 909	—	3.866	1489.183
3.90	11 818	—	4.078	1701.749
4.20	12 727	—	4.285	1925.679
4.50	13 636	—	4.486	2160.009

With the help of the above table stage-discharge curve may be plotted.

SUMMARY OF MAIN POINTS

1. The mobile boundary channels include rivers and unlined alluvial candls; the boundaries of which are made up of loose soil which can be easity eroded and transported by the flowing water.
2. Depending on the shape, size and composition of the sediment particles and the velocity of the flowing water the sediment particles are transported in one or more of the following modes:
   (i) Sliding or rolling along the bed
   (ii) Saltation or skipping
   (iii) Movement in suspension.
3. The sediment transported by a stream or channel may be divided into the three catagories as
   (i) Bed load
   (ii) Suspended load
   (iii) Wash load
4. The tractive force required to initiate general movement of the sediment is called *critical tractive force* $\tau_{cr.}$

According to shield the value of τ_{cr} is given by

$$\frac{\tau_{cr}}{(r_s - r_w)d} = 0.06$$

5. The bed of a stream transporting sediment under usual conditions of flow may develop bed configurations such as ripples dunes, standing wave and antidunes, which affect the resistance to flow.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Flow Measurement and Laboratory Experiments

Chapter 29

29.1 INTRODUCTION

Fluid flow measurement constitute an essential part of the study of hydraulics and fluid mechanics. Such measurements are always required to be carried out both in the laboratory as well as in the field, in order to obtain information regarding the fluid and flow characteristics. In laboratory a large number of experiments can be performed to clearly demonstrate the various principles of hydraulics and fluid mechanics. Such experiments help in better understanding of the subject and developing an aptitude for research work. All these experiments either in the laboratory or in the field necessarily involve the measurement of various quantities such as pressure, velocity, discharge, etc. The various methods adopted for such measurements are listed below, followed by a list of various experiments commonly performed in the laboratory.

29.2 FLUID FLOW MEASUREMENTS

(a) Measurement of Pressure. The various devices used for pressure measurement are of two types viz., manometers and mechanical gages. Their further subdivisions are as follows :

(i) *Manometers.* (1) Piezometer tube, (2) Single column manometer, (3) U-tube differential manometer, (4) Inverted U-tube differential manometer, (5) Micro-manometer, (6) Inclined manometer.

(ii) *Mechanical Gages.* (1) Pressure gage, (2) Vacuum gage (3) Compound gage. There are different types of these gages such as Bourdon tube, diaphragm, bellows, dead weight, etc.

All the above indicated pressure measuring devices have been described in Chapter 2.

In addition to the above noted pressure measuring devices there are several other pressure measuring devices which involve the measurement of some electrical quantity to indicate the pressure as well as its variation. Among such pressure measuring devices the most common are the *pressure transducers.* A pressure transducer usually has a thin diaphragm, on one side of which the fluid pressure acts. The movements of the diaphragm caused by the changes of pressure are indicated by an electrical strain gage on the diaphragm which with proper calibration would facilitate the measurement of pressure. Alternatively the change of electrical capacitance between the moving diaphragm and a fixed plate may be measured which would also facilitate the measurement of pressure. Most of the pressure transducers will respond to very high pressure and to fluctuations of pressure of several cycles/sec.

Electronic pressure cells such as *piezoelectric cells* are another pressure measuring devices suitable for measuring very high pressures as well as pressure fluctuations. These devices use a crystal of quartz

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

or other material which when subjected to the pressure of fluid, produces across itself a small but measurable difference of electrical potential. The quartz crystal for example will respond to pressure as great as 350 kg(f)/cm^2 and to fluctuations of pressure from 1 to 10,000 cycles/sec. Some pressure gages utilize the increase of electrical resistance exhibited by metals under very high pressures.

(**b) Measurement of Liquid Surface Elevation or Depth of Flow.** For the measurement of liquid surface elevation or the depth of flow the following devices are commonly used :

(1) Point gage, (2) Hook gage, (3) Float gage.

A *point gage* consists of a thin vertical rod, pointed at its lower end. The rod is attached to a rack and pinion arrangement by means of which the rod is lowered until its point just touches the liquid surface (that is the points appears to touch its reflection in the liquid surface). The liquid surface elevation is then read from a vernier scale attached to the rod. Sometimes electric circuit may be used in which case the correct position of the rod may be indicated by the completion of an electric circuit when the point just touches the liquid surface.

A *hook gage* also consists of a thin vertical rod the end of which is bent round to form U with the point facing upwards. Again by means of rack and pinion arrangement the point is brought to the liquid surface from below and the reading is taken on the attached vernier scale when the point just touches the liquid surface. Somewhat greater accuracy is obtained with a hook gage.

Although these gages may be used in a flowing liquid, but it is preferred to use them in a stilling well provided at the side of the flow channel and connected to it at the bottom.

A *float gage* consists of a wide float attached to a counter weight by a thin cord or wire passing through a frictionless pulley. The water surface elevation may be read on a graduated scale. Several automatic liquid surface recording float gages have been developed which are commonly used for recording the water surface elevation in rivers as well as in channels. The float gage is generally used in a stilling well provided by the side of the channel and connected to it at the bottom.

(c) Measurement of Velocity. For the measurement of local velocities at different points in the flow field, the following devices are commonly adopted:

(1) Pitot tube, (2) Pitot cylinder, (3) Pitot sphere, (4) Currentmeter, (5) Hot-wire anemometer (for gases), (6) Floats.

These devices have also been described earlier in the corresponding chapters. The mean velocity of flow may be determined by integrating the velocity distribution profile over the entire section of the flow passage. Often an indirect method is used for determining the mean velocity of flow which involves the measurement of the discharge and the area of the flow passage.

Some of the special techniques used for measuring the velocity of flow in rivers and large pipes are (i) Salt velocity method, and (ii) Inertia pressure method.

The *salt velocity method* has been devised by C.M. Allen and E.A. Taylor. In this method a concentrated solution of salt is suddenly injected into the water. At each of two cross-sections on the downstream of the injection point a pair of electrodes is inserted in the flow, the two pairs being connected in parallel. When the salt enriched water passes between a pair of electrodes its higher conductivity causes the electric current to rise briefly to a peak. The mean velocity of the flow stream is determined from the distance between the two electrode sections and the time interval between the appearance of the two current peaks. However, an elaborate apparatus is needed for the automatic recording of the data.

The *inertia pressure method*, which has been devised by N.R. Gibson, may be used for determining the mean velocity of liquids flowing in long pipe-lines. This method involves rapidly closing a valve at the downstream end of the pipe and recording a diagram of pressure p against time t for a point

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

immediately upstream of the valve. Then as indicated earlier for a pipe of uniform cross section and length l,

$$\int pdt = \frac{wlV}{g} \qquad \ldots(29.1)$$

The value of the integral is determined graphically and the original mean velocity V may then be calculated.

(d) Measurement of Discharge

(*i*) *For Pipes.* For the measurement of discharge in closed conduits (pipes) the following devices are commonly adopted:

(1) Flow meters — inferential and positive displacement type, (2) Venturi meter, (3) Orifice meter, (4) Nozzlemeter, (5) Bend meter or Elbow meter, (6) Rotameter.

Most of the above noted devices have been described in the previous chapters.

(*ii*) *For Channels.* For the measurement of discharge in open channels the following devices are commonly adopted :

(1) Sharp Crested Weirs, (2) Broad Crested Weirs, (3) Metering Flumes.

All these devices have also been described in the earlier chapters. Another method for the measurement of the discharge both in pipes as well as in channels may be that of measuring the volume or weight of the flowing liquid for a known interval of time. The discharge may also be obtained by determining the mean velocity of flow from the velocity distribution at a section and multiplying the mean velocity and the area of cross-section of the flow passage.

For rivers and large pipe-lines in which thorough mixing of the liquid occurs the salt dilution methods may be used for determining the discharge. A concentrated solution of salt is fed into the flowing stream at a known steady rate q, preferably from a number of points distributed over the cross-section. The concentration of salt is measured some distance downstream after the salt has become thoroughly diffused throughout the liquid. Let the mass of salt per unit volume of the original stream be s_0, the mass of salt per unit volume of the added solution be s_1 and the mass of salt per unit volume measured downstream be s_2. If the rate of flow of stream is Q then, when steady conditions have been attained, the mass of salt reaching the downstream sampling point in unit time is given by

$$Qs_0 + qs_1 = (Q + q)\, s_2$$

From which

$$Q = q\left(\frac{s_1 - s_2}{s_2 - s_0}\right) \qquad \ldots(29.2)$$

The concentration may be determined by chemical titration or by electrical conductivity measurements.

For measuring the discharge from a small reservoir or tank the devices commonly adopted are orifices and mouthpieces, which have been described in previous chapters.

(e) Measurement of Shear Stress. The measurement of shear stress can be accomplished by both direct and indirect methods. Shear stress can be measured directly by means of either Stanton tube or Preston tube.

A *Stanton tube* is basically a flattened pitot tube with a rectangular sharp edge opening of very small height (a few hundreths of centimetre). The tube is held with its opening facing upstream and the

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

lower surface of the opening coincident with the surface of the plate itself so that the whole of the opening is well immersed in the laminar sublayer of the plate. The total head reading of this tube minus the static pressure (i.e., dynamic pressure) is then simply related to velocity gradient at the surface and therefore to the viscous or shear stress at the plate. However, the exact relation between the dynamic pressure and the boundary shear stress must be obtained by calibration of the tube at the wall of a pipe or channel where skin friction can be readily determined from the pressure gradient. But extreme care and great technical precision are required if this method is to yield results of adequate accuracy.

Preston tube is another surface pitot tube used for the measurement of shear stress, which is similar to an ordinary circular pitot tube, but the diameter of the tube is small enough for the tube to be confined to the inner region of the boundary layer, (for which the diameter d of the pitot tube must be of the order of one-tenth of the boundary layer thickness or less). When the Preston tube is placed in contact with the boundary surface and wholly immersed in the inner region of the boundary layer, then the difference between the pitot pressure P (i.e., total head tube reading) and the static pressure p_0 (i.e., $P - p_0$, the dynamic pressure) would depend only on mass density ρ and kinematic viscosity υ of the fluid, boundary shear stress τ_0 and the diameter d of the tube. Using dimensional analysis the following functional relationship may be obtained amongst the above parameters.

$$\frac{(P - p_0)d^2}{\rho\upsilon^2} = \phi\left(\frac{\tau_0 d^2}{\rho\upsilon^2}\right) \qquad \ldots(29.3)$$

The exact relationship may however be obtained by calibration of the tube, which can be used to determine the boundary shear stress by measuring the dynamic pressure for the tube held in contact with the boundary. It has been determined experimentally that the calibration for pipes is not precisely the same as that for flat plates. Thus for flow in pipes the above relation is found to be of the form

$$\log_{10}\left(\frac{\tau_0 d^2}{4\rho\upsilon^2}\right) = -1.396 + 0.875 \log_{10}\left[\frac{(P - p_0)d^2}{4\rho\upsilon^2}\right] \qquad \ldots(29.4)$$

whilst for the boundary layer on a flat plate the relation is

$$\log_{10}\left(\frac{\tau_0 d^2}{4\rho\upsilon^2}\right) = -1.366 + 0.877 \log_{10}\left[\frac{(P - p_0)d^2}{4\rho\upsilon^2}\right] \qquad \ldots(29.5)$$

As compared with Stanton tube a Preston tube has advantages of greater robustness, repeatability of calibration, and the pressure difference recorded are greater.

Indirect methods can also be used to determine shear stress as indicated below. For laminar flow the boundary shear stress τ_0 can be calculated, if the velocity distribution above the boundary is known, since from the Newton's law of shear

$$\tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=0}$$

Similarly for steady flow in pipes the shear stress τ at a distance r from the centre of the pipe is given by

$$\tau = \frac{r}{2}\frac{\partial p}{\partial x}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where $(\partial p / \partial x)$ is the pressure gradient with respect to distance along the pipe. Shear stress can be calculated from this equation for both turbulent flow and laminar flow in pipes, since its derivation is independent of the nature of the flow.

The boundary shear stress at any section of an open channel can be determined by $\tau_0 = wRS_0$ where w is the specific weight of water, R is hydraulic mean depth S_0 is the bed slope.

Further for the turbulent flow the shear stress at a point of boundary can be determined by

$$\tau_0 = \rho \left[\frac{v_2 - v_1}{\frac{2.3}{\kappa} \log_{10}(y_2 / y_1)} \right]^2$$

in which v_1 and v_2 are the velocities parallel to the boundary surface at distance y_1 and y_2 measured from and normal to the boundary. The above equation can be derived by solving for the difference between the velocities v_1 and v_2, as described by the von Karman logarithmic velocity distribution law. In the above expression κ (Greek 'kappa') may be assumed as 0.4. The accuracy of the above equation is limited by variation in κ and the accuracy with which the velocities at y_1 and y_2 can be measured.

29.3 FLOW VISUALIZATION TECHNIQUES

Several techniques have been developed for making flow visible. Filaments of dye, small solid particles, globules of an immiscible liquid or minute hydrogen bubbles produced by electrolysis may be used to indicate the flow pattern of a liquid. In gases, smoke or visible vapours may be used to visualise the flow pattern.

In a gas local changes of density, especially the abrupt changes occurring across a shock wave, may be made visible by adopting the techniques such as shadowgraph method, Schlieren or Topler system and interferometer. All these techniques utilize this property that the change in density of a gas is accompanied by a change of refractive index. Out of these three techniques the shadowgraph method is the simplest which is described below.

In *shadowgraph* light from a point source, or in a parallel beam is passed through the gas flow and on to a screen. If the gas density is uniform the light rays, even if refracted, will have an unchanged spacing and hence these will illuminate the screen uniformly. However, if there is a sharp density gradient in a direction normal to the light beam, the rays will either diverge or converge according to whether the density gradient increases or decreases. Thus the portions of high density side where the rays converge will be indicated on the screen by a brighter band and that on the low density side where the rays diverge will be indicated on the screen by a darker band.

29.4 LIST OF EXPERIMENTS

The various experiments that are normally conducted in laboratory are grouped under the following heads:

(a) General

(1) Experimental determination of the metacentric height.
(2) Verification of Bernoulli's theorem.
(3) Verification of momentum principle.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(4) Determination of the coefficient of viscosity.
(5) Determination of C_c, C_v, C_d of a circular orifice under constant head.
(6) Determination of C_d of orifices of different shapes viz., circular, square, triangular, rectangular, etc., by constant head and falling head methods.
(7) Determination of C_d of cylindrical, convergent and divergent mouthpieces.
(8) Determination of C_d of submerged orifice.

(b) Pipes

(1) Determination of coefficients for head losses due to sudden enlargement, contraction, bend, valve, etc.
(2) Determination of discharge by integration of velocity distribution obtained with the help of a pitot tube.
(3) Determination of discharge coefficient of a venturimeter.
(4) Determination of discharge coefficient of an orifice-meter.
(5) Determination of discharge coefficient of elbow meter or pipe bend meter
(6) Determination of discharge constant of a bye-pass.
(7) Study of transition from laminar to turbulent flow and determination of critical velocity.
(8) Determination of Darcy's *f* for commercial pipes.
(9) Study of water hammer phenomenon.

(c) Channels

(1) Determination of discharge coefficients of sharp-edged rectangular and triangular notches.
(2) Determination of discharge coefficient of broad crested weir.
(3) Determination of discharge coefficient of standing wave flume.
(4) Calibration of currentmeter.
(5) Determination of velocity distribution in a channel by a currentmeter.
(6) Determination of velocity distribution in a channel by pitot tube.
(7) Determination of Manning's *n* and Chezy's *C*.
(8) Study of flow in rectangular channel with a hump in the bottom.
(9) Study of a hydraulic jump downstream of a slopping glacis and on the glacis.
(10) Determination of end depth at a free overfall at the end of a channel.

(d) Advanced

(1) Verification of Stokes' law.
(2) Verification of Darcy's law for flow through porous media.
(3) Study of free vortex phenomenon.
(4) Study of various characteristics of diffusion of a submerged jet.
(5) Study of the establishment of laminar flow in a smooth pipe.
(6) Study of distribution of flow in an offtake channel.
(7) Study of submerged hydraulic jump below a spillway.
(8) Study of flow through a flumed channel.
(9) Study of streamline patterns by Hele-Shaw's apparatus.
(10) Plotting of flow-nets by electrical analogy method.
(11) Study of the pressure distribution around an object viz., cylinder, sphere, airfoil etc., in a wind tunnel.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(12) Calibration of hot-wire anemometer in a wind tunnel.
(13) Study of boundary layer growth in a wind tunnel.
(14) Study of effect of artificial and natural bottom roughness on Manning's *n*.

(e) Machines

(1) Determination of characteristics curves for a hydraulic ram.
(2) Determination of characteristic curves and specific speed of a Pelton wheel.
(3) Determination of characteristic curves and specific speed of a Francis turbine.
(4) Determination of characteristic curves and specific speed of a Kaplan turbine.
(5) Determination of characteristic curves and specific speed of a Centrifugal pump.
(6) Determination of characteristic curves of a Reciprocating pump.

29.5 WRITING OF REPORT

A complete report of an experiment conducted in the laboratory should include the following points:

(a) **Figure.** On left blank page of laboratory journal a neat sketch or diagram of the experimental set-up should be given, which should clearly show the various parts of the set-up.

(b) **Date.** On right side page at top left corner date of conducting the experiment should be given.

(c) **Experiment Number.** On right side page at top right corner, serial number of the experiment should be written.

(d) **Object.** On right hand page the report of the experiment should start with object of the experiment which should clearly indicate the aim of the experiment.

(e) **Apparatus.** After the object, a list of the apparatus used for experiment should be given.

(f) **Theory or Formula.** A brief theory of the experiment or the relevant formula used should be given. The units of measurement for various terms in the formula should be clearly mentioned.

(*g*) **Procedure.** A brief outline of the procedure followed for the experiment should be given in steps in third person.

(h) **Observations.** The various observations collected during the conduction of the experiment should be given in a tabular form.

(i) **Calculations.** The calculations involved in arriving at the various results of the experiment should also be given in tabular form, either along with the observations, or otherwise a separate table for calculations may be given. A specimen calculation for one or two sets of observations should be given in detail on left side blank page.

(j) **Graphs.** Necessary graphs should be plotted in order to represent the results obtained from the experiment.

(k) **Results.** The results obtained from the experiment should be given.

(l) **Discussion.** A brief discussion of the experiment should be given which should include precautions, sources of error and comparison of the results obtained with the theoretical results.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Multiple Choice Questions

1. A fluid is a substance that
   (a) always expands until it fills any container.
   (b) is practically incompressible.
   (c) cannot remain at rest under action of any shear force.
   (d) cannot be subjected to shear forces.
2. Newton's Law of viscosity relates
   (a) pressure, velocity and viscosity.
   (b) shear stress and rate of angular deformation in a fluid.
   (c) shear stress, temperature, velocity and viscosity.
   (d) pressure, viscosity and rate of angular deformation.
3. Viscosity has the dimensions
   (a) $FL^{-2}T$ (b) $FL^{-1}\ T^{-1}$
   (c) FLT^{-2} (d) $FL^{2}T$.
4. Falling drops of rain acquire spherical shape on account of
   (a) viscosity (b) surface tension
   (c) vapour pressure (d) compressibility.
5. Surface tension has the dimensions
   (a) F (b) FL^{-1}
   (c) FL^{-2} (d) FL^{-3}
6. Capillarity is due to
   (a) adhesion
   (b) cohesion
   (c) adhesion and cohesion both
   (d) neither adhesion nor cohesion.
7. A prefect gas
   (a) has zero viscosity
   (b) has constant viscosity
   (c) is incompressible
   (d) satisfies $p = \rho RT$.
8. The height of liquid in a capillary tube
   (a) increases with an increase in diameter
   (b) increases with a decrease in diameter
   (c) decreases with an increase in surface tension
   (d) increase with an increase in specific weight.
9. The bulk modulus of elasticity
   (a) is independent of pressure and viscosity
   (b) increase with the pressure
   (c) has the dimensions of $(1/p)$
   (d) is larger when the fluid is more compressible.
10. For $\upsilon = 3 \times 10^{-4}$ stoke and $\rho = 0.8$ gm/cm^3, μ in msl/m-sec, is
    (a) 2.45×10^{-6} (b) 3.06×10^{-6}
    (c) 3.83×10^{-6} (d) 0.79×10^{-6}
11. Pressure at a point in a static mass of liquid depends upon
    (a) the shape and size of bounding container
    (b) the depth below the free liquid surface
    (c) the specific weight of liquid and depth below the free liquid surface
    (d) the specific weight of liquid, depth below the free liquid surface and the shape and

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

size of the bounding container.

12. The differential equation for pressure variation in a static fluid may be written (*z*-measured vertically upward)
    (a) $dp = -wdz$ (b) $dp = \rho dz$
    (c) $dp = wdz$ (d) $dp = -zd\rho$.
13. Select the correct statement
    (a) Absolute pressure = Gage pressure – Atmospheric pressure
    (b) Gage pressure = Absolute pressure – Atmospheric pressure
    (c) Absolute pressure = Atmospheric pressure + Vacuum pressure
    (d) Gage pressure = Atmospheric pressure + Vacuum pressure.
14. The pressure, in meters of oil (sp. gr. 0.8), equivalent to 80 m of water is
    (a) 64 (b) 80
    (c) 100 (d) 88
15. In an isothermal atmosphere, the pressure
    (a) remains constant
    (b) decreases linearly with elevation
    (c) increases exponentially with elevation
    (d) varies in the same way as the density.
16. A mercury-water manometer has a gage difference of 0.6 m (difference in elevation of menisci). The difference in pressure, measured in metres of water, is
    (a) 0.6 (b) 7.56
    (c) 8.16 (d) 7.86
17. In order to avoid a correction for the effect of capillarity in manometers used for measuring pressure, the diameter of tube should be
    (a) less than 3 mm
    (b) more than 3 mm but less than 5 mm
    (c) equal to 5 mm
    (d) greater than or equal to 6 mm.
18. The centre of pressure for a plane surface immersed vertically in a static mass of liquid is
    (a) always below the centroid of the area
    (b) always above the centroid of the area
    (c) always coincident with the centroid of the area
    (d) sometimes above and sometimes below the centroid of the area.
19. The horizontal component of force on a curved surface is equal to the
    (a) weight of liquid vertically above the curved surface
    (b) weight of liquid retained by the curved surface
    (c) product of pressure at its centroid and area
    (d) force on a vertical projection of the curved surface
20. The vertical component of pressure force on a submerged curved surface is equal to
    (a) its horizontal component
    (b) the force on a horizontal projection of the curved surface
    (c) the weight of liquid vertically above the curved surface
    (d) the product of pressure at centroid and surface area.
21. A small plastic boat loaded with nuts and bolts is floating in a bath tub. If the cargo is dumped into the water, allowing the boat to float empty, then the water level in the tub will
    (a) rise (b) fall (c) not change
22. Bouyant force is
    (a) the resultant force on a body due to the fluid surrounding it
    (b) the resultant force acting on a floating body
    (c) equal to the weight of liquid displaced
    (d) the force necessary to maintain equilibrium of a submerged body.
23. The line of action of the buoyant force acts through the
    (a) centre of gravity of any submerged body
    (b) centroid of the volume of any floating body
    (c) centroid of the displaced volume of fluid
    (d) centroid of the horizontal projection of the body.
24. A body floats in stable equilibrium
    (a) when its metacentric height is zero
    (b) when its centre of gravity is below its centre of buoyancy

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(c) when GB—(I/V) is positive and G is above B
(d) when metacentre is above the centre of gravity

25. A tank 2 m square contains 1 m of water. If the tank is accelerated in the horizontal direction parallel to a pair of sides at the rate of 4 m/s^2, then for no spilling of water the minimum height of the tank should be
(a) 1 m (b) 1.4077 m
(c) 1.8155 m (d) 2 m.

26. A closed cubical box 0.8 m on each edge, is half filled with water and the other half being filled with oil, sp. gr. 0.75. If the box is accelerated vertically upward at the rate of 4.905 m/s^2, the pressure difference between bottom and top, in kg (f) m^2 is
(a) 1800 (b) 1575
(c) 1050 (d) 525

27. If the box of Q. 26 is accelerated in horizontal direction parallel to a pair of sides at the rate of 4.905 m/s^2, the maximum pressure on the top of the box in metres of water is
(a) 0.225 (b) 0.300
(c) 0.563 (d) 0.825

28. When a liquid rotates at constant angular velocity about a vertical axis as a rigid body, the pressure intensity
(a) decrease as the square of the radial distance
(b) increase linearly as the radial distance
(c) varies inversely as the elevation along any vertical line
(d) varies as the square of the radial distance

29. A right circular cylinder, open at the top is filled with liquid, sp. gr. 1.2. and rotated about its vertical axis at such speed that half the liquid spills out. The pressure at the centre of the bottom is
(a) zero
(b) one-fourth its value when cylinder was full
(c) one-half its value when cylinder was full
(d) same as before rotation

30. A forced vortex
(a) turns in an opposite direction to a free vortex
(b) has the velocity decreasing with the radius
(c) occurs when fluid rotates as a solid
(d) has the velocity decreasing inversely with the radius

31. A free vortex
(a) has the velocity decreasing with the radius
(b) has the velocity increasing with the radius
(c) has constant velocity
(d) has the velocity varying inversely with the square of radius.

32. A control volume refers to
(a) a fixed region in space
(b) a specified mass
(c) a closed system
(d) a reversible process only.

33. One dimensional flow is
(a) steady uniform flow
(b) uniform flow
(c) flow which neglects changes in a transverse direction
(d) restricted to flow in a straight line.

34. A practical example of steady non-uniform flow is given as the
(a) motion of a river around bridge piers
(b) steadily increasing flow through a pipe
(c) steadily increasing flow through a reducing section
(d) constant discharge through a long, straight tapering pipe

35. Continuity equation
(a) expresses the relation between energy and work
(b) relates the momentum per unit volume for two points on a streamline
(c) relates mass rate of flow along a stream tube
(d) constant discharge through a long, straight tapering pipe.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

36. Steady flow occurs when
    (a) conditions do not change with time at any point
    (b) conditions are the same at adjacent points at any instant
    (c) when $(\partial v/\partial s)$ is constant
    (d) conditions change steadily with the time.
37. A streamline
    (a) is the line connecting the mid-points of flow cross section
    (b) is defined for uniform flow only
    (c) is drawn normal to the velocity vector at every point
    (d) is fixed in space in steady flow.
38. Uniform flow occurs
    (a) whenever the flow is steady
    (b) when $(\partial v/\partial t)$ is every where zero
    (c) only when the velocity vector at any point remains constant
    (d) when $(\partial v/\partial s) = 0$.
39. For steady rotational flow of a fluid Bernoulli's equation
    (a) cannot be derived
    (b) can be derived for the entire flow field
    (c) can be derived for the points lying on the same streamline
    (d) can be derived only if the fluid is incompressible.
40. Stream lines, streak lines and path lines are all identical in case of
    (a) uniform flow (b) steady flow
    (c) unsteady flow
    (d) non-uniform flow.
41. Navier-Strokes equation is useful in the analysis of
    (a) Non-viscous flows
    (b) turbulent flow
    (c) viscous flows
    (d) both viscous and turbulent flows.
42. Euler's equation of motion can be integrated when it is assumed that
    (a) the continuity equation is satisfied
    (b) the fluid is incompressible
    (c) a velocity potential exists and the density is constant
    (d) the flow is rotational and incompressible.
43. In irrotational flow of an ideal fluid
    (a) a velocity potential exists
    (b) all particles must move in straight lines
    (c) the motion must be uniform
    (d) the velocity must be zero at a boundary.
44. If stream function $\psi = 2xy$, then the velocity at a point (1, 2) is equal to
    (a) 2 (b) 4
    (c) $\sqrt{20}$ (d) 16
45. An equipotential line
    (a) has no velocity component tangent to it
    (b) is same as streamline
    (c) has constant dynamic pressure
    (d) has no velocity component normal to it.
46. If successive measurements with a Prandtl-pitot tube indicate that the tip piezometer reading varies only across the flow and the side piezometer reading varies only in the direction of flow, the motion is evidently
    (a) uniform, irrotational
    (b) non-uniform, rotational
    (c) uniform, rotational
    (d) non-uniform, irrotational.
47. Specify which the following must be fulfilled by the flow of any fluid, real or ideal, laminar or turbulent
    (a) Newton's of viscosity
    (b) Velocity at boundary must be zero relative to boundary
    (c) The continuity equation
    (d) Velocity normal to a solid boundary is zero.
48. A source in two dimensional flow
    (a) is a point from which fluid is imagined to flow uniformly in all directions
    (b) is a line from which fluid is imagined to flow uniformly in all directions
    (c) has streamlines that are concentric circles
    (d) has a velocity potential independent of the radius.
49. The kinetic energy correction factor
    (a) has the units of velocity head

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) is expressed by $\frac{1}{A}\int_A (v/V)dA$

(c) is expressed by $\frac{1}{A}\int_A (v/V)^2 dA$

(d) is expressed by $\frac{1}{A}\int_A (v/V)^3 dA$.

50. The momentum correction factor is expressed as

(a) $\frac{1}{A}\int_A (v/V)dA$ (b) $\frac{1}{A}\int_A (v/V)^2 dA$

(c) $\frac{1}{A}\int_A (v/V)^3 dA$ (d) $\frac{1}{A}\int_A (v/V)^4 dA$.

51. The equation $\Sigma F_x = \rho Q\ (\Delta V_x)$ requires the following assumptions for its derivation
(a) the flow is steady and uniform
(b) the flow is steady and the velocity of flow is constant over the end cross-sections
(c) the flow is uniform and fluid is frictionless
(d) the fluid is frictionless and the velocity of flow is constant over the end cross-sections.

52. For laminar flow in pipes the momentum correction factor is
(a) less than 1 (b) 1.03
(c) 1.33 (d) 2.0

53. The ratio of actual discharge to theoretical discharge through an orifice is
(a) $C_c\ C_v$ (b) $C_c\ C_d$
(c) $C_v\ C_d$ (d) C_d/C_v

54. Coefficient of resistance C_r for an orifice is equal to

(a) $(1-C_v^2)$ (b) $\left(\frac{1}{C_v^2}-1\right)$

(c) $\left(\frac{1}{C_v^2 C_d^2}-1\right)$ (d) $\left(1-\frac{C_d^2}{C_v^2}\right)$

55. The loss of head due to sudden enlargement in a pipe is expressed by

(a) $\frac{V_1^2-V_2^2}{2g}$ (b) $\frac{V_1^2-V_2^2}{g}$

(c) $\frac{(V_1-V_2)^2}{2g}$ (d) $\frac{(V_1-V_2)^2}{g}$

56. The discharge through a V-notch varies as
(a) $H^{1/2}$ (b) H
(c) $H^{3/2}$ (d) $H^{5/2}$

57. The head loss in turbulent flow in pipe
(a) Varies directly as the velocity
(b) Varies inversely as the square of the velocity
(c) Varies inversely as the square of the diameter
(d) Varies approximately as the square of the velocity.

58. In laminar flow through a pipe discharge varies
(a) linearly as the viscosity
(b) as the square of radius
(c) inversely as the pressure drop
(d) inversely as the viscosity.

59. The hydraulic grade line is
(a) always above the energy grade line
(b) the velocity head below the energy grade line
(c) always above the closed conduit
(d) always slopping downward in the direction of flow.

60. In parallel-pipe problems
(a) the discharge is he same through all the pipes
(b) the head losses through each pipe are added to obtain the total head loss
(c) the head loss is the same through each pipe
(d) a direct solution gives the flow through each pipe when the total flow is known.

61. In networks of pipes
(a) the head loss around each elementary circuit must be zero

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) the head loss in all circuits is the same
(c) the elevation of hydraulic grade line is assumed for each junction
(d) elementary circuits are replaced by equivalent pipes.

62. The growth of boundary layer is supported when
(a) $(\partial p/\partial x)$ is positive (b) $(\partial p/\partial x)$ is zero
(c) $(\partial p/\partial x)$ is negative

63. The displacement thickness of boundary layer is
(a) the distance to the point where $(v/V) = 0.99$
(b) the distance the main flow is shifted from the boundary
(c) the distance where $v = V_*$.
(d) one half the actual thickness of the boundary layer.

64. Turbulent boundary layer thickness is proportional to
(a) $1/x^{1/5}$ (b) $x^{1/5}$
(c) $x^{2/5}$ (d) $x^{4/5}$

65. Boundary layer separation is caused by
(a) release of bubbles from the fluid when the pressure goes below the vapour pressure
(b) an adverse pressure gradient
(c) reduction of pressure gradient to zero
(d) the boundary layer thickness reducing to zero value.

66. For a uniform laminar flow in a channel with depth of flow D, the point velocity is equal to mean velocity of flow at a depth d below the free surface which is equal to
(a) 0.5 D (b) 0.577D
(c) 0.675 D (d) 0.707 D.

67. The terminal velocity of a small sphere settling in a viscous fluid varies as the
(a) inverse of the fluid viscosity
(b) inverse square of the diameter
(c) inverse of the diameter
(d) first power of its diameter.

68. For laminar flow in a pipe of circular cross-section, the Darcy friction factor, f, is
(a) independent of Reynolds number but dependent on pipe wall roughness
(b) inversely proportional to Reynolds number and independent of pipe wall roughness
(c) directly proportional to Reynolds number and independent of pipe wall roughness
(d) independent of radial distance from pipe axis.

69. The Prandtl mixing length is
(a) a universal constant
(b) zero at the pipe wall
(c) independent of shear stress
(d) independent of radial distance from pipe axis.

70. The value of friction factors for smooth pipes for Reynolds number equal to 10^6 is approximately
(a) 0.0001 (b) 0.001
(c) 0.01 (d) 0.1

71. For hydrodynamically rough boundary the friction coefficient
(a) is constant
(b) varies inversely with Reynolds number
(c) is function of Reynolds number and relative roughness
(d) is dependent on relative roughness only.

72. In open-channel flow
(a) the energy grade line coincides with the free surface
(b) hydraulic grade lines coincide
(c) the hydraulic grade line can never rise
(d) the hydraulic grade line and free surface coincide

73. Tranquil flow must always occur
(a) above normal depth
(b) below normal depth
(c) above critical depth
(d) below critical depth

74. Flow at critical depth occurs when
(a) the specific energy is a maximum for a given discharge
(b) the normal depth and critical depth coincide for a channel

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(c) the velocity is given by $\sqrt{2gh}$

(d) any change in depth requires more specific energy.

75. Critical depth in a channel is expressed by
(a) $(Q^2T/gA^3) = 1$ (b) $(QT^2/gA^3) = 1$
(c) $(QT/gA^3) = 1$ (d) $(Q^2A^3/gT^3) = 1$.

76. Conveyance of a channel section is directly proportional to
(a) discharge
(b) area of cross-section
(c) bed-slope
(d) Manning's coefficient

77. For most economical triangular channel section the ratio of its hydraulic radius R and depth of flow y is equal to
(a) $\frac{1}{2}$ (b) $1/\sqrt{2}$
(c) $1/2\sqrt{2}$ (d) $1/4\sqrt{2}$

78. Hydraulic jump is used for
(a) increasing the depth of flow
(b) reducing the energy of flow
(c) decreasing the velocity of flow
(d) reducing turbulence.

79. Hydraulic jump occurs when
(a) flow is supercritical
(b) flow is subcritical
(c) flow is supercritical and downstream depth is adequate

80. An elementary wave can travel upstream in a channel with 1.0 m depth and 2 m/sec velocity, with a velocity
(a) 5.132 m/sec (b) 1.132 m/sec
(c) 3.132 m/sec (d) 2.132 m/sec.

81. The speed of propagation of a small amplitude wave in quiet water with depth y is
(a) $(2gy)^{1/2}$ (b) $(gy^2)^{1/3}$
(c) $(gy/2)^{1/2}$ (d) $(gy)^{1/2}$

82. The discharge per metre width in a very wide rectangular channel is $\sqrt{9.81}$ m^3/sec. The critical depth for this flow is
(a) 1 m (b) $(9.81)^{1/4}$ m
(c) $(9.81)^{1/3}$ m (d) $(9.81)^{1/2}$ m.

83. For the channel of Q. 82 if Manning's n = 0.01, the critical slope of the channel is
(a) 0.981 (b) 0.0981
(c) 0.00981 (d) 0.000981

84. When the slope of an open channel changes from steep to mild
(a) a hydraulic jump is always formed in the steep slope reach
(b) a hydraulic jump is always formed in the mild slope reach
(c) a hydraulic jump may occur on either the steep slope reach or the mild slop reach depending on the relative values of the sequent depth and the normal depth on the mild slope reach
(d) no hydraulic jump is formed in any of the two reaches.

85. The head loss in a pipe running full is dependent on length of pipe, diameter of pipe, velocity of flow, roughness height of pipe walls, density and dynamic viscosity of the fluid. The number of dimensionless parameters that may be formed are
(a) 6 (b) 4
(c) 3 (d) 1

86. Reynolds number may be defined as the ratio of
(a) viscous forces to inertial forces
(b) elastic forces to pressure forces
(c) inertial forces to viscous forces
(d) gravity forces to inertial forces.

87. Mach number is ratio of inertia forces to
(a) pressure forces
(b) elastic forces
(c) surface tension forces
(d) gravity forces.

88. The scale ratio for acceleration according to Froude model law is
(a) g_r (b) $L_r^{1/2} g_r^{1/2}$
(c) $\mu_r \rho_r^{-1} L_r^{-1}$ (d) $L_r\, g_r$.

89. The wake
(a) is a region of high pressure

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(b) always occurs after a separation point
(c) is the principal cause of skin friction
(d) always occurs when deformation drag predominates

90. Pressure drag results from
(a) skin friction
(b) deformation drag
(c) occurrence of a wake
(d) development of stagnation point.

91. The lift on a body immersed in a fluid stream is
(a) due to buoyant force
(b) always in the opposite direction to gravity
(c) the resultant fluid force of the body
(d) the dynamic fluid-force component exerted on the body normal to the approach velocity.

92. Body A has twice the submerged weight, twice the projected area and twice the drag coefficient of body B. The ratio of the terminal fall velocities of the two bodies in air would be
(a) $1/\sqrt{2}$ (b) $\sqrt{2}$
(c) 2 (d) 8

93. A cylinder is kept on a horizontal boundary past which ideal fluid flows perpendicular to cylinder axis. It will then experience
(a) no lift
(b) lift in the vertical downward direction
(c) lift in the vertical upward direction
(d) lift in the direction inclined at 45° to the vertical.

94. The drag of a body moving in air at supersonic velocity
(a) is reduce by streamlining of the rear
(b) is increased by streamlining of the rear
(c) remains practically unchanged by streamlining of the rear
(d) is increased by streamlining of the front.

95. The reversible polytropic process is
(a) adiabatic frictionless
(b) given by p/ρ = constant
(c) given by p/ρ^k = constant
(d) given by p/ρ^n = constant.

96. Select the expression that does not given the speed of a sound wave
(a) $\sqrt{kRT}$ (b) $\sqrt{p/\rho^k}$
(c) $\sqrt{dp/d\rho}$ (d) $\sqrt{K/\rho}$

97. A normal shock wave
(a) is reversible
(b) may occur in a converging tube
(c) is irreversible
(d) is isentropic.

98. A normal shock wave is analogous to
(a) an elementary wave in still liquid
(b) the hydraulic jump
(c) subcritical flow in open channel
(d) flow of liquid through an expanding nozzle.

99. Hot-wire anemometer is used to measure
(a) temperature of gases
(b) velocity of liquid
(c) velocity of gases
(d) pressure of gases.

100. Effect of compressibility of fluid can be neglected if Mach number is
(a) equal to 1
(b) greater than 1
(c) less than 1 but greater than 0.4
(d) less than 0.4.

101. For flow through a converging-diverging tube
(a) if the Mach number at exit is greater than unity no shock wave has developed in the tube
(b) the Mach number at the throat is always unity
(c) if the critical pressure ratio is exceeded, Mach number at the throat is greater than unity
(d) For sonic velocity at the throat, one and only one pressure or velocity can occur at a given section downstream.

102. The force exerted by a jet on a curved plate is
(a) less than that on flat plate
(b) equal to that on a flat plate

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(c) more than that on a flat plate
(d) sometimes more and sometimes less than that on a flat plate

103. An impulse turbine
(a) always operates submerged
(b) makes use of a draft tube
(c) is most suited for low-head installations
(d) operates by initial complete conversion to kinetic energy.

104. Francis turbine is
(a) an impulse turbine
(b) a reaction turbine
(c) a tangential flow turbine
(d) an axial flow turbine.

105. Kaplan turbine is
(a) a high head axial flow turbine
(b) a mixed flow turbine
(c) a low head axial flow turbine
(d) an impulse turbine.

106. The specific speed of a turbine is given by

(a) $\frac{N\sqrt{P}}{H^{3/2}}$ (b) $\frac{N\sqrt{P}}{H^{5/4}}$

(c) $\frac{N\sqrt{P}}{H^{3/4}}$ (d) $\frac{N\sqrt{P}}{H^{5/2}}$

107. The specific speed of a pump is defined as the speed of a unit
(a) of unit size with unit discharge at unit head
(b) of such a size that it requires unit power at unit head
(c) of such a size that it delivers unit discharge at unit head
(d) of such a size that it delivers unit discharge at unit power.

108. Cavitation is caused by
(a) high velocity
(b) low barometric pressure
(c) low pressure
(d) high pressure.

109. If p is the absolute pressure at point, the cavitation parameter is defined by

(a) $\frac{P_v - p}{\rho V^2/2}$ (b) $\frac{p_{atm} - p_v}{\rho V^2/2}$

(c) $\frac{p - p_v}{wV^2/2}$ (d) $\frac{p - p_v}{\rho V^2/2}$

110. Unit power of a turbine is given by
(a) $P/H^{1/2}$ (b) $P/H^{3/2}$
(c) $P/H^{3/4}$ (d) $P/H^{5/2}$.

111. The following two relationship are necessary for homologous turbine units
(a) H/ND^3 = constant; Q/N^2D^2 = constant
(b) $Q/D^2\sqrt{H}$ = constant; H/N^2D^2 = constant
(c) p/QH = constant; H/N^2D^2 = constant
(d) $N\sqrt{Q}/H^{3/2}$ = constant;
$N\sqrt{P}/H^{3/4}$ = constant.

112. In a hydraulic ram
(a) Rankine efficiency is less than D, Aubuisson efficiency
(b) Rankine efficiency is equal to D, Aubuission efficiency
(c) Rankine efficiency is more than D, Aubuisson efficiency

113. An intensifier is used for
(a) increasing velocity
(b) increasing pressure
(c) decreasing velocity
(d) decreasing pressure.

114. If a hydro plant operate under a head of 100 m, it may be classified as
(a) a very high speed plant
(b) high head plant
(c) medium head plant
(d) low head plant.

115. Run-of-river plants are usually
(a) low head plants
(b) medium head plants
(c) high head plants
(d) very high head plants.

116. Two tubro-generators each of capacity 20000 kW have been installed at a hydel power station. During a certain period if the load on the hydel plant varies from 10000 kW to 30000 kW, the utilization factor is

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) 50% (b) 75%
(c) 80% (d) 100%

117. Hydraulic jump is expected when slope of a channel
(a) changes from mild to steep
(b) changes from steep to steeper
(c) changes from steep to mild
(d) changes from mild to milder.

118. Flow has Froude number less than one
(a) if normal depth is less than critical depth
(b) if normal depth is more than critical depth
(c) if normal depth is equal to critical depth

119. Flow net can be drawn
(a) for irrotational flow only
(b) for rotational flow only
(c) for both irrotational nor rotational flow
(d) for neither irrotational nor rotational flow.

120. The velocity distribution in a pipe is parabolic for
(a) uniform turbulent flow
(b) non-uniform turbulent flow
(c) uniform laminar flow
(d) neither laminar nor turbulent flow.

121. For laminar flow in a pipe if the centre line velocity is 0.1 m/sec, the average velocity is
(a) 5 cm/sec (b) 10 cm/sec
(c) 15 cm/sec (d) 20 cm/sec

122. For laminar flow in a pipe the energy correction factor is
(a) 1.03 (b) 1.05
(c) 1.33 (d) 2.00

123. If the velocity components in the x and y direction are $u = 3x - 2xy$, $v = 1 - 2y^2$, then components of velocity in the z direction is
(a) $3z + 6yz$ (b) $-3z + 6yz$
(c) $-3z - 6yz$ (d) $3z - 6yz$.

124. Shear stress $\tau = A + B\,(\partial v/\partial y)^n$(for $n > 1$) shows
(a) a thixotropic substance
(b) a Newtonian fluid
(c) an ideal plastic
(d) an ideal fluid.

125. If the percentage error in measurement of depth in a triangular channel is $(\Delta y/y)$, the percentage error in the measurement of discharge $(\Delta Q/Q)$ is
(a) $(\Delta y/y)$ (b) 1.5 $(\Delta y/y)$
(c) 2.0 $(\Delta y/y)$ (d) 2.5 $(\Delta y/y)$.

126. The horizontal and vertical scale ratios are L_H and L_V respectively, the discharge scale ratio as per Froude model law will be
(a) $L_H^{1/2}L_V^2$ (b) $L_H L_V^{3/2}$
(c) $L_H^2 L_V^{1/2}$ (d) $L_H^3 L_V^{-1/2}$.

127. The scale ratio in a model of a spillway is 1:100. If the discharge in the prototype is 2000 m^3 sec, the discharge in the model is
(a) 20 m^3/sec (b) 2 m^3/sec
(c) 0.2 m^3/sec (d) 0.02 m^3/sec

128. $ML^{-1}T^{-2}$ is the dimension of
(a) Force (b) Pressure
(c) Energy (d) Power

129. If coefficient of contraction is C_c and h_1 and h_2 are the velocity heads at sections 1 and 2, the head loss due to sudden contraction is
(a) $\left(\frac{1}{C_c}-1\right)^2 h_1$ (b) $\left(\frac{1}{C_c}-1\right)^2 h_2$
(c) $\left(\frac{1}{C_c^2}-1\right) h_1$ (d) $\left(\frac{1}{C_c^2}-1\right) h_2$.

130. The relation $(\partial\tau/\partial y) = (\partial p/\partial y)$ is valid for laminar flow between two plates when
(a) both the plates are stationary
(b) both the plates are moving
(c) one plate is moving and the other one is stationary
(d) any of the above cases is considered.

131. The force scale ratio for Reynolds' model law using the same fluid in model and prototype is
(a) 1 (b) L_r^2
(c) L_r^3 (d) $L_r^{5/2}$

132. In turbulent flow, a smooth pipe and a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

rough pipe shall have the same friction factor if
(a) the flow is highly turbulent
(b) the friction factor is independent of Reynolds number
(c) the flow is in transition zone from smooth to rough pipe
(d) the Reynolds number is the same and the factor k/δ' is less than about 4

133. The friction velocity in a pipe is
(a) (τ_0/ρ) (b) $\sqrt{(\tau_0/\rho)}$
(c) $\sqrt{\tau_0/\mu}$ (d) $(\tau_0/\rho)^2$

134. The shear stress in a fluid flowing in a circular pipe
(a) is constant over the section
(b) is zero at the walls and increase linearly to the centre
(c) is zero at the centre and varies linearly with the radius
(d) varies parabolically across the section

135. The discharge scale ratio for Froude model law is
(a) $\sqrt{L_r}$ (b) L_r^2
(c) $L_r^{5/2}$ (d) L_r^3

136. In turbulent flow the friction factor f for a smooth pipe depends upon
(a) the Reynolds number and the relative roughness
(b) the relative roughness only
(c) the size of the pipe and the discharge only
(d) the Reynolds number only.

137. In turbulent flow the friction factor f for a rough pipe depends upon
(a) the relative roughness only
(b) the Reynolds number only
(c) the size of the pipe and the discharge only
(d) the Reynolds number and the relative roughness.

138. Mach angle is denoted by
(a) $\sin^{-1}$ (M) (b) $\sin^{-1}$ (1/M)
(c) cosec^{-1} (1/M) (d) $\tan^{-1}$ (1/M)

139. The critical pressure ratio for air under isentropic flow is
(a) 1.0 (b) 0.833
(c) 0.634 (d) 0.528

140. The most economical circular channel section giving maximum discharge is the one
(a) flowing with $y \approx 0.95$ diameter
(b) which gives maximum velocity
(c) which is flowing full
(d) corresponding to the least perimeter.

141. If a sphere of diameter 10 mm falls in castor-oil of kinetic viscosity 10 stokes, with terminal velocity of 15 mm/s, the coefficient of drag on the sphere is
(a) 10 (b) 100
(c) 160 (d) 200

142. The distance y from the pipe boundary at which the point velocity is equal to the average velocity for turbulent flow is
(a) 0.223 R (b) 0.323 R
(c) 0.423 R (d) 0.523 R
where R is the radius of the pipe.

143. In Couette flow for the condition of zero discharge the value of dimensionless pressure gradient $P = \left[\frac{B^2}{2\mu V}\left(-\frac{\partial p}{\partial x}\right)\right]$ is
(a) – 1 (b) – 3
(c) 1 (d) 3

144. A ship model of scale 1:100 has a wave resistance of 1 N at its design speed. The corresponding wave resistance in prototype will be
(a) 1 000 N (b) 10 000 N
(c) 100 000 N (d) 1 000 000 N

145. The excess pressure inside a soap bubble of radius r is
(a) $\frac{\sigma}{r}$ (b) $\frac{2\sigma}{r}$
(c) $\frac{3\sigma}{r}$ (d) $\frac{4\sigma}{r}$

146. A triangular channel has side slope of 2

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

horizontal to vertical and discharge of 0.1 m^3/s. the critical depth is

(a) 0.1195 m (b) 0.2195 m

(c) 0.3195 m (d) 0.1195 m

147. A turbine where in entire total energy of water available is converted to kinetic energy is called

(a) reaction turbine

(b) mixed flow turbine

(c) impulse turbine

(d) axial flow turbine.

148. A large quantity of water is required to be admitted into a Francis turbine of a certain diameter to develop power under a low head. This can be achieved by

(i) increasing B/D (= n) ratio only;

(ii) increasing flow ratio ψ only;

(iii) increasing both B/D ratio and flow ratio ψ;

(iv) keeping B/D ratio and flow ψ constant of this which is correct:

(a) only (i) (b) only (ii)

(c) only (iii) (d) only (iv)

149. For the design of the most economical trapezoidal channel section under the condition that the bottom width be kept constant which of the following is (are) true

(i) $z = \frac{1}{\sqrt{3}}$; (ii) $A = \frac{y^2}{z}$;

(iii) $\frac{B}{y} = \frac{1+z^2}{z}$; (iv) $\frac{B}{y} = \frac{1-z^2}{z}$

of these

(a) only (i) is correct (b) only (iii) is correct

(c) only (ii) is correct

(d) both (i) and (iii) are incorrect.

150. Consider the following statements:

(i) The velocity of flow in a channel at a depth of 0.6 y below the free surface is found to be very close to the mean velocity

(ii) The velocity of flow in an open channel is uniform across the section. The maximum velocity occurs at the free surface

(iii) A constant velocity section is one in which the velocity is constant for all depths of flow

(iv) The discharge in a small open channel can be measured by using a venturiflume or by constructing a weir across it.

Of these statements

(a) only (i) is correct

(b) only (i) and (ii) are correct

(c) only (i), (iii) and (iv) are correct

(d) all (i), (ii), (iii) and (iv) are correct.

151. For a wide rectangular channel the dynamic equation for Gradually Varied Flow (GVF) is given by

(a) $\frac{dy}{dx} = S_0 \frac{1-\left(\frac{y_n}{y}\right)^3}{1-\left(\frac{y_c}{y}\right)^3}$

(b) $\frac{dy}{dx} = S_0 \frac{1+\left(\frac{y_n}{y}\right)^3}{1-\left(\frac{y_c}{y}\right)^3}$

(c) $\frac{dy}{dx} = S_0 \frac{1-\left(\frac{y_n}{y}\right)^3}{1+\left(\frac{y_c}{y}\right)^3}$

(d) $\frac{dy}{dx} = S_0 \frac{1+\left(\frac{y_n}{y}\right)^3}{1+\left(\frac{y_c}{y}\right)^3}$

152. Consider the following for a hydraulic jump in a prismatic channel

(i) the specific energy decreases due to the jump formation

(ii) the specific force increases after the jump

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(iii) the specific energy increases due to the jump formation
(iv) the specific force remains constant.
Of these statements
(a) only (i) is correct
(b) only (ii) is correct
(c) both (i), (iv) are correct
(d) both (ii) and (iii) are correct.

153. The displacement thickness of a boundary layer is given by

(a) $L^* = \int_0^\infty \left(1 - \frac{v}{V}\right) dy$

(b) $L^* = \int_0^\infty \frac{v}{V}\left(1 - \frac{v}{V}\right) dy$

(c) $L^* = \int_0^\infty \frac{v}{V}\left(1 - \frac{v^2}{V^2}\right) dy$

(d) $L^* = \int_0^\infty \left(1 + \frac{v}{V}\right) dy$

154. Consider the statement:
Statement I: The same Bernoulli's equation is applicable to all the points in the flow field if the flow is irrotational.
Statement II: The value of constant in the equation is different for different streamlines if the flow is rotational.
Of these statements
(a) both I and II are correct
(b) both I and II are incorrect
(c) only I is correct
(d) only II is correct.

155. In river models
(a) Reynolds model law is the basic similitude criteria
(b) Froude model law is the basic similitude criteria
(c) Both Reynolds and Froude model laws are equally important
(d) Distortion is occasionally used.

156. The instrument suitable for measurement of small pressure differences in liquid is
(a) U-tube differential manometer
(b) Two piezometer manometer
(c) Diaphragm pressure gage
(d) Inverted U-tube manometer.

157. For hydrodyamically smooth boundary with usual notation

(a) $\frac{k}{\delta'} > 6.0$ (b) $\frac{k}{\delta'} = 6.0$

(c) $\frac{k}{\delta'} < 0.25$ (d) $\frac{k}{\delta'}$ = 0.25 to 6.0.

158. For hydrodynamically rough boundary with usual notation

(a) $\frac{k}{\delta'} > 6.0$ (b) $\frac{k}{\delta'} = 6.0$

(c) $\frac{k}{\delta'} < 0.25$ (d) 0.25 to 6.0.

159. Consider the following statements:
Statement I: When a nozzle is fitted at the end of a long pipeline the discharge increases.
Statement II: The velocity of flow at the nozzle end is more than that in the case of pipe without nozzle, the head in both the cases being same.
(a) both I and II are correct
(b) both I and II are incorrect
(c) only I is correct
(d) only II is correct.

160. **Assertion (A):** The coefficient of discharge for a mouthpiece is higher than that for an orifice.
Reason (R): The discharge through an orifice varies as $H^{1/2}$ whereas the discharge through a mouthpiece varies as $H^{3/2}$ (where H is the head causing the flow in both cases)
Select the correct answer.
(a) A is true but R is not the correct explanation of A
(b) A is true but R is the correct explanation of A
(c) A is true but R is false
(d) Both A and R are false.

161. **Assertion (A):** A Cipolletti weir may be considered as a rectangular weir without end contractions.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Reason (R): The side slopes are so provided that the effect of end contractions is compensated.

Select the correct answer.

(a) Both A and R are true but R is not the correct explanation of A
(b) Both A and R are true and R is the correct explanation of A
(c) A is true but R is false
(d) A is false but R is true.

162. Assertion (A): The equation $\tau = -\frac{\partial p}{\partial x}\frac{r}{2}$ is applicable for only laminar flow through circular pipes where τ is shear stress at distance r from centre and $(\partial p/\partial x)$ is pressure gradient in the direction of flow.
Reason (R): In the derivation of this equation the flow in pipes is assumed to be laminar.

Select the correct answer.

(a) A is true but R is not the correct explanation of A
(b) A is true and R is the correct explanation of A
(c) A is true but R is false
(d) Both A and R are false.

163. Assertion (A): Bernoulli's equation is applicable to any point in the flow field provided the flow is steady and irrotational.
Reason (R): The integration of Euler's equation of motion to derive Bernoulli's equations involves the assumptions that velocity potential exists and the conditions do not change with time at any point.

Select the correct answer.

(a) Both A and R are true but R is not the correct explanation of A
(b) Both A and R are true and R is the correct explanation of A
(c) A is true but R is false
(d) A is false but R is true.

164. For a Borda's mouthpiece running full the value of coefficient of discharge is approximately equal to

(a) 0.5 (b) 0.6
(c) 0.7 (d) 0.8

165. For a Borda's mouthpiece running free the value of coefficient of discharge is approximately equal to

(a) 0.5 (b) 0.6
(c) 0.7 (d) 0.8

166. For the design of the most economical trapezoidal channel section under the condition that the depth of flow be kept constant which of the following is true.

(a) $z = \frac{1}{\sqrt{3}}$ (b) $\frac{B}{y} = \frac{1+z^2}{z}$

(c) $A = \frac{y^2}{z}$ (d) $\frac{B}{y} = \frac{1-z^2}{z}$

167. Match List I and List II and select the correct answer using the code given below:

List I	*List II*
A. Stokes' law	1. Wake
B. Logarithmic velocity distribution	2. Laminar flow past a sphere
C. Parabolic velocity distribution	3. Turbulent flow in pipe
D Form drag	4. Laminar flow in pipes

	A	**B**	**C**	**D**
(a)	1	2	3	4
(b)	2	3	4	1
(c)	2	4	3	1
(d)	4	3	1	2

168. Which of the following pairs is not correctly matched?

(a) Stability of floating body — Metacentre
(b) Flownet — Streamline
(c) Manning's equation — Channel flow
(d) Froude number — Viscosity

169. Which of the following pairs is correctly matched?

(a) Prandtl — Flow through channels
(b) Continuity equation — Law of conservation of energy
(c) Karman Vortex Street — Flow past a cylinder
(d) Mixing length — Laminar flow

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

170. Match List I and List II and select the correct answer using the code given below:

List I	List II
A. Specific force	1. Head loss due to friction
B. Specific energy	2. Rapidly varied flow
C. Hydraulic jump	3. Alternate depths
D. Darcy-Weisbach equation	4. Conjugate depths.

	A	B	C	D
(a)	1	2	3	4
(b)	4	3	2	1
(c)	2	3	4	1
(d)	3	4	1	2

171. The speed of an elementary wave in a still liquid is given by
(a) $(gy^2)^{1/3}$ (b) $2y/3$
(c) $\sqrt{2gy}$ (d) $\sqrt{gy}$

172. The pitot-static tube measures
(a) dynamic pressure
(b) static pressure
(c) total pressure
(d) different in static and dynamic pressure.

173. The viscosity of gases
(a) increases with decrease in temperature
(b) increases with increase in temperature
(c) does not change with change in temperature
(d) increases upto certain temperature and then decreases.

174. The dimensions of dynamic viscosity are
(a) ML T^{-1} (b) ML^{-1} T
(c) ML^{-1} T^{-1} (d) ML^{-2} T

175. Froude number is a significant parameter in the study of
(a) water hammer
(b) hydraulic jump
(c) flow through pipe
(d) flow of compressible fluids

176. In a capillary rise in a glass tube the weight of the liquid raised is supported by
(a) the atmospheric pressure
(b) the friction on the walls of the tube
(c) both the atmospheric pressure and friction on the walls of the tube
(d) vertical component of surface tension.

177. A vertical gate 4 m wide and 6 m deep holds liquid (sp.wt.w) with free surface at its top. The moment of the total pressure about the bottom of the gate is
(a) 64 w (b) 128 w
(c) 144 w (d) 288 w

178. For a plane lamina immersed in a fluid the centre of pressure is always
(a) located at the centroid of the lamina
(b) located above the centroid of the lamina
(c) independent of the centroid of the lamina
(d) located at the centroid of the pressure prism.

179. Streamlines representing a fluid flow field are the lines along which
(a) the velocity of flow is constant
(b) the velocity potential is constant
(c) the stream function is constant
(d) both the velocity potential and the stream function are constant

180. A pipe of diameter D is to be replaced by n pipes each of diameter d laid in parallel. The value of d is given by
(a) $d = (D/n)$ (b) $d = (D/n^{2/5})$
(c) $d = (D/n^{1/2})$ (d) $d = (D/n^{3/2})$

181. At a sudden expansion in a horizontal pipe
(a) total energy line rises in the direction of flow
(b) velocity head increasing in the direction of flow
(c) hydraulic grade line rises in the direction of flow
(d) total energy line is below hydraulic grade line

182. In a fluid flow, point A is at a higher elevation than point B and the head loss between these points is H_L. The total head at A and B are H_A and H_B respectively. The flow will take place
(a) from B to A if $H_A + H_L = H_B$
(b) from A to B if $H_A + H_L = H_B$
(c) from B to A if $H_B + H_L = H_A$
(d) always from A to B

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

183. In a pipeline the hydraulic grade line is above the pipe centre line in the longitudinal section at a point A and below the pipe centre line at another point B. From this it can be inferred that
(a) negative pressure prevails at A and positive pressure prevails at B
(b) positive pressure prevails at A and negative pressure prevails at B
(c) positive pressure prevails at both A and B
(d) negative pressure prevails at both A and B

184. A highly viscous liquid is flowing with velocity V in a pipe of diameter d and length L. If Re is Reynolds' number and g is acceleration due to gravity, the head loss h_L is given by

(a) $h_L = \text{Re}\dfrac{L}{d}\dfrac{V^2}{2g}$ (b) $h_L = \dfrac{\text{Re}}{16}\dfrac{L}{4d}\dfrac{V^2}{2g}$

(c) $h_L = \dfrac{16}{\text{Re}}\dfrac{4L}{d}\dfrac{V^2}{2g}$ (d) $h_L = \dfrac{16}{\text{Re}}\dfrac{L}{4d}\dfrac{V^2}{2g}$

185. A weir with a finite crest width B is termed as broad crested weir when the upstream head H above the crest of the weir is related to B as
(a) $(H/B) > 1.6$ (b) $0.4 \le (H/B) \le 1.6$
(c) $0.1 \le (H/B) \le 0.4$ (d) $(H/B) < 0.1$

186. Two pipe systems are said to be equivalent in their lengths when
(a) the friction factor and discharge are same in both the systems
(b) the same head loss is produced by the same discharge in both the systems
(c) both the systems are in series
(d) the discharge and diameter are same in both the systems

187. In a horizontal pipe of diameter D and mean velocity of flow V, if Δp is the pressure difference between two sections L distance apart, then wall shear stress τ_0 is given by
(a) $\tau_0 = (\Delta pD/4L)$ (b) $\tau_0 = (\Delta pL/D)$
(c) $\tau_0 = (\Delta pLV^2/8)$ (d) $\tau_0 = V(\Delta pL/8)^{1/2}$

188. The drag on a very small sphere falling in a highly viscous liquid varies
(a) inversely with the velocity
(b) directly with the velocity
(c) as the square root of the velocity
(d) as the square of the velocity

189. A Prandtl pitot tube attached to the nose of an aircraft, gave a pressure coefficient of 1.04. The aircraft is flying at a Mach number of
(a) 0.04 (b) 0.4
(c) 1.04 (d) 2.08

190. If a multijet Pelton turbine has n number of jets, then its specific speed is directly proportional to
(a) n (b) $n^{3/4}$
(c) $n^{1/2}$ (d) $n^{5/4}$

191. Match List I with List II and select the correct answer using the codes given below the lists: (y_n = Normal depth y_c = Critical depth and y = depth of gradually varied flow)

List I (*Flow Regimes for gradually varied flow*)	*List II* (*Type of gradually varied flow profile*)
A. $y_c > y_n > y$	1. C_1
B. $y_c < y_n < y$	2. M_1
C. $y_n > y > y_c$	3. S_3
D. $y > y_c > y_n$	4. M_2
	5. S_1

	A	B	C	D
(a)	1	2	3	4
(b)	3	4	5	2
(c)	3	2	4	5
(d)	5	4	3	1

192. A liquid of density ρ and bulk modulus K flows with a mean velocity V in a long rigid pipe of diameter D. A sudden closure of a valve at the end of the pipe produces a maximum water hammer head h_w which is equal to

(a) $\dfrac{V\sqrt{\rho}}{g\sqrt{K}}$ (b) $\dfrac{V\sqrt{K}}{g\sqrt{\rho}}$

(c) $\dfrac{V\sqrt{g}}{\rho\sqrt{K}}$ (d) $\dfrac{V\rho}{\sqrt{gK}}$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

193. In a laminar flow through a circular pipe of diameter 200 mm, the maximum velocity is found to be 1 m/s. The velocity at a radial distance of 50 mm from the axis of the pipe will be
(a) 0.25 m/s (b) 0.50 m/s
(c) 0.75 m/s (d) 1.00 m/s

194. An aeroplane having a wing span of 16 m and chord of 2.5 m weights 110 kN. If it gets airborne at a velocity of 300 km per hour, in air of mass density 1.2 kg/m^3, then the coefficient of lift is nearly
(a) 0.8 (b) 0.7
(c) 0.6 (d) 0.5

195. A harbour model has a horizontal scale of 1/150 and a vertical scale of 1/60. The interval between successive high tides in the model will be nearly
(a) 5 hour (b) 90 min
(c) 40 min (d) 15 min

196. The head over a V-notch at the end of a channel is 0.75 m. If an error of 1.5 mm is possible in the measurement of the head, then he percentage error in computing the discharge will be
(a) 0.3 (b) 0.5
(c) 1.0 (d) 1.5

197. The head over a rectangular notch at the end of a channel is 0.5 m. If an error of 1 mm is possible in the measurement of the head, then the percentage error in computing the discharge will be
(a) 0.3 (b) 0.5
(c) 1.0 (d) 1.5

198. Which of the following rules are used in choosing the repeating variables in dimensional analysis?
(a) Repeating variable should include the dependent variable
(b) Repeating variable should combine among themselves to from a dimensionless group
(c) Repeating variable should contain at least one dimensionless parameter
(d) Repeating variable should contain all the primary units used in describing the variables in the question.

199. When a certain discharge is flowing in a horizontal pipe, a mercury-water U-tube manometer connected to the entrance and throat of a venturimeter fitted in the pipe recorded a deflection of 250 mm. If the same discharge flowed through the same pipe kept at an inclination of 45° to the horizontal, then the corresponding deflection recorded by the U-tube manometer will be
(a) 250 mm (b) 250 $\sqrt{2}$ mm
(c) 250/ $\sqrt{2}$ mm (d) 250/2 mm

200. A cylindrical vessel with a constant plane area of 1 m^2 is rotated about its vertical axis such that the liquid inside the vessel is about to spill. If the height of the vessel is 2 m and the height of the parabola 1 m, then the volume (in m^3) of the liquid in the vessel will be
(a) 2 (b) 1.5
(c) 1.0 (d) 0.5

201. Which one of the following velocity fields represents a possible fluid flow?
(a) $u = x; v = y$ (b) $u = x^2; v = y^2$
(c) $u = xy; v = x^2y^2$ (d) $u = x; v = -y$

202. The mean velocities at two ends of a stream tube 100 mm apart are 2.5 m/s and 3 m/s. The convective tangential acceleration mid-way is
(a) zero (b) 12.50 m/s^2
(c) 13.75 m/s^2 (d) 15.00 m/s^2

203. At room temperature, the dynamic and kinematic viscosity of water
(a) are both greater than that of air
(b) are both less than that of air
(c) are respectively greater than and less than that of air
(d) are respectively less than and greater than that of air

204. An open rectangular wagon 5 m long partly filled with water is subjected to an acceleration of 1.96 m/s^2. For no spilling of water the level below the open top upto which water may be filled is
(a) 0.5 m (b) 0.6 m
(c) 0.7 m (d) 0.8 m

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

205. The discharge of 3 m^3/s flows in a canal, 2 m wide, at a depth of 1.2 m. If the width of the canal is reduced to 1.5 m by a canal transition, then neglecting losses, the depth of flow after the contraction will be
(a) 1.12 m (b) 1.20 m
(c) 1.28 m (d) 1.60 m

206. The Hardy Cross method of hydraulic analysis of pipe networks, besides satisfying the continuity and energy principles, must also satisfy the condition that
(a) algebraic sum of the head losses around any closed loop is zero
(b) flow into any junction equals the out flow from it
(c) flow in each pipe has head loss according to Darcy Weisbach or any other pipe head loss equation
(d) momentum principle is followed

207. One end of a two-dimensional water tank has the shape of a quadrant of a circle of radius 2 m. When the tank is full, the vertical component of the force per unit length on the curved surface will be
(a) 9.81π kN (b) 19.62π kN
(c) 29.43π kN (d) 39.24π kN

208. An irrigation canal has a steady discharge Q at a section where a cross-regulation (gated) is provided for control purposes. If the gate of the regulator, which is normally fully open, is suddenly lowered down to a half-open position, then a rapidly varied unsteady flow results, both upstream and downstream of the gate. In such a case, it would take the form of a
(a) +ve surge moving upstream and a –ve surge moving downstream
(b) +ve surge moving downstream and a –ve surge moving upstream
(c) +ve surge moving upstream and a +ve surge moving downstream
(d) –ve surge moving upstream and a –ve surge moving downstream

209. A solid cylinder of length L, diameter D and specific gravity 0.6 floats in netural equilibrium in water with its axis vertical. What is the ratio of L to D ?
(a) $\frac{\sqrt{3}}{2}$ (b) $\frac{2\sqrt{3}}{5}$
(c) $\frac{4}{5\sqrt{3}}$ (d) $\frac{5}{4\sqrt{3}}$

210. Match List I (Causes) and List II (Effects) pertaining to rotation and vertical horizontal motion of liquid masses at constant acceleration and select the correct answer using the codes given below the lists :

List I (*Causes*)	*List II* (*Effects*)
A. The form of the free surface of liquid in a rotating open vessel	1. An inclined plane
B. The form of surface of the liquid in a tanker moving	2. A paraboloid of revolution
C. The pressure at any point in a container's base moving with upward acceleration	3. Greater than the hydrostatic value
D. The pressure at any point in a container's base moving with downward acceleration	4. Less than the hydrostatic value

Codes :

	A	B	C	D
(a)	4	1	3	2
(b)	4	3	1	2
(c)	2	3	1	4
(d)	2	1	3	4

211. For a turbulent boundary layer (under zero pressure gradient), the velocity profile is described by the one-fifth power law. What is the ratio of displacement thickness to boundary layer thickness ?
(a) $\frac{1}{7}$ (b) $\frac{1}{6}$
(c) $\frac{1}{5}$ (d) $\frac{1}{4}$

212. Oil ($\mu = 0.44$ P_a.s, $\rho = 888$ kg/m^3) is filled in the space between two parallel plates which are 18 mm apart. The upper plate is moving

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

at a velocity of 4 m/s. What is the shear stress on the plate ?

(a) 97.8 *Pa* (b) 48.9 *Pa*
(c) 79.8 *Pa* (d) 87.9 *Pa*

213. A 1 : 30 scale model of a submarine is to be tested in a wind tunnel for its drag when it is operating at 15 km/h in ocean. The kinematic viscosity of air is 1.51×10^{-5} m^2/s and for water 1.02×10^{-6} m^2/s. What is the velocity of air in wind tunnel that should be maintained for kinematic similarity ?

(a) 1500 m/s (b) 2000 m/s
(c) 1850.5 m/s (d) 2500.5 m/s

214. Two long pipes in parallel are used to carry water between two reservoirs. The diameter of one pipe is twice that of the other. Both the pipes have the same value of friction factor neglect minor losses. What is the ratio of flow rates through the two pipes ?

(a) 2.8 (b) 5.6
(c) 8 (d) 11.3

215. Consider the following statements regarding the conditions to be satisfied for the maximum discharge through a trapezoidal channel section with side slope 1 : n, bed width b, flow depth d and having a fixed bed slope.

1. Slopping sides should have an angle of 30° with vertical.
2. Hydraulic means depth equals half the flow depth.
3. Length of sloping side should be equal to twice the bottom width.

Which of the statements given above are correct

(a) 1, 2 and 3 (b) 1 and 2
(c) 2 and 3 (d) 1 and 3

216. In a reach of open channel flow with $q = 2$ m^2/s, depth of flow at section A is 1.5 m and at section B, 1.6 m. The difference in specific energies at two sections can be correspondingly taken as 0.09 m. The bed slope is 1 in 2000. The mean energy slope between the sections is given as 0.003. What is the Length of reach AB?

(a) 54 m with A on the upstream
(b) 54 m with B on the upstream
(c) 36 m with A on the upstream
(d) 36 m with B on the upstream

217. A centrifugal pump discharges 260 litres of water per second when running at 600 rpm. The impeller diameter at the outlet is 80 cm. It develops a head of 15.3 m. What is the approximate minimum starting speed ?

(a) 425 rpm (b) 450 rpm
(c) 475 rpm (d) 500 rpm

218. A combination of centrifugal pumps of specific speed 500 and overall efficiency 80% running at 800 rpm is to be used to pump 40 lps of water to a height of 75 m. What should be the arrangement ?

(a) 3 pumps in series
(b) 3 pumps in parallel
(c) 4 pumps in series
(d) 4 pumps in parallel

219. Match List I (Machines) with List II (Associated with) and select the correct answer using the codes given below the lists :

List I (*Machines*)	*List II* (*Associated with*)
A. Centrifugal pump	1. Percent slip
B. Reciprocating pump	2. Bucket
C. Francis turbine	3. Guide blade
D. Pelton wheel	4. Volute chamber

Codes :

	A	B	C	D
(a)	4	3	1	2
(b)	4	1	3	2
(c)	2	3	1	4
(d)	2	1	3	4

220. A pelton wheel operates at 630 rpm taking 3 m^3/s of water under a head of 256 m with a speed ratio of 0.48 (Given $\sqrt{19.62} = 4.43$). What is the diameter of the impeller ?

(a) 0.90 m (b) 1.03 m
(c) 1.42 , (d) 1.80 m

221. An impulse turbine of 3 m diameter is rated at 10000 kW at 300 rpm under a head of 500 m. The turbine is operated under the head of 400 m.

What is the power developed ?

(a) 14000 kW (b) 10000 kW

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(c) 9000 kW (d) 7000 kW

222. An impulse turbine of 3 m diameter is rated at 10000 kW at 300 rpm under a head of 500 m. The turbine is operated under the head of 400 m. What is the speed at which it would run ?
(a) 419 rpm (b) 335 rpm
(c) 268 pm (d) 215 rpm

223. **Assertion (A) :** At the critical state of flow, the specific force is a minimum for the given discharge.
Reason (R) : For a minimum value of specific force, the first derivative of force with repsect to depth should be unity.
Select the correct answer.
(a) Both A and R are individually true and R is the correct explanation of A.
(b) Both A and R are individually true but R is not the correct explanation of A.
(c) A is true but R is false.
(d) A is false but R is true.

224. **Assertion (A) :** The vertical co-ordinate of the apex of the Q-curve is $\frac{2}{3}$ E.
Reason (R) : The horizontal co-ordinate of the apex of the E-curve is 1.5 times $\left(\frac{q^2}{g}\right)^{1/3}$.
Select the correct answer :
(a) Both A and R are individually true and R is the correct explanation of A.
(b) Both A and R are individually true but R is not the correct explanation of A.
(c) A is true but R is false.
(d) A is false but R is true.

225. **Assertion (A) :** The relative velocity of water of a Petton Turbine equals the difference between the absolute velocities of water and vane.
Reason (R) : For a Pelton Turbine the inlet blade angle is 180°.
Select the correct answer :
(a) Both A and R are individually true and R is the correct explanation of A.
(b) Both A and R are individually true but R is not the correct explanation of A.
(c) A is true but R is false.
(d) A is false but R is true.

226. As the depth of immersion of a vertical plane surface increases the location of centre of pressure
(a) Falls closer to the centre of gravity of the area.
(b) Moves away from the centre of gravity of the area.
(c) Ultimately coincides with the centre of gravity of the area.
(d) Falls much below the centre of gravity of the area.

227. Which of the following fluids can be classified as non-Newtonian ?
1. Kerosene oil 2. Diesel oil
3. Human blood 4. Tooth paste
5. Water
Select the correct answer using the codes given below :
(a) 1 and 2 (b) 3 and 4
(c) 2 and 5 (d) 1 and 5

228. Consider the following statements related to buoyancy in fluid statics :
1. Principle of buoyancy is applicable both to floating bodies and to submerged bodies.
2. Archimedes formulated the first theory of byoyancy.
3. In analyzing buoyancy of a floating body it is assumed that the resultant vertical force passes through centre of pressure.
4. In a freebody diagram of a floating body summation of all horizontal force is taken as zero.
Which of these statements are correct ?
(a) 1, 3 and 4 (b) 1, 2 and 4
(c) 1, 2 and 3 (d) 2, 3 and 4

229. When a body moves through still water at a constant velocity of 4.5 m/s, the velocity of water at 0.8 m, ahead of the nose of the body is 3.0 m/s. What will be the difference in pressure between the nose and the point 0.8 m ahead of it ?

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(a) 875 N/m^2 (b) 1000 N/m^2
(c) 1125 N/m^2 (d) 1250 N/m^2

230. Match List I (Equation/Principle) with List II (Property) and select the correct answer using the codes given below the lists :

List I (*Equation/Principle*)	*List II* (*Property*)
A. Energy equation	1. Force on a moving vane
B. Continuity equation	2. Lawn sprinkler
C. Moment of momentum principle	3. Pressure at a point in a pipeline
D. Impulse momentum principle	4. Flow at two sections of a tapering pipe

Codes :

	A	B	C	D
(a)	4	3	2	1
(b)	3	4	1	2
(c)	4	3	1	2
(d)	3	4	2	1

231. Consider the following statements related to concept of continuity equation and the concept of control volume in deriving equation.
1. Continuity equation relates velocity component and density of the fluid at a point in a fluid flow.
2. Continuity equation assumes that no void occurs in the fluid and fluid mass is neither created nor destroyed.
3. The shape of control volume for deriving the equation of continuity is assumed to be a parallelopiped.
4. For incompressible fluids the equation of continuity does not contain the viscosity terms.

Which of these statements are correct ?
(a) 1, 2, 3 and 4 (b) 1 and 2
(c) 2, 3 and 4 (d) 1 and 4

232. Consider the following parameters related to fluid flow .
1. Vorticity 2. Velocity potential
3. Stream function

Which of these parameters exist both in rotational and irrotational flows ?
(a) 1 and 2 (b) 2 and 3
(c) 1 and 3 (d) 1, 2 and 3

233. Which one of the following statements is not correct ?
(a) Cavitation forms where the local pressure drops upto the vapout pressure condition due to increase in local velocity.
(b) Cavitation takes place in turbines, runners of pumps, spillways where local velocity increases considerably.
(c) Cavitation may be prevented by maintaining the ambient pressure relatively higher at probable points of occurrence by raising their levels.
(d) Boundaries likely to be affected by cavitation can be prevented against pitting by covering it with tough materials like stainless steel or rubber.

234. The stream function for a two-dimensional flow is given by $\psi = 2xy$. The resultant velocity at a point P (2, 3) is
(a) 8.45 units/s (b) 7.21 units/s
(c) 6.44 units/s (d) 5.18 units/s

235. A thin smooth plate 1 m wide and 2 m long is towed through water at a velocity of 2m/s Assuming that boundary layer remains laminar (kinematic viscosity = 10^{-6} m^2/s), then drag on each side of the plate is
(a) 5.3 N (b) 26.6 N
(c) 53 N (d) 72.5 N

236. The distorted models used in the hydraulic studies are those which
(a) have exaggeration of the vertical scale and horizontal scale.
(b) are not geometrically similar to the prototypes.
(c) have same vertical and horizontal scale as that of the prototypes.
(d) have same Froude number as that for the prototypes.

237. A Cipolletti weir discharges water with the head of water over the crest being 250 mm. If the head due to velocity of approach is 0.01 m, what will be the excess percentage of flow, when corrected for velocity of

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

approach as compared to when not so corrected ?

(a) 3.2 % (b) 4.2 %

(c) 5.3 % (d) 6.3 %

238. Consider the following statements related to a horizontal venturimeter :

1. The velocity of flow in the main pipe is greater and the pressure is lesser than that at the throat section.
2. The velocity of flow in the main pipe is lower and the pressure is larger than that at the throat section.
3. The pressure difference between the main pipe and throat section is positive.
4. The pressure difference between the main pipe and throat section is negative.

Which of these statements are not correct ?

(a) 1 and 4 (b) 1 and 2

(c) 2 and 4 (d) 3 and 4

239. Match List I with List II and select the correct answer using the codes given below the lists :

List I	*List II*
A. Flow developement	1. Surge tank
B. Pipe network	2. Entrance length
C. Water hammer	3. Darcy–Weisbach equation
D. Friction loss	4. Hardy–Cross method

Codes :

	A	B	C	D
(a)	2	4	1	3
(b)	4	2	3	1
(c)	2	4	3	1
(d)	4	2	1	3

240. Which one of the following conditions must be staisfied in the Hardy–Cross analysis of pipe network ?

(a) Continuity principle demands that flow into a network junction is equal to the flow out of it ?

(b) Momentum equation must be satisfied so that the force in each loop is balanced.

(c) Dracy–Weisbach head loss equation is to be used in computing head loss in elementary circuits. The equation is of the form $h_L = kQ^n$.

(d) $\Sigma h_L = \Sigma kQ^n$ must be equated to zero. If not, a flow correction factor ΔQ is to be made for arriving at a solution.

241. Which one of the following phenomena in a pipe flow is termed as water hammer ?

(a) The sudden rise of pressure in a, long pipe due to sudden closure of valve.

(b) The rise of pressure in a pipe due to gradual closure of valve.

(c) The rise of negative pressure.

(d) The zero pressure in a pipe flow.

242. Match List I with List II and select the correct answer using the codes given below the Lists :

List I	*List II*
A. The type of vortex flow in which no external torque rotates the fluid	1. The fall of liquid at the axis of rotation
B. Flow of liquid inside the impeller of a centrifugal pump	2. Free vortex flow
C. Free surface of a forced vortex flow	3. Parabolic
D. In forced vortex, the rise of liquid level at ends	4. Forced vortex flow

Codes :

	A	B	C	D
(a)	4	2	3	1
(b)	2	4	3	1
(c)	4	2	1	3
(d)	2	4	1	3

243. The approximate discharge over a 4 m long rectangular weir (with suppressed end contractions) with head over the crest as 0.36 m is

(a) 4.2 litres/s (b) 2.4 m^3/s

(c) 6.1 litres/s (d) 1.6 m^3/s

244. The steps involved in the procedure of computing length of backwater curves are given below with usual notations :

1. Insert all ΔH values and average value of slope in the differential equation.
2. Assume a new depth of flow y for the other end of the reach and insert

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

corresponding values of A, P, v, R, $\frac{v^2}{2g}$, H and S .

3. Determine or assume the initial conditions of depth, channel characteristics and discharge.
4. Compute average S value and the change in specific head ΔH.
5. Repeat the process for each reach adding resulting ΔL values to obtain total required distance ΔL until the depth or distance desired is reached.

The correct sequence of these steps is

(a) 1–2–3–4–5 (b) 3–1–5–2–4
(c) 3–2–4–1–5 (d) 1–4–5–2–3

245. Consider the following statements which relate to different types of water surface profiles. Curve types conform to usual classification y_c and y_n (critical and normal depths) :
1. Type-3 curves lie between y_c and y_n
2. All curves where $y_c < y_n$ are unaffected from any upstream disturbance.
3. All curves where $y_c > y_n$ are unaffecterd by downstream disturbances.
4. All curves approaching the y_n line approach it asymptotically except for C curve where $y_n = y_c$.

Which of these statements are correct ?

(a) 1, 2, 3 and 4 (b) 1, 3 and 4
(c) 2, 3 and 4 (d) 1 and 2

246. Flow duration curve is a plot of
(a) Flow against its time of occurrence in chronological order.
(b) Flow in ascending order against percentage time in chronological order.
(c) Flow that equalled or exceeded against percentage time.
(d) Flow against duration of time for which it is sustained.

247. Pondage in a hydropower station is defined as
(a) Impounding of considerabe amount of excess water during seasons of surplus flow.
(b) A regulating body of water in the form of relatively small amount of runoff to regulate flow variation in daily or weekly power requirements.
(c) Excess runoff to last for years.
(d) Excess runoff for a few hours only.

248. The specific speed of a turbine under a head of 150 m to develop 2000 h.p. while running at 300 rpm is

(a) 10–35 (b) 35–60
(c) 60–300 (d) 300–1000

249. Two geometrical pumps are running at the same speed of 1000 rpm and lifting water against the heads of 25 m and 16 m respectively. The first pump is having an impeller diameter of 300 mm. The impeller diamter of second pump shall be

(a) 192 mm (b) 240 mm
(c) 300 mm (d) 469 mm

250. In the case of semi-circular vanes, the theoretical maximum efficiency of the wheel can be

(a) 50% (b) 67%
(c) 75% (d) 100%

251. For a hydro-electric project with reaction turbine, the draft tube at the exit from the turbine is
(a) always immersed in water
(b) always above the water
(c) may either be above or below the water
(d) above or below the water depending on the unit speed of the turbine

252. For a hydraulically efficient rectangular channel section, the ratio of width to normal depth is

(a) 0.5 (b) 1.0
(c) 2.0 (d) $2\sqrt{3}$

253. Consider the following statements about a hydraulic ram :
1. Hydraulic ram does not need external power.
2. It works on the fundamental principle of water hammer.
3. The efficiency of a hydraulic ram is only of the order of 8 to 10%.
4. It can be termed as a low, intermitted discharge low-head pumping installation.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Which of these statements are correct ?
(a) 2 and 3 (b) 1, 2, 3 and 4
(c) 1 and 2 (d) 1 and 3

254. The ideal efficiency of a single-jet Pelton wheel is given by

$$\frac{2}{V^2}[Vu + u(V-u)\cos\phi - u^2]$$

(symbols have their usual meanings)
If friction on the buckets reduces the relative velocity by a factor k_1 and if windage losses reduces the effectiveness of the affected parameters by a factor k_2, then the resulting efficiency multiplied by V^2 is given by
(a) $2k_1k_2\,[Vu + u\,(V-u)\cos\phi - u^2]$
(b) $2\,[k_1\,Vu + k_2u\,(V-u)\cos\phi - u^2]$
(c) $2\,[k_2\,Vu + k_1\,k_2\,u\,(V-u)\cos\phi - u^2]$
(d) $2\,[k_2\,Vu + k_1k_2\,u\,(V-u)\cos\phi - k_2u^2]$

255. If the radius of the centrifugal pump impeller is reduced from 10 cm to 9 cm, the head developed by the pump will change from 10 m to
(a) 9 m of water (b) 8.1 m of water
(c) 9.487 m of water (d) 11.11 m of water

256. Match List I and List II and select the correct answer using the codes given below the lists :

List I (*Turbine*)	*List II* (*Specific speed*)
A. Pelton	1. 25
B. Propeller	2. 75
C. Kaplan	3. 500
D. Francis	4. 800
	5. 900

Codes :

	A	B	C	D
(a)	1	4	5	2
(b)	3	4	2	5
(c)	4	1	5	3
(d)	1	5	4	2

257. The Chezy's coefficient C is related to Darcy–Weisbach friction factor as
(a) $C = \sqrt{g/8f}$ (b) $C = \sqrt{8g/f^{1/4}}$
(c) $C = \sqrt{8g/f}$ (d) $C = \sqrt{f/8g}$

258. Consider the following statments regarding friction factor variation in a pipe flow :
1. In hydrodynamic rough boundaries, friction factor f is related to relative roughness (ε/D), (ε is roughness and D is the diameter of pipe).
2. In hydrodynamically smooth boundaries friction factor f depends both on Reynolds number Re and relative roughness.
3. If the friction factor $f = \frac{0.316}{\text{Re}^{1/4}}$, then the value of friction factor 'f' relates to that of the region of turbulent flow in smooth boundary.
4. Friction factor is directly related to the nature of boundary layer in a pipe flow.

Which of these statements are correct ?
(a) 1, 2 and 4 (b) 1, 2 and 3
(c) 2, 3 and 4 (d) 1, 3 and 4

259. The head-discharge characteristic curve of two centrifugal pumps in
1. parallel can be obtained by doubling the ordinates
2. series can be obtained by doubling the abscissa

Which of these statements is/are correct ?
(a) 2 alone (b) 1 alone
(c) Neither 1 nor 2 (d) 1 and 2

260. Consider the following statements :
1. Run-of-river plants can be located on any river.
2. Runaway speed of a turbine is generally 180% of the normal speed
3. Underground power stations are suited to areas susceptible to land slides.
4. Higher the specific speed, higher will be the discharge and head.

Which of these statements are correct ?
(a) 1 and 2 (b) 2 and 3
(c) 1 and 4 (d) 2 and 4

261. Consider the following statements:
1. Pumps in series operation allow the head to increase.
2. Pumps in series operation increase the flow rate.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

3. Pumps in parallel operation increase the flow rate.
4. Pumps in parallel operation allow the head to increase.

Which of these statements are incorrect ?

(a) 1 and 3 (b) 1 and 4
(c) 2 and 4 (d) 3 and 4

262. The maximum permissible suction lift for centrifugal pump in practice (at seal level at 30°C)

(a) 12 m (b) 10 m
(c) 6 m (d) 3 m

263. In the model of a highway bridge constructed to a scale of 1 : 25, the force of water on the pier was measured to be 0.5 kg(f) the force on the prototype pier will be

(a) 7501.5 kg(f) (b) 7622.5 kg(f)
(c) 7812.5 kg(f) (d) 7916.5 kg(f)

ANSWERS

1. (c)	2. (b)	3. (a)	4. (b)	5. (b)	6. (c)
7. (d)	8. (b)	9. (b)	10. (a)	11. (c)	12. (a)
13. (b)	14. (c)	15. (c)	16. (b)	17. (d)	18. (a)
19. (d)	20. (c)	21. (b)	22. (c)	23. (c)	24. (d)
25. (b)	26. (c)	27. (b)	28. (d)	29. (a)	30. (c)
31. (a)	32. (a)	33. (c)	34. (d)	35. (c)	36. (a)
37. (d)	38. (d)	39. (c)	40. (b)	41. (c)	42. (b)
43. (a)	44. (c)	45. (a)	46. (b)	47. (c)	48. (b)
49. (d)	50. (b)	51. (b)	52. (c)	53. (a)	54. (b)
55. (c)	56. (d)	57. (d)	58. (d)	59. (b)	60. (c)
61. (a)	62. (a)	63. (b)	64. (d)	65. (b)	66. (b)
67. (a)	68. (b)	69. (b)	70. (c)	71. (d)	72. (d)
73. (c)	74. (b)	75. (a)	76. (b)	77. (c)	78. (b)
79. (c)	80. (b)	81. (d)	82. (a)	83. (d)	84. (c)
85. (b)	86. (c)	87. (b)	88. (a)	89. (b)	90. (c)
91. (d)	92. (a)	93. (a)	94. (c)	95. (d)	96. (b)
97. (c)	98. (b)	99. (c)	100. (d)	101. (a)	102. (c)
103. (d)	104. (b)	105. (c)	106. (b)	107. (c)	108. (c)
109. (d)	110. (b)	111. (b)	112. (a)	113. (b)	114. (c)
115. (a)	116. (b)	117. (c)	118. (b)	119. (a)	120. (c)
121. (a)	122. (d)	123. (b)	124. (a)	125. (d)	126. (b)
127. (d)	128. (b)	129. (b)	130. (d)	131. (a)	132. (c)
133. (b)	134. (c)	135. (c)	136. (d)	137. (a)	138. (b)
139. (d)	140. (a)	141. (c)	142. (a)	143. (b)	144. (d)
145. (d)	146. (b)	147. (c)	148. (a)	149. (d)	150. (c)
151. (a)	152. (c)	153. (a)	154. (a)	155. (b)	156. (d)
157. (c)	158. (a)	159. (d)	160. (a)	161. (b)	162. (d)
163. (b)	164. (c)	165. (a)	166. (a)	167. (b)	168. (d)
169. (c)	170. (b)	171. (d)	172. (a)	173. (b)	174. (c)
175. (b)	176. (d)	177. (c)	178. (d)	179. (c)	180. (b)
181. (c)	182. (a)	183. (b)	184. (c)	185. (c)	186. (b)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

187. (a)	188. (b)	189. (b)	190. (c)	191. (c)	192. (b)
193. (c)	194. (b)	195. (c)	196. (b)	197. (a)	198. (d)
199. (a)	200. (b)	201. (d)	202. (c)	203. (c)	204. (a)
205. (a)	206. (a)	207. (a)	208. (a)	209. (d)	210. (d)
211. (b)	212. (a)	213. (c)	214. (b)	215. (b)	216. (c)
217. (c)	218. (c)	219. (b)	220. (b)	221. (d)	222. (c)
223. (c)	224. (b)	225. (c)	226. (a)	227. (b)	228. (b)
229. (c)	230. (d)	231. (a)	232. (c)	233. (c)	234. (b)
235. (a)	236. (b)	237. (c)	238. (a)	239. (a)	240. (a)
241. (a)	242. (b)	243. (d)	244. (c)	245. (c)	246. (c)
247. (b)	248. (a)	249. (b)	250. (d)	251. (a)	252. (a)
253. (b)	254. (d)	255. (b)	256. (a)	257. (c)	258. (d)
259. (c)	260. (a)	261. (a)	262. (b)	263. (c)	

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix I

Main Relations of Fluid Mechanics in Vector Notation

Any vector may be decomposed into three components along mutually perpendicular axes, say x, y, z axes. The component is a scalar quantity. The vector may be expressed in terms of its scalar components by use of unit vectors i, i, k parallel to the x, y, z axes respectively. A unit vector is a vector having unit magnitude. Thus if a is a vector having scalar components a_x, a_y, a_z along the three axes then

$$a = ia_x + ja_y + ka_z$$

The unit vectors combine as follows:

$$i \,.\, i = j \,.\, j = k \,.\, k = 1; \; i \,.\, j = j \,.\, k = k \,.\, i = 0$$

$$i \times j = k \,;\; j \times k = i \,;\; k \times i = j = -\, i \times k \text{ etc.}$$

The scalar (or dot) product of two vectors a and b is

$$\begin{aligned} a \,.\, b &= (ia_x + ja_y + ka_z) \,.\, (ib_x + jb_y + kb_z) \\ &= a_x b_x + a_y b_y + a_z b_z \end{aligned}$$

The vector (or cross) product of two vectors a and b is

$$\begin{aligned} a \times b &= (ia_x + ja_y + ka_z) \times (ib_x + jb_y + kb_z) \\ &= i(a_y b_z - a_z b_y) + j(a_z b_x - a_x b_z) + k(a_x b_y - a_y b_x) \end{aligned}$$

The above expression can be conveniently written in determinant form as

$$a \times b = \begin{vmatrix} i & j & k \\ a_x & a_y & a_z \\ b_x & b_y & b_z \end{vmatrix}$$

The various quantities as well as relations commonly used in Fluid Mechanics may be expressed in vector form as indicated below.

Velocity:

If velocity vector V has components u, v and w along three axes x, y, and z in the Cartesian system then

$$V = iu + jv + kw$$

where $u = u\,(x, y, z, t)$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

$$v = v(x, y, z, t)$$
$$w = w(x, y, z, t)$$

The magnitude of velocity V is given by

$$|V| = \sqrt{u^2 + v^2 + w^2}$$

In vector analysis, vector differential operator ∇ '*del*' is defined as follows:

$$\nabla = i\frac{\partial}{\partial x} + j\frac{\partial}{\partial y} + k\frac{\partial}{\partial z}$$

Thus for example for any function ϕ

$$\nabla\phi = i\frac{\partial\phi}{\partial x} + j\frac{\partial\phi}{\partial y} + k\frac{\partial\phi}{\partial z}$$

Further the *gradient* of any function is defined as

$$\text{grad } \phi = i\frac{\partial\phi}{\partial x} + j\frac{\partial\phi}{\partial y} + k\frac{\partial\phi}{\partial z}$$

Thus by comparison, we have

$$\nabla\phi = \text{grad } \phi$$

The velocity vector V may by expressed in terms of the velocity potential as

$$V = -\text{grad } \phi = -\nabla\phi$$

Acceleration:

The acceleration vector a may be represented as

$$a = \frac{DV}{Dt} = \frac{\partial V}{\partial t} + V\nabla V$$

where $$a = ia_x + ja_y + ka_z$$

The three components of the acceleration vector are

$$a_x = \frac{Du}{Dt} = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}$$

$$a_y = \frac{Dv}{Dt} = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}$$

$$a_z = \frac{Dw}{Dt} = \frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}$$

Continuity Equation:

The continuity equation in the most general form may be expressed in vector notation as

$$\frac{D\rho}{Dt} + \rho\nabla.V = 0$$

which is equivalent to its following form in Cartesian system

$$\frac{\partial\rho}{\partial t} + u\frac{\partial\rho}{\partial x} + v\frac{\partial\rho}{\partial y} + w\frac{\partial\rho}{\partial z} + \rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = 0$$

or $$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The scalar (or dot) product of any quantity with ∇ is interpreted as *divergence* or in short form '*div*'. Thus in the above expression

$$\nabla.V = div\ V$$

Further it can be readily shown that

$$\nabla.V = \text{div } V = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$$

Equations of Motion:

The equations of motion can be written in vector form as

$$\frac{DV}{Dt} = B - \frac{1}{\rho}\nabla p\ ; \quad \text{or} \quad \frac{DV}{Dt} = B - \frac{1}{\rho}\ \text{grad}\ p$$

where B is body force having components X, Y and Z so that

$$B = iX + jY + kZ$$

Further body forces are assumed to be derived from a potential Ω such that

$$B = -\text{ grad } \Omega$$

or

$$X = -\frac{\partial \Omega}{\partial x}, Y = -\frac{\partial \Omega}{\partial y}, Z = -\frac{\partial \Omega}{\partial z}$$

Circulation and vorticity:

In expressing these terms in vector form the operator *curl* is introduced, which is interpreted as the vector (or cross) product of ∇ with any quantity. Thus

$$\text{curl } V = \nabla \times V$$

Circulation Γ may be expressed in vector form as

$$\Gamma = \iint_A \text{curl } V \cdot dA$$

The vorticity ζ is defined as curl of velocity vector V, i.e.,

$$\zeta = \text{curl } V = \nabla \times V$$

Further it can be shown that

$$\nabla \times V = i\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) + j\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) + k\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

The quantities in paranthesis are vorticity components which are twice the value of rotation components

$$\omega_x, \omega_y, \omega_z$$

$$\therefore \quad \nabla \times V = i\,2\omega_x + j\,2\omega_y + k\,2\omega_z$$

The rotation of a fluid at a point may thus be defined as

$$w = \frac{1}{2}\ \text{curl } V = \frac{1}{2}\ \nabla \times V$$

Irrotational flow:

The condition for the flow to be irrotational or the existence of velocity potential may thus be expressed in the vector form as

$$\text{curl } V = \text{curl } (-\text{ grad } \phi) = -\nabla \times \nabla \phi = 0$$

which is equivalent to the condition in Cartesian system as

$$\omega_x = \omega_y = \omega_z = 0$$

Further any function ϕ that satisfies the Laplace equation is a possible irrotational fluid-flow case. The Laplace equation can be obtained in vector form as

$$\nabla . V = -\nabla . \nabla\phi = -\nabla^2 \phi = 0$$

and it is expressed as

$$\nabla^2\phi = 0$$

which is equivalent to its form in Cartesian system as

$$\frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} + \frac{\partial^2\phi}{\partial z^2} = 0$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix II

Comparative Study of Flow of Incompreessible and Compressible fluids

Particulars (1)	*Incompressible Flow* (2)	*Compressible Flow* (3)
1. Continuity Equation		
(a) Integral Form-One Dimensional Steady Flow	$AV = \text{Constant}$	$\rho AV = \text{Constant}$
(b) Differential Form		
(i) One Dimensional Steady Flow	$\frac{dA}{A} + \frac{dV}{V} = 0$	$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dV}{V} = 0$
Unsteady Flow	$\frac{\partial A}{\partial t} + \frac{\partial (AV)}{\partial s} = 0$	$\frac{\partial (\rho A)}{\partial t} + \frac{\partial (\rho AV)}{\partial s} = 0$
(ii) Two Dimensional Steady Flow	$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$	$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0$
Unsteady Flow	$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$	$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0$
(iii) Three Dimensional Steady Flow	$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$	$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0$

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Contd.

Particulars (1)	*Incompressible Flow* (2)	*Compressible Flow* (3)
Unsteady Flow	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$	$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$
2. Momentum Equation	$\sum F_x=\rho AV[(V_x)_2-(V_x)_1]$	$\sum F_x=[(\rho AVV_x)_2-(\rho AVV_x)_1]$
3. Energy Equation		
(a) In terms of energy per unit weight of fluid	$\frac{p}{w}+z+\frac{V^2}{2g}=$ constant	$E_n+\frac{V^2}{2g}+z=$ constant; in which $E_n=\frac{p}{w}+I=$ Enthalpy per unit weight of fluid $I=$ Internal heat energy of fluid to be expressed in terms of equivalent mechanical energy per unit weight of fluid.
(*b*) In terms of energy per unit mass of fluid	$\frac{p}{\rho}+gz+\frac{V^2}{2}=$ constant	$E_n+\frac{V^2}{2}+gz=$ constant; in which $E_n=\frac{p}{\rho}+I=$ Enthalpy per unit mass of fluid or specific enthalpy $I=$ Internal heat energy of fluid to be expressed in terms of equivalent mechanical energy per unit mass of fluid. The necessary conversion factors for these may be seen in Appendix III.
4. Dimensionless Stagnation pressure or Pressure coefficient	$\frac{p_2-p_1}{\frac{\rho_1 V_1^2}{2}}=1$	$\frac{p_2-p_1}{\frac{\rho_1 V_1^2}{2}}=1+\frac{1}{4}(M_a)_1^2+\frac{2-k}{24}(M_a)_1^4+.....$
5. Flow through a pipe of Varying Cross-section	$\frac{dA}{A}<0;\frac{dV}{V}>0$ (*Convergent Nozzle*) $\frac{dA}{A}>0;\frac{dV}{V}<0$ (*Divergent diffuser*)	(*a*) *For Subsonic Flow(i.e., M_a<1)* $\frac{dA}{A}<0;\frac{dV}{V}>0$ (*Convergent nozzle)* $\frac{dA}{A}>0;\frac{dV}{V}<0$ (*Divergent Diffuser)*

Contd.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Contd.

Particulars (1)	*Incompressible Flow* (2)	*Compressible Flow* (3)
	$\frac{dA}{A}=0;\frac{dV}{V}=0$ (*Pipe of Uniform Cross-section*)	(*b*) *For Supersonic Flow (i.e.,* $M_a>1$*)* $\frac{dA}{A}>0;\frac{dV}{V}>0$ (*Divergent Nozzle*) $\frac{dA}{A}<0;\frac{dV}{V}<0$ (*Convergent Diffuser*) (*c*) *For Sonic Flow (i.e.,* $M_a=1$*)* $dA=0$, *i.e.,* the sonic flow will occur at a section of minimum cross-sectional area.
6. Other Phenomena of Similar Nature	Change of supercritical flow to subcritical flow in open channels is through the formation of a hydraulic jump. It is accompanied by an increase in depth and decrease in the velocity of flow.	Change of supersonic flow to subsonic flow is through the formation of shock waves. It is accompanied by an increase in the pressure and density of fluid and a decrease in the velocity.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix III

Some Important Conversion Factors

FOR ENGLISH (FPS), METRIC (MKS) AND SI UNITS

1. LENGTH

1 cm	=	0.394 in;	1 in	=	2.54 cm
1 m	=	3.28 ft;	1 ft	=	0.305 m
1 km	=	0.621 mile	1 mile	=	1.609 km

2. AREA

1 cm^2	=	0.155 in^2;	1 in^2	=	6.452 cm^2
1 m^2	=	10.764 ft^2;	1 ft^2	=	0.093 m^2
1 hectare	=	100 are = 10^4 m^2 = 2.471 acres			
1 acre	=	43 560 ft^2 = 4 840 sq. yd. = 0.4047 hectare			

3. VOLUME

1 cm^3	=	0.061 in^3;	1 in^3	=	16.387 cm^3
1 m^3	=	35.315 ft^3;	1 ft^3	=	0.0283 m^3
1 litre	=	0.0353 ft^3;	1 ft^3	=	28.3 litres
1 litre	=	0.220 imp. gal;	1 imp. gal	=	4.546 litres
1 litre	=	0.264 US gal;	1 US gal	=	3.785 litres

4. MASS

1 msl	=	0.672 slug;	1 slug	=	1.488 msl
1 kg(m)	=	2.205 lb(m)	1 lb(m)	=	0.453 6 kg(m)
1 kg(m)	=	0.068 5 slug	1 slug	=	14.59 kg(m)
1 slug	=	32.2 lb(m);	1 lb(m)	=	0.031 08 slug
1 msl	=	9.81 kg(m);	1 kg(m)	=	0.1020 msl

5. FORCE

1 kg(f)	=	2.205 lb(f);	1 lb(f)	=	0.4536 kg(f)

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

1 N	=	0.1020 kg(f);	1 kg(f)	=	9.81 N
1 N	=	10^5 dynes;	1 lb(f)	=	32.2 pdl
1 N	=	0.2248 lb(f);	1 lb(f)	=	4.448 N
1 tonne	=	1000 kg = 0.984 ton			

6. MASS DENSITY

1 msl/m^3	=	0.019 slug/ft^3;	1 slug/ft^3	=	52.56 msl/m^3
1 gm/cc	=	1 kg(m)/litre = 1000 kg(m)/m^3			
1 gm/cc	=	102 msl/m^3 = 1.94 slug/ft^3			
1 gm/cc	=	62.4 lb(m)/ft^3;	1 lb(m)/ft^3	=	0.01602 gm/cc
1 slug/ft^3	=	0.515 4 gm/cc			

7. WEIGHT DENSITY

1 kg(f)/m^3	=	0.0624 lb(f)/ft^3;	1 lb(f)/ft^3	=	16.03 kg(f)/m^3
1 kg(f)/m^3	=	9.81 N/m^3;	1 N/m^3	=	0.1020 kg(f)/m^3

8. PRESSURE, STRESS

1 kg(f)/cm^2	=	14.22 lb(f)/in^2;	1 lb(f)/in^2	=	0.0703 kg(f)/cm^2
1 kg(f)/m^2	=	0.205 lb(f)/ft^2;	1 lb(f)/ft^2	=	4.882 kg(f)/m^2
1 N/m^2	=	0.1020 kg(f)/m^2;	1 kg(f)/m^2	=	9.81 N/m^2
1 N/m^2	=	0.0209 lb(f)/ft^2;	1 lb(f)/ft^2	=	47.88 N/m^2

9. VISCOSITY

1 kg(f)-s/m^2	=	0.205 lb(f)-s/ft^2 = 9.81 N.s/m^2			
1 lb(f)-s/ft^2	=	4.882 kg(f)-s/m^2 = 47.88 N.s/m^2			
1 poise	=	0.002 09 lb(f)-s/ft^2			
1 lb-s/ft^2	=	478.8 poise	1 poise	=	0.1 N.s/m^2
1 N.s/m^2	=	0.1020 kg(f)-s/m^2			

10. KINEMATIC VISCOSITY

1 m^2/s	=	10.764 ft^2/s;	1 ft^2/s	=	0.092 9 m^2/s
1 stoke	=	0.001 0764 ft^2/s;	1 ft^2/s	=	929 stokes

11. SURFACE TENSION

1 kg(f)/m	=	0.672 lb(f)/ft;	1 lb(f)/ft	=	1.488 kg(f)/m
1 N/m	=	0.06858 lb(f)/ft;	1 lb(f)/ft	=	14.583 N/m
1 N/m	=	0.1020 kg(f)/m;	1 kg(f)/m	=	9.81 N/m

12. WORK, ENERGY, QUANTITY OF HEAT

1 kg(f)-m	=	7.233 ft-lb(f);	1 ft-lb(f)	=	0.138 2 kg(f)-m
1 J	=	0.737 6 ft-lb(f);	1 ft-lb(f)	=	1.356 J
1 J	=	0.1020 kg(f)-m;	1 kg(f)-m	=	9.81 J
1 kWh	=	3.60×10^6 J = 0.367×10^6 kg(f)-m			
1 Btu	=	778 ft-lb(f) = 1055 J = 0.293 Wh			
1 CHU	=	1400 ft-lb(f) = 1897 J = 0.527 Wh			
1 kcal	=	426.7 kg(f)-m;	1 kg(f)-m	=	2.344 cal
1 J	=	N.m = 10^7 ergs			

13. POWER

1 hp	=	550 ft-lb(f)/sec = 746 W = 76.04 kg(f)-m/sec
1 metric hp	=	75 kg(f)-m/sec = 0.986 hp = 735.5 W

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

1 W	=	1 J/s = 1 N.m/s = 10^7 ergs/sec			
1 kW	=	1.341 hp = 737.6 ft-lb(f)/sec			
1 kW	=	1897 CHU/hr = 3417 Btu/hr			
1 metric hp	=	632.2 kcal/hr;	1 kW	=	860 kcal/hr

14. TEMPERATURE

T°C (abs)	=	TK = t°C + 273.15
T°F (abs)	=	T°R* = t°F + 460
1 K	=	(5/9)° R*

15. GAS CONSTANT

$$1 \text{ ft-lb(f)/lb(m) deg C abs} = \left(\frac{5}{9}\right) \text{ ft-lb(f)/lb (m) deg F abs}$$

1 ft-lb(f)/lb(m) deg C abs	=	2.99 joule/kg (m) deg C abs
1 kg(f)-m/kg(m) deg C abs	=	9.81 joule/kg(m) deg C abs
1 joule/kg(m) deg C abs	=	1 m^2/sec^2 deg C abs
	=	0.1020 m/deg C abs

16. ANGLE

1 rad	=	57.296°	1°	=	0.017 453 rad

* deg. Rankine

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix IV

Source, Sink and Doublet

In the study of ideal-fluid flow certain mathematical terms are considered which are defined below.

Three Dimensional Source

A three dimensional source is defined as a point from which fluid is imagined to be issued radially at a uniform rate in all directions. It is characterised by its 'strength' m which is the rate of flow of fluid passing through any surface enclosing the source. The radial velocity V_r at any point at a distance r from the source is thus equal to the strength m divided by the surface area of the sphere of radius r and centre at the source, or

$$V_r = \frac{m}{4\pi r^2}$$

Further if ϕ is the velocity potential due to the source, then since $V_r = -(\partial\phi / \partial r)$ and $V_\theta = 0$

$$-\frac{\partial\phi}{\partial r} = \frac{m}{4\pi r^2}$$

which on integration gives

$$\phi = \frac{m}{4\pi r}$$

Again if ψ is the stream function then in spherical polar coordinates

$$V_r = -\frac{1}{r^2 \sin\theta}\frac{\partial\psi}{\partial\theta} = \frac{m}{4\pi r^2}$$

which on integration gives

$$\psi = \frac{m}{4\pi}\cos\theta$$

The flow pattern resulting from a source may be represented by equipotential lines and streamlines which are given by ϕ = constant and ψ = constant respectively.

Three Dimensional Sink

A three dimensional sink is defined as a point towards which fluid is imagined to flow radially at a

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

uniform rate from all directions. Thus a sink may be considered as a negative source and hence the strength of sink is $-m$. The radial velocity V_r at a distance r from the sink, velocity potential ϕ and stream function ψ for the sink are therefore given by the following expressions.

$$V_r = -\frac{m}{4\pi r^2}; \phi = -\frac{m}{4\pi r}; \psi = -\frac{m}{4\pi}\cos\theta$$

Two Dimensional Source

A two dimensional source is defined as a line from which fluid is imagined to be issued at right angles to the line from every point on it at a uniform rate in all directions. Thus a two dimensional source is also known as *line source* and it is to be distinguished from a three dimensional source which is a *point source.* The 'strength' m of a two dimensional source is the volume of fluid issued per unit time per unit length of the source. Thus the radial velocity V_r at any point at a distance r from the source at right angles to it is equal to the strength m divided by the surface area of a cylinder of radius r and unit length with its axis coinciding with the source, or

$$V_r = \frac{m}{2\pi r}$$

Further if ϕ is the velocity potential due to the source, then since $V_r = -(\partial\phi/\partial r)$ and $V_\theta = 0$

$$-\frac{\partial\phi}{\partial r} = \frac{m}{2\pi r}$$

which on integration gives

$$\phi = -\frac{m}{2\pi}\log r$$

Again if ψ is the stream function then in cylindrical polar coordinates

$$V_r = -\frac{1}{r}\frac{\partial\psi}{\partial\theta} = \frac{m}{2\pi r}$$

which on integration gives

$$\psi = -\frac{m}{2\pi}\theta$$

In this case also the flow pattern may be represented by the lines of constant ϕ (equipotential lines) and constant ψ (streamlines).

Two Dimensional Sink

A two dimensional sink is defined as a line towards which fluid is imagined to flow at right angles to the line at every point on it at a uniform rate in all directions. Thus a two dimensional sink may be considered as a negative two dimensional source, and the expressions for V_r, ϕ and ψ for a two dimensional sink may be obtained by introducing $-m$ in place of m in the corresponding expressions derived for a two dimensional source.

It is evident from the definition of source and sink that a source implies the creation of fluid and a sink implies the annihilation of fluid. Further it may be stated that though source and sink are entirely imaginary, these are quite useful in obtaining flow patterns in several cases of ideal-fluid flow.

Doublet

A doublet is defined as a combination of a source of strength m and a sink of strength $-m$, placed a small distance δs apart which are allowed to approach each other (i.e., $\delta s \to 0$) in such a manner that

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

the product of their strength and the distance between them (i.e., $m\delta s$) remains a finite constant equal to μ, called the strength of the doublet. The line joining source and sink is known as the axis of the doublet and is considered positive in the direction from sink to source. A doublet may also be classified as three dimensional doublet and two dimensional doublet depending upon the source and sink being three dimensional and two dimensional respectively. Further by using the principle of superposition the expressions for the velocity potential and the stream function for three dimensional and two dimensional doublets may be obtained as noted below.

Three dimensional doublet

$$\phi = \frac{\mu}{r^2}\cos\theta; \quad \psi = -\frac{\mu}{r}\sin^2\theta$$

Two dimensional doublet

$$\phi = \frac{\mu}{r}\cos\theta; \quad \psi = -\frac{\mu}{r}\sin\theta$$

The flow pattern resulting from a doublet may also be represented by lines of constant ϕ and constant ψ.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix V

Cavitation

When a liquid flows into a region where its pressure is reduced to vapour pressure, it starts vaporizing (or boiling), and vapour pockets or bubbles are formed in the liquid. These vapour bubbles are carried along with the flowing liquid until a region of higher pressure is reached, where they suddenly collapse as the vapour condenses to liquid again. Moreover, these bubbles may also collapse even at the point where they are formed, because of the momentary local increase in the pressure at this point. When a vapour bubble collapses a cavity is formed and the surrounding liquid rushed in to fill it. This process of formation of vapour bubbles and their collapsing is called *cavitation.* In addition to this, liquids generally contain some dissolved air. This dissolved air is released as the pressure is reduced and thus air pockets or bubbles are also formed in the liquid. The air pockets are often termed as air lock due to which *air cavitation* occurs.

When the vapour bubble collapse the liquid rushing from all directions collides at the centre of the cavity, thus giving rise to very high local pressure (up to 7×10^8 N/m^2 or 7030 kg(f)/cm^2). Any solid surface in the vicinity of the cavities is also subjected to these intense pressures, because, even if the cavities are not actually at the solid surface, the pressures are propagated from the cavities by pressure waves similar to those encountered in water hammer. The formation and collapse of the large number of bubbles on a boundary surface subject that surface to intense local stressing. The boundary surface is thus alternately stressed and relieved of the stresses, thereby ultimately damaged by fatigue, resulting in serious erosion of the surface which is known as *pitting*. The phenomenon of cavitation is always accompanied by considerable noise and vibrations. Moreover, when the cavitation occurs the flow is considerably disturbed, the flow passage is damaged and the efficiency is lowered. The air cavitation is less damaging then vapour cavitation to surfaces but it also reduces the efficiency of the system considerably.

In any system of flowing liquid, the *cavitation parameter* σ is useful in characterizing the susceptibility of the system to cavitate. It is defined by

$$\sigma = \frac{p - p_v}{(\rho V^2/2)}$$

in which p is the absolute pressure at the point under consideration in the flowing system, p_v is the vapour pressure of the liquid, ρ is mass density of the liquid and V is the reference velocity of flow of liquid. The cavitation parameter is a form of pressure coefficient and it is a dimensionless number. In

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

two geometrically similar systems if the valve of σ is same, they would be equally likely to cavitate or they would have the same degree of cavitation. Further when σ = 0, the pressure is reduced to vapour pressure of the liquid and the vaporization (or boiling) of liquid would take place.

Tests have shown that chemically pure liquids can sustain high tensile stresses (of the order of several hundreds of N/m^2 or $kg(f)/cm^2$), which is in contradiction of the concept of the formation of the cavities when pressure is reduced to vapour pressure. On the other hand in the case of commercial or practical liquids when vapour pressure is reached there is spontaneous vaporization (or boiling) and hence the cavitation occurs. As such it has been generally accepted that the vapour bubbles form and grow around some nuclei which must be present in the liquid. The nature of the nuclei is not thoroughly understood, but they may be microscopic dust particles or other contaminants, which are widely dispersed through practical liquids.

The complete phenomenon of cavitation viz., the formation of the vapour bubbles on nuclei, their growth, their movement into a region of higher pressure and finally their collapse, may take place in a few thousandths of a second. Further the surface tension of the vapour bubbles appears to be an important property accounting for the high pressure pulses resulting from the collapse of a vapour bubble.

The best protection against cavitation (if practicable) is to adopt such a hydraulic design of the system which would avoid the development of low pressure. For this the absolute pressure head in terms of water should not be allowed to fall below about 2.13 m (i.e., about 2×10^4 N/m^2 or 0.213 $kg(f)/cm^2$). However, it may not be always possible to develop completely cavitation free regions in various flow systems such as turbines, centrifugal pumps etc. In such cases use of special cavitation resistant materials or coatings may be made. Some of the cavitation resistant materials are aluminium-bronze (83% copper, 10–12% aluminium, 4-6% iron), stainless steel (with different percentages of chromium, nickel and iron) etc.

As stated in previous chapters, turbines and centrifugal pumps are subject to cavitation. In addition to these, ship propellers and many hydraulic structures are also subject to cavitation. The curved vanes of turbine runners and pump impellers are particularly susceptible to cavitation on their convex sides and may have localized areas where cavitation causes pitting or failure. The location of turbines and pumps considerably affect the cavitation in these hydraulic machines. A special cavitation factor called *Thoma's cavitation factor* (or number or index) represented by σ is considered for turbines as well as centrifugal pumps which has been indicated in the respective chapters. In addition to this another parameter called *suction specific speed* is also used to represent the cavitation characteristics of turbines and centrifugal pumps which has also been indicated in the respective chapters.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix VI

Flow in Curved Channels

In the alignment of the channels, curves or bends are often required to be provided. The flow around a curved path is quite complex because the streamlines of the flows are not only curvilinear but also interwoven resulting in spiral currents and cross waves. Moreover, the centrifugal force acting on the flow around a bend produces a unique feature known as *superelevation*, that is, a rise in the water surface at the outer bank with an accompanying lowering at the inner bank. Also the velocity distribution in the channel in the bends is very irregular and the coefficients α and β are usually far greater than unity. Furthermore the characteristics of flow in curved channels are also affected by the nature of the flow viz., subcritical and supercritical. In general subcritical flow shows smooth water surface and slight superelevation, whereas supercritical flow exhibits cross-wave disturbance patterns on the surface and thus exaggerates the superelevation.

By assuming a theoretical free-vortex motion for the flow along curved channel, it is possible to obtain the expressions for the velocity distribution and the water surface profile at the section of maximum surface depression. However, this assumption holds good only for the subcritical flow.

For the free-vortex motion the following expression can be written:

$$v = \frac{C}{r} \qquad \text{...(i)}$$

where v is the filamental velocity in the curve at a radial distance r from the centre of curvature, and C is the circulation constant in a free vortex motion.

If E be the specific energy at any section and y the depth of flow at a distance r from the centre of curvature, then

$$y = E - \frac{v^2}{2g} \qquad ..(ii)$$

The mean velocity of flow is

$$V = \frac{\int_{r_i}^{r_0} (C/r)dr}{(r_0 - r_i)} = \frac{C}{r_0 - r_i} \log_e \left(\frac{r_0}{r_i} \right)$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

where r_0 and r_i are the radii of curvature of the outer and inner banks respectively. Also the mean depth of flow is

$$y_m = \frac{\int_{r_i}^{r_0} y\,dr}{(r_0 - r_i)} = \frac{\int_{r_i}^{r_0} \left(E - \frac{C^2}{2gr^2}\right) dr}{r_0 - r_i}$$

or
$$y_m = E - \frac{C^2}{2gr_0 r_i} \quad \text{...(iv)}$$

Now the discharge is

$$Q = Vy_m(r_0 - r_i) = C\left(E - \frac{C^2}{2gr_0 r_i}\right)\log_e\left(\frac{r_0}{r_i}\right) \quad \text{...(v)}$$

If Q, r_0, r_i and E are given, the constant C can be determined from Eq. (v). The velocity and dpeth at any radius r are then obtained by Eqs. (i) and (ii). The superelevation Δh of the water surface can thus be obtained as

$$\Delta y = (y_0 - y_i) = \left(\frac{v^2}{2g}\right)_i - \left(\frac{v^2}{2g}\right)_0 = \frac{C^2}{2gr_0^2 r_i^2}(r_0^2 - r_i^2) \quad \text{...(vi)}$$

As in the case of highway and railroad curves, the bed of curved channel may also be provided with a certain cross slope, so that it would produce the desired cross slope in the water surface, which in turn would provide the necessary centripetal force. The expression defining the transverse bed profile may however be obtained as indicated below.

If H_0 represents the total energy referred to the bed level at a section just upstream of the bend, y_0 and v_0 are depth and velocity of flow respectively at this section, y and v are respectively the depth and velocity of flow at any section of the bend as defined earlier, and z is the height of the bed at any section of the bend above the bed level of the channel at the upstream section, then assuming no loss of energy, one obtains

$$z + y + \frac{v^2}{2g} = H_0 = y_0 + \frac{v_0^2}{2g}$$

But
$$y = \frac{q}{v} = \frac{qr}{C}; \quad \text{and } C = v_0 r_0$$

where q is the discharge per unit width. Thus by substitution the following expression may be obtained which defines the transverse bed profile at the channel bend

$$z + \frac{qr}{v_0 r_0} + \frac{v_0^2 r_0^2}{2gr^2} = H_0 \quad \text{...(vii)}$$

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix VII

Control Valves for Pipes

Valves are required to be introduced in the pipelines for one or all the following purposes:

(i) To stop the flow of liquid altogether.

(ii) To regulate the rate of flow of the liquid.

(iii) To regulate the head or pressure in the pipelines.

According to the motion of the valve element, the control valves may be classified in the following three categories:

(a) Valve element moves transversely across the water-way.

(b) Valve element rotates about an axis transverse to the water-way.

(c) Valve element slides axially or longitudinally.

(a) With transverse motion

In this category of valves, the following two types of valves may be considered.

(*i*) *Full-way, or sluice or gate valve.* This is a very commonly used valve, in which the moving element has the form of a disc or circular gate, wedge-shaped in cross-section, having a nut which engage with the thread of the operating spindle as shown in the accompanying figure. In the fully-open position, the gate is withdrawn clear of the water-way and hence this valve is designated as full-way valve.

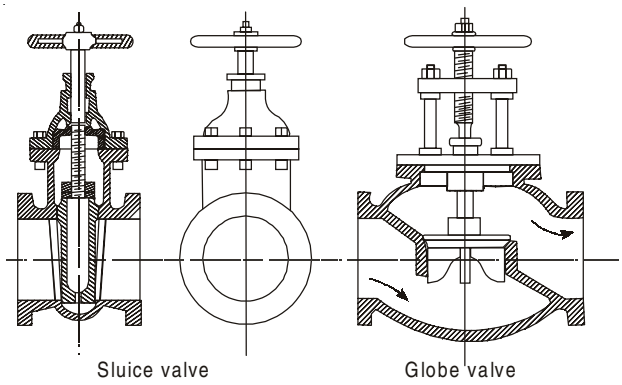


Figure App.VII–1

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

(*ii*) *Globe valve or stop valve.* As shown in the App.VII – 2 in this case the valve element is wing-guided mitre or disc, which in the closed position of the valve is forced on its circular seat by the screwed spindle. When this spindle releases the valve element, the pressure of the incoming liquid is sufficient to lift it, thus opening the valve.

(b) With rotary motion

The following two types of valves may be considered in this category.

(*i*) *Butterfly valve.* In this valve the cylindrical valve body forms virtually a continuation of the pipe itself, and it houses a moving element of elliptical shape. Rotation of the vale spindle brings the valve element either into the closed position, or into the fully open position in which it is parallel with the pipe axis as shown in App. VII–2

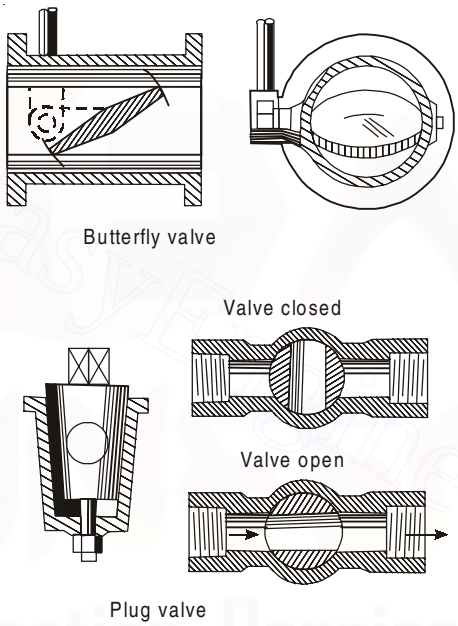


Figure App. VII–2

(*ii*) *Plug valve.* A conical plug has a port which in the open position of the valve is in line with the pipe. After being turned through 90°, the plug completely closes the water-way, as shown in the sectional view in the accompanying figure.

(c) With axial motion

In this category the following two types of valves may be included:

(*i*) *Sleeve valve.* As shown in the figure a cylindrical sleeve surrounding the body of the valve can be moved longitudinally so as to open or close the water-way. Such valves are usually fixed at the outlet end of the pipe, where it serves as an energy disperser.

(*ii*) *Needle valve.* It is another type of valve with an axially moving element. The operating piston is formed integrally with the needle which regulates the area of the water-way. High pressure water admitted to one side or the other of the piston causes the needle to take up the desired position. Figure App. VII–3 shows the valve (a) in the fully open position, (b) in the fully closed position, and (c) in the partly closed position.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

The needle valve resists cavitation damage quite successfully, because as shown in the figure in the case of needle valve the zone of intense eddying lies near the axis of the water-way, far away from the pipe walls. The sluice valve is however more susceptible to cavitation damage, particularly when used continuously in partly open position. However, in spite of this drawback the sluice valve is the most

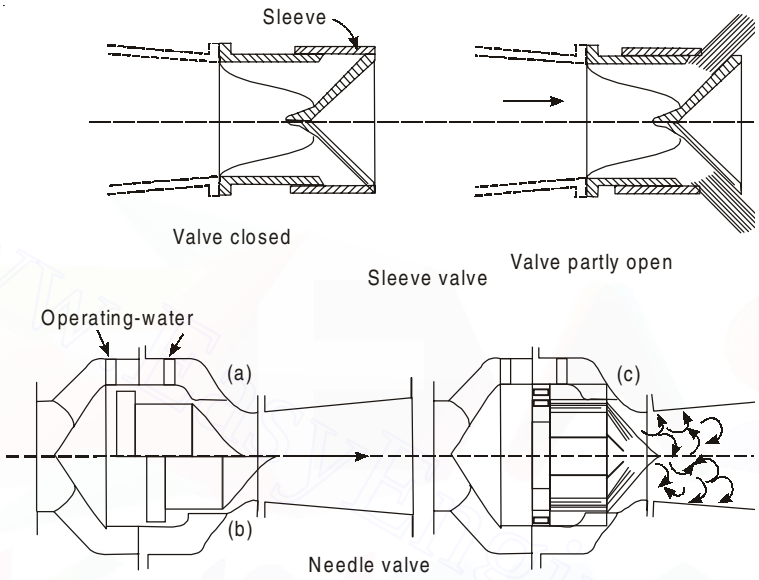


Figure App.VII –3

commonly used control valve, being suitable for the pipes of very small as well as very large diameters. The globe valve is most widely used in the domestic water tap. However, the globe valves are also used for the pipelines of very large diameters. The butterfly valves are generally used for regulating the flow of water in penstocks supplying water to the turbines installed in hydropower stations. Similarly the needle valves are also used for regulating the flow of liquid through large diameter pipes such as penstocks.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Appendix VIII

Hydraulic Transport of Solid Material in Pipes

The solid material may be transported in pipes through a liquid medium which in most of the cases is water. For many years the hydraulic transport of solid material in pipes has been successfully used in a few specialized fields such as dredging, land reclamation etc. However, this mode of transport of solid material is becoming more and more popular and it is being adopted for the transportation of coal, ash, natural sand, gravel, metal ores, crushed rock etc. Since the transport of solid material in pipes involves the flow of liquid-solid mixture, it is evident that the flow characteristics for this case would be different from those for the flow of pure water or any other liquid (free from solid material) in pipes. Moreover, the behaviour of the flow of a given liquid-solid mixture would be quite different in the horizontal and vertical pipes. As such in order to study the characteristics of flow of liquid-solid mixtures in pipes, experimental studies have been carried out by several investigators. The results of some of these investigations are briefly indicated below.

It has been observed that when liquid carries in suspension solid grains or particles not larger than about 0.15 mm, then the mixture can be regarded as a simple liquid having the same viscosity as that of liquid, but of a density increased by the presence of the suspended solids. The specific gravity S of such a liquid-solid mixture may be expressed as

$$S = S_l + C_v(S_s - S_l) \qquad ...(i)$$

where S_l is the specific gravity of the pure liquid, C_v is the concentration of solids by volume (i.e., unit volume of mixture contains a volume C_v of solids) and S_s is the true specific gravity in air of the dry solid particles. For water since $S_l = 1$ the above expression becomes

$$S = 1 + C_v (S_s - 1) \qquad ...(ii)$$

In a horizontal pipe carrying a mixture of water and sand grains in suspension, the concentration of solids is more near the bottom of the pipe than near the top of the pipe. Further it has been observed that when the sand grains are relatively small having a mean diameter of only 0.18 mm, then the velocity distribution curve in a vertical plane is symmetrical about the pipe axis and it is same as that for pure water. This implies that for the flow of mixture of water and sand gains of diameter less than about 0.18 mm in pipes, the various turbulent flow formulae for the flow of pure water in pipes may be applied, no matter whether the pipe is horizontal or vertical or inclined.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

When the size of the solid material transported along with the liquid increases, then in a horizontal pipe the solids tend more and more to settle to the bottom of the pipe. The velocity distribution curve becomes unsymmetrical, because the velocity near the bottom of the pipe is considerably reduced. Further in this case the frictional resistance increases with the increase in the concentration of the solids. Figure App. VIII–1 shows the results of a study conducted for the flow of liquid-solid mixture consisting of water and lumps of coal of about 12.5 mm diameter in a pipe of 75 mm diameter. It may be noted from the figure that as the concentration of solids is increased from 5% to 20% the head loss is considerably increased. Further when the mixture is flowing at a mean velocity of about 2.4 m/s (or 8 ft/s) or more the suspended lumps of coal do not affect the friction loss very much as it is only a little greater than that for clear water. As the mean velocity diminishes, the head loss also diminishes, but at one stage the head loss again rises as the mean velocity falls. Finally at low flow rates the head loss of each concentration of mixture tends to become independent of the mean velocity of flow. If corresponding to these constant values of the head loss, the values of friction factor f are computed then it is found that these values of f may be about 20 times higher than the equivalent values of f for clear water. The increase in the head loss at low velocities of flow may be due to the fact that at low velocities much of the solid material remains deposited at the bottom of the pipe, on account of which a portion of the pipe cross-section gets virtually blocked. As such the true area available for the flow is much less than the nominal area ($\pi d^2/4$) and the actual mean velocity is higher than the nominal velocity [$Q/(\pi d^2/4)$].

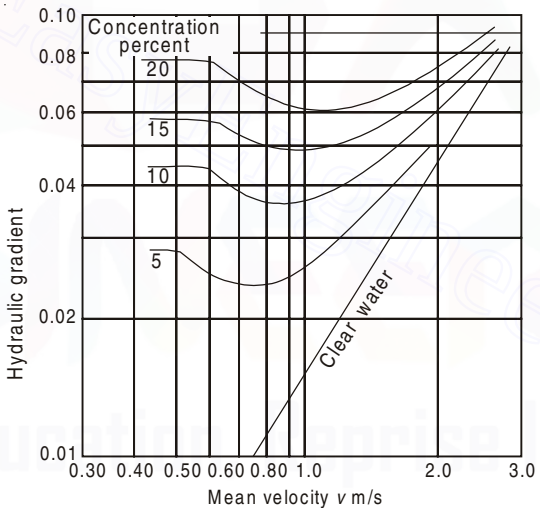


Figure App. VIII-1 *Relation between concentration of solids, mean velocity and hydraulic gradient*

When a liquid-solid mixture flows through a vertical pipe, the conditions are relatively simple as compared to those in a horizontal pipe. In a vertical pipe there is no question of differential velocities or differential settlement. However, in this case the solid particles tend to slip backwards through the water and therefore the mean rate at which the solids rise is less then the mean upward water velocity. This tendency of solids would result in higher friction loss.

For hydraulic transport of solids special centrifugal pumps are utilised whose impellers are capable of passing the largest pieces likely to occur in the mixture. Sometimes the solids and water in the desired proportions are fed to a hopper connected to the suction branch of the pump. The pump then delivers the mixture into the pipeline at a pressure sufficient to overcome the frictional resistance and any static head. When the solids are to be recovered, screening or other separating plant is used at the delivery end of the pipeline.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Bibliography

1. ADDISON, H, *A Treastise on Applied Hydraulics*, Chapman and Hall.
2. ALBERTSON, M.L., BARTON, J.R., AND SIMONS, D.B. , *Fluid Mechanics for Engineers*, Prentice-Hall.
3. BAKHMETEFF, B.A., *Hydraulics of Open Channels*, McGraw-Hill Co.
4. BARNA, P.S., *Fluid Mechanics for Engineers*, Butterworths.
5. BEITLER, S.R. AND LINDAHL, E.J, *Hydraulic Machinery*, Ronald Press Co.
6. BROWN, J.G., *Hydro-Electric Engineering Practice*, Blackie.
7. CHOW, V.T., *Open Channel Hydraulics*, McGraw-Hill Co.
8. DAUGHERTY, R.L., AND FRANZINI, J.B., *Fluid Mechanics with Engineering Applications*, McGraw-Hill Co.
9. DAUGHERTY, R.L., AND INGERSOLL, J.C., *Fluid Mechanics*, McGraw-Hill Co.
10. DAVIS, C.V., *Handbook of Applied Hydraulics*, McGraw-Hill Co.
11. DODGE, R.A., AND THOMPSON, M.J., *Fluid Mechanics*, McGraw-Hill Co.
12. DUNCAN, W.J., THOM, A.S, AND YOUNG A.D., *An Elementary Treatise on the Mechanics of Fluids*, Arnold Press.
13. ELEVATORSKI, E.A., *Hydraulic Energy Dissipators*, McGraw-Hill Co.
14. FRANCIS, J.R.D., *A Text Book of Fluid Mechanics*, Arnold Press.
15. GILES, R.V., *Theory and Problems of Fluid Mechanics and Hydraulics*, Schaum.
16. JAGDISH LAL, *Hydraulic Machines*, Metropolitan Book Co., Delhi.
17. JAGDISH LAL, *Hydraulic Machines*, Metropolitan Book Co., Delhi.
18. HUGHES, W.F., AND BRIGHTON, J.A., *Theory and Problems of Fluid Dynamics*, Schaum.
19. KAUFMANN, W., *Fluid Mechanics*, McGraw-Hill Co.
20. KOVALEV, N.N., *Hydroturbines-Design and Construction*, Israel Program for Scientific Translations, Jerusalem.
21. LANGHAAR, H.L, *Dimensional Analysis and Theory of Models*, John Wiley.
22. LEWITT, E.H., *Hydraulic and the Mechanics of Fluids*, Pitman.
23. LI, W.H., AND LAM, S.H., *Principles of Fluid Mechanics*, Addision-Wisley.
24. MASSEY, B.S., *Mechanics of Fluid*, Van Nostrard Co.
25. NECHLEBA, M., *Hydraulic Turbines, Their Design and Equipment*, Artia-Prague.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

26. PAO, R.H.F., *Fluid Mechanics*, John Wiley.
27. PARADISE, R.S., *Problems in Hydraulics*, Blackie.
28. POWELL, R.W., *An Elementary Text in Hydraulic and Fluid Mechanics*, Mac Millan.
29. ROUSE, H., *Elementary Mechanics of Fluids*, John Wiley.
30. ROUSE, H., *Fluid Mechanics for Hydraulic Engineers*, Dover.
31. ROUSE, H., *Engineering Hydraulics*, John Wiley.
32. RUSSELL, G.E., *Hydraulics*, Oxford
33. SCHLICHTING, H., *Boundary Layer Theory*, McGraw-Hill Co.
34. SHAMES, I.H., *Mechanics of Fluids*, McGraw-Hill Co.
35. S MITH, D.R.L., AND HOUGHTON, J., *Fluid Mechanics*, B.I.P. Bombay.
36. STEPANOFF. A.J., *Centrifugal and Axial Flow Pumps*, John Wiley.
37. STREETR, V.L., *Fluid Mechanics*, McGraw-Hill Co.
38. TROSKOLANSKI, A.T., *Hydrometry*, Pergaman Press.
39. VENNARD, J.K., *Elementary Fluid Mechanics*, John Wiley.
40. WEBER, N.B., *Fluid Mechanics for Civil Engineers*, Span.
41. WISLICENUS, G.F., *Fluid Mechanics of Turbomachinery*, Dover.
42. WOODWARD, S.M., AND POSEY, C.J., *Hydraulics of Steady Flow in Open Channels*, John Wiley.

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Author Index

Allen, C.M., 1342
Archimedes, 3, 155
Bazin, H., 459, 709
Bernoulli, D., 293, 294, 300
Blasius, H., 672
Bourdon, E., 63
Bresse, J.A.C., 799
Chezy, A., 500, 709
Cipolletti, C., 465
Colebrook, C.F., 677
Couette, M.F.A., 616
Darcy, H., 500
Euler, L., 229, 288, 857
Francis, J.B., 459
Froude, W., 364, 498, 856
Ganguillet, E., 709
Garde, R.J., 1331
Gibson, N.R., 1342
Hagen, G.H.L., 607
Herschel, C., 305
Hugoniot, H., 959
Joukowski, M.W.,918
Kaplan, V., 1044
Karman, T.V., 906
Kennedy, R., 1333
Kutta, M. W., 918
Kutter, W.R., 709
Lacey, G., 1334
Lamb, H., 906
de Lavel, C.G.P., 955
Mach, E., 857
Magnus, H.G., 918
Manning, R., 501, 710
Moody, L.F., 677, 1105
Newton, I., 12, 286, 857
Nikuradse, J., 663
Oseen, C.W., 901
Parshall, R.L., 738
Pascal, B., 47
Pelton, L.A., 1028
Poiseuille, J.L.M., 607
Prandtl, L., 3, 317, 567
Ranga Raju, K.G., 1331
Rankine, W., 959
Rayleigh, Lord, 843
Rehbock, T., 461
Reynolds, O., 494, 658, 856
Shields, A., 1327
Stanton, T.E., 674
Stokes, G.G., 11, 622
Strouhal, V., 908
Taylor, E.A., 1342
Thoma, D., 1106
Torricelli, E., 386
Venturi, G.B., 305
Weber, W.E., 858
Weisbach, J., 500
White, C.M., 1329

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Subject Index

A

Absolute pressure, 48
Absolue system, 3
Acceleration, 246
  convective, 247
  local, 247
  normal, 248
  tangential, 248
Acceleration
  of piston, 1138
  uniform horizontal, 193
  uniform rotational, 199
  uniform vertical, 196
Adhesion, 14
Adiabatic process, 939
Adverse slope, 790
Air foil
  drag of, 911
  lift on, 919
Air vessel, 1149
Allievi formula, 530
Alternate depth, 723
Angle of attack, 920
Angular deformation, 252
Archimedes, Principle of, 155
Aspect ratio, 920
Atmosphere, equilibrium of, 40
  adiabatic or isentropic, 42
  isothermal, 41
  polytropic, 44
  standard, 46
Atmospheric pressure, 48
Avogadro's law, 10
Average height of
  protrusions, 662

B

Base units, 4
Bazin's formula, 459, 709
Bend meter, 314
Bend, losses in, 407
Bernoulli's equation, 293, 294, 300
Blasius formula, 672
Body force, 286
Borda's mouthpice, 413
Boundary layer, 567, 661
  laminar, 570, 577
  transition, 570
  turbulent, 570, 580
Bourdon gage, 63
Branching pipes, 512
Bresse's method, 799
Broad crested weir, 470
Buckingham's pi method, 844
Bulk modulus of elasticity, 13
Buoyancy, 155
  centre of, 155
Buoyant force, 155
Butterfly valve, 1393
Bye pass, 511

C

Canal lock, 423
Capillarity, 16
Cauchy number, 857
Cavitation,
  in pumps, 1209
  in turbines, 1106
Celerity of elastic waves, 944
Centipoise, 11
Centistoke, 11
Centre of buoyancy, 155
Centre of pressure, 93
Centrifugal pumps, 1178
Characteristice curvres,
  of pumps, 1158, 1204
  of turbines, 1097
Chezy's equation, 500, 709
Cipolletti weir, 465
Circulation, 254

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Coefficient of
contraction, 386
discharge, 387
drag, 893
dynamic viscosity, 11
lift, 893
resistance, 388
velocity, 386
Cohesion, 14
Colebrook and White
equation, 677
Commercial pipe,
resistance of, 676
Compressible flows, 302, 939
Compressibility, 13
Conjugate depth, 802
Conservation of mass, 236
Continuity equation, 236, 942
Contraction, loss at sudden, 403
Control surface, 236
Control volume, 236
Convective acceleration, 247
Convergent divergent
mouthpiece, 411
Convergent divergent nozzle, 955
Convergent mouthpiece, 412
Correction factors,
energy, 301
momentum, 354
Couette flow, 616
Critical depth, 723
Critical flow
in channels, 728
in pipes, 495
Critical pressure ratio, 954
Critical Reynolds number, 495
Critical slope, 789
Critical velocity, 495, 723
Current meter, 739
Curved surfaces, pressure on, 103
Cylinder, drag on, 904
lift on, 915

E

d' Alembert's paradox, 353
Darcy-Weisbach formula, 500
Dash-pot mechanism, 630
Deformation drag, 897
Deformation of fluid
angular, 251
linear, 251
Delivery stroke, 1134
Density, 7
Depth, alternate, 723
conjugate, 802
critical, 723
initial, 802
sequent, 802
subcritical, 723
supercritical, 724
Derived units, 5
Dimensional analysis, 837
Dimensions of physical
quantities, 837
Dimensional homogeneity, 841
Dimensionally homogeneous
equation, 841
Dimensionless numbers, 856
Discharge coefficient
of mouthpiece, 409, 414, 416
of orifice, 387, 393
of weirs, 456
Discharge meaurement
in rivers, 742
Displacement thickness, 568
Distorted models, 861
Draft tube, 1037
Drag
deformation, 897
form, 897
friction, 894
induced, 925
pressure, 894
residual, 912
total, 894
Drag coefficient, 896
effect of compressibility
on, 913
effect of free surface on, 912
of three dimensional
bodies, 900
of two dimensional
bodies, 904
Drag on, airfoil, 911
cylinder, 904
disc, 898
plate, 910
sphere, 900
Dupuit's equation, 510
Dynamic similarity, 854
Dynamic viscosity, 11

E

Economical channel section, 712
Eddy kinematic viscosity, 659
Eddy viscosity, 659
Efficiency, Froude of
propeller, 364
hydraulic of pumps, 1195
hydraulic of turbines, 1026
manometric of pumps, 1192
mechanical of pumps, 1193
mechanical of turbines, 1026
overall of pumps, 1193
overall of turbines, 1026
volumetric of pumps, 1193
volumetric of turbines, 1026
Efficient cross-section of
channel, 712
Elastic modulus of fluids, 13
Elastic waves, 944, 947
Elbow meter, 314
Electrical analogy, 263
Electronic pressure cells, 1341
Energy correction factor, 301
Energy equation
for compressible flow, 302, 942

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

for incompressible flow, 300
Energy gradient, 506
Energy line, total, 503
Energy losses, 400, 502
Energy thickness, 569
English system units, 3
Enthalpy, 942
Entropy, 941
Entry loss, 405, 502
Equation of continuity, 236, 942
energy, 293, 302, 942
momentum, 353, 942
motion, 288
state, 9
Equilibrium of floating body, 158
neutral, 159
stable, 159
unstable, 159
Equipotential lines, 260
Equivalent pipe, 509
Equivalent sand grain roughness, 676
Eulerian method, 229
Euler number, 857
Euler's equation of motion, 288
Exit loss, 405, 502
Expansion, gradual, 406
sudden, 401

F

Fanno curve, 958
Flat plate, moving, 982
stationary, 979
Floating bodies, 155
Floats, 740
Flow net, 260
methods of drawing, 262
limitations of, 265
use of, 264
Flow through nozzle, 523
Fluid properties, 7
Fluids,
compressible, 40, 293, 939
ideal, 2, 12
Newtonian, 12
plastics, 12
real or practical, 2
thixotropic, 12
Force, due to elasticity, 287, 855
due to gravity, 287, 854
due to inertia, 287, 854
due to pressure gradient, 287, 855
due to surface tension, 287, 855
due to turbulence, 287
due to viscosity, 287, 567, 854
Forces on immersed surfaces, 93
Forces on fixed and moving plates, 979, 982
Form drag, 897
Francis formula, 459
Francis turbine, 1036
Free liquid jet, 317
Friction factor, 500
Friction factor diagram, 678
Froude efficiency of propeller, 364
Froude number, 856
Froude's experiments, 498

G

Gas constant, 9
Ganguillet-Kutter Formula, 709
Gage pressure, 48
Geometric similarity, 853
Globe valve, 1393
Governing of turbines, 1048
Gradually varied flow, 783
equation, 785
integration of equation, 797
Graphical method of flow net, 262
Gravity force, 287, 855
Guide vanes, 1036

H

Hagen-Poiseuille formula, 607
Hardy-Cross method, 532
Head, manometric, 1186
piezometric, 294
potential, 294
pressure, 39, 294
static, 1186
total, 294
velocity, 294
Head loss, due to friction, 499
Head loss, due to other causes, 400, 502
Homologous, 854
Hook gage, 1342
Hot-film anemometer, 963
Hot-wire anemometer, 962
Hydraulic accumulator, 1247
Hydraulic couplings, 1258
Hydraulic crane, 1252
Hydraulic depth, 704
Hydraulic efficiency
of pumps, 1195
of turbines, 1026
Hydraulic grade line, 503
Hydraulic gradient, 506
Hydraulic intensifier, 1249
Hydraulic jump, 801
Hydraulic lift, 1254
Hydraulic mean depth, 499, 704
Hydraulic radius, 499, 704
Hydraulic press, 1251
Hydraulic ram, 1255
Hydraulic transport of solids in pipes, 1395
Hydrodynamically,
rough surface, 662
smooth surface, 662
Hypersonic flow, 947

I

Ideal fluid, 2, 12
Impulse momentum equation, 351
Impulse turbine, 1024
Indicator diagram, 1144

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Induced drag, 925
Instantaneous closure
of valve, 526
Integration of Euler's
equation, 291
Intergration of varied flow
equation, 797
Irrotational flow, 233, 252
Isentropic process, 940
Isentropic supersonic flow, 957
Isothermal process, 939

J

Jet contraction, 386
Jet, free, 317
Jet, reaction of, 357
Jet propulsion, 357
Journal bearing, 627
Jump, Hydraulic, 801

K

Kaplan turbine, 1044
Karman constant, 663
Karman-Prandtl equations, 665
Karman vortex trail, 906
Kennedy's theory, 1333
Kinematic eddy viscosity, 659
Kinematic similarity, 854
Kinematic viscosity, 11
Kinetic energy correction
factor, 301
Kutta-Joukowski equation, 918

L

Lacey's theory, 1334
Lagrangian method, 229
Laminar flow, 233, 601
between parallel plates, 612
in channels, 619
in circular pipes, 603
through porous media, 620
Laminar sublayer, 570, 582
Laplace equation, 256
Large orifice, 394
de Laval nozzle, 955
Length of jump, 805
Lift, coefficient, 896
definition, 915
development, 915
due to circulation, 916
effect of compressibility, 925
on an airfoil, 919
Liquid jets, 317
List of experiments, 1345
Local acceleration, 247
Location of hydraulic jump, 806
Logarithmic velocity
distribution, 665
Long pipe, 507
Loss, due to friction, 499
due to sudden
contraction, 403, 502
due to sudden
expansion, 401, 502
in pipe fittings, 407, 503

M

Mach angle, 947
Mach cone, 947
Mach number, 857, 946
Magnus effect, 918
Manning's formula, 501, 710
Manning's roughness
factor, table of, 710
Manometers, 49
Manometric, efficiency, 1192
head, 1187
Mass density, 7
Measurement of
depth, 1342
discharge, 305, 742, 959
flow direction, 963
pressure, 49, 1341
rainfall, 1280
snowfall, 1283
velocity, 314, 739, 960, 1342
Metacentre, 157
Metacentric height, 157
Metric slug, 4
Metric system, 3
Micromanometer, 61
Mild slope, 789
Minor losses, 502
Mixing length, 660
Model laws, 858
Model scale ratios, 859
Models, distorted, 861
undistorted, 861
Modular limit of weir, 473
Momentum, angular, 365
correction factor, 354
equation, 351, 942
Momentum thickness, 568
Moody's diagram, 678
Mouthpieces, 407
Multicylinder pumps, 1158
Multistage pumps, 1202

N

Nappe, 454
Navier-Stokes' equation, 288
Needle valve, 1393
Newton number, 857
Newtonian fluids, 12
Nikuradse's experiments, 662
Non-uniform flow, 232
in channels, 783
Normal depth, 708
No-slip condition, 567
Notch, rectangular, 455
trapezoidal, 465
V or triangular, 463
Nozzle
convergent, 952
convergent-divergent, 955
Nozzle meter, 313
NPSH, 1208

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

O

Omitted variable, 850
One dimensional flow, 233
Open channels, flow in, 702
  Optimum cross-section, 712
Orifice, discharge through, 384
  large, 394
Orifice meter, 310
Orifice submerged, partially, 399
  totally, 398
Orifice tank, 357
Oseen's formula, 901

P

Parshall flume, 738
Pascal's law, 47
Path line, 235
Pelton wheel, 1028
Performance, of
  pumps, 1158, 1204
  of turbines, 1088
Perimeter, wetted, 704
'Pi' method Buckingham's, 844
Piezometer tube, 49
Piezometric head, 294
Pipe bend, force on, 355
  as a meter, 314
Pipe networks, 531
Pipe, varying sections, 517, 950
Pipe with side tappings, 519
Pipes, branched, 512
Pipe, byepass, 511
Pipes in parallel, 510
Pipes in series, 508
Pipe-siphon, 515
Pipe tapering, 517
Pitot cylinder, 963
Pitot sphere, 964
Pitot static tube, 316
Pitot tube, 314, 960
Plug valve, 1393
Poise, 11
Polar diagram, 928
Potential flow, 257
Prandtl Glauert rule, 925
Prandtl's boundary
  layer theory, 567
Prandtl's pitot tube, 317
Precipitation, 1280
Pressure, 36
  absolute, 48
  atmospheric, 48
  gage, 48
  total, 93
  vacuum, 48
Pressure, centre of, 95
  equation, 293
Pressure coefficient, 857, 948
Pressure diagram, 102
Pressure distribution around
  airfoil, 912
  hydrostatic, 105
Pressure drag, 897
Pressure force, 287, 855
Pressure gages, 49, 63
Pressure gradient, 601
Pressure head, 39, 294
Pressure rise due to
  gradual closure of valve, 526
  instantaneous closure
  of valve, 526
Pressure transducers, 1341
Pressure variation, 36
Preston tube, 1344
Principle of conservation
  of energy, 236
  of mass, 236
  of momentum, 236
Prismatic channel, 703
Properties of common fluids, 18
Propeller, 362
Propeller turbine, 1027
Propeller type currentmeter, 739
Proportional weir, 475
Pumps, air lift, 1260
  centrifugal, 1178
  reciprocating, 1132

R

Radial flow, 323
Rain gages, 1280
Rankine-Hugoniot equations, 959
Rapid flow in open channel, 783
Rayleigh line, 958
Rayleigh-Pitot equation, 961
Reaction turbine, 1027
Real fluids, 2
Reciprocating pump, 1132
  components, 1132
Rehbock's formula, 461
Relative equilibrium, 190
Relief valve, 1050
Residual drag, 912
Resistance coefficient, 388
Reversible process, 940
Reynolds equation, 287
Reynolds experiment, 494
Reynolds number, 495, 856
Rheology, 12
Rotameter, 313
Rotational flow, 233, 251
Rotation, pure, 251
Rough boundary, 662
Rough pipes, resistance of, 671
Runaway speed, 1051
Runoff, 1288

S

Salt dilution method, 1343
Salt velocity method, 1342
Scale effect, 862
Scale ratios for models, 853
Separation, 583
Sharp crested weir, 455
Shear velocity, 582, 608
Shear stress in turbulent flow, 658

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

Shock wave, 956
normal, 957
'SI' units, 4
Similarity, 853
Similitude, 853
Similarity laws for
general models, 858
pumps, 1200
turbines, 1102
Siphon, 515
Siphon spillway, 474
Slip, 1138
Slipper bearing, 623
Slopes, open channel, 789
adverse, 790
critical, 789
horizontal, 790
mild, 789
steep, 790
Smooth boundary, 662
Smooth pipes, resistance, 671
Sonic velocity, 944, 947
Specific energy, 723
curve, 724
Specific force, 726
curve, 727
Specific gravity, 8
Specific speed, pumps, 1198
turbines, 1092
Specific volume, 8
Specific weight, 7
Speed ratio, 1033
Spillway, 473
Stability of floating bodies, 158
Stagnation point, 314
Stagnation pressure, 314, 949
Stall point, 924
Stalling angle, 924
Standing wave, 737, 801
Stanton diagram, 674
Stanton tube, 1343
Starting vortex, 922
Steady flow, 231
energy equation, 293
Steep slope, 790
Step method of integration, 797
Stokes law, 622
Streak line, 235
Stream filament, 235
Stream function, 257
Stream line, 260
Stream lined body, 897
Stream tube, 234
Strickler's formula, 712
Strouhal number, 908
Subcritical flow, 723
Sublayer, laminar, 570, 582
Submerged, orifice, 398
weir, 472
Suction specific speed, 1107, 1209
Suction stroke, 1133
Supercritical flow, 724
Superfluous variable, 850
Surface profiles, 790
Surface tension, 14
Surges, 809
Surge tanks, 1051
System headcurves, 1210

T

Tail race, 1024
Tangential acceleration, 248
Thickness of boundary layer, 567
laminar sublayer, 570, 582
Thixotropic liquids, 12
Thoma's cavitation
factor, 1106, 1209
Thomson's theorem, 923
Three dimensional flow, 232
Time of emptying, 417, 467
Tip vortices, 926
Torricelli's formula, 386
Total energy line, 503
Total head, 294
Tranquil flow, 723
Transitions, channels, 732
Transition from laminar to
turbulent flow, 495
Translation, pure, 251
Transmission of power
through pipes, 522
Transport of solids in pipes, 1395
Turbines, types, 1027
efficiency, 1024
Francis, 1036
heads, 1024
impulse, 1027
Kaplan, 1044
Pelton, 1028
performance, 1088
reaction, 1027
working proportions,
1033, 1040, 1045
Turbulent boundary
layer, 570, 580
Turbulent flow, 233, 658
velocity distribution, 663
Turbulent mixing, 659
Turbulent shear stress, 658
Two dimensional flow, 232
Types of flow, 231

U

Undistorted models, 861
Uniform depth, 708
Uniform flow, 231, 703
Unit discharge, 1090
Unit power, 1090
Unit speed, 1089
Units of measurement, 3
Universal gas constant, 9
Unsteady flow, 231

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

V

Vacuum pressure, 48
Vapour pressure, 12
Variables, dependent, 843
  independent, 843
  non-repeating, 845
  repeating, 845
Varied flow
  gradually, 783
  rapidly, 783, 801
Velocity, approach, 385, 456
  critical, 495, 723
  defect, 665
  fluctuation, 660
  gradient, 663
  head, 294
  local, 606
  maximum, 606
  shear, 582, 608
Velocity distribution
  between parallel plates, 612
  in boundry layer, 570
  in open channels, 706
  in turbulent flow, 663
  near rough boundary, 669
  near smooth boundary, 666
Velocity, measurement of, 314, 739, 960, 1342
Velocity of sound, 944, 947
Velocity of whirl, 995
Velocity potential, 256
Venacontracta, 384
Ventilation of nappe, 461
Venturi flume, 736
Venturi meter, 305
Viscometer
  coaxial-cylinder, 633
  capillary tube, 635
  falling sphere, 637
  Redwood, 637
  Saybolt, 636
Viscosity
  dynamic, 11
  eddy, 659
  kinematic, 11
Viscosity index, 630
Volumetric efficiency
  pumps, 1193
  turbines, 1026
Volute casing, 1182
Vortex, free, 319
  forced, 320
  Rankine, 321
  spiral, 323
  Starting, 922
Vortex pair, 907
Vortex street, 907
Vorticity, 254

W

Wake, 895, 897, 898, 906
Water hammer, 526
Weber number, 858
Weir, broad crested, 470
  Cipolletti, 465
  proportional, 475
  rectangular, 455
  sharp crested, 455
  submerged, 472
  trapezoidal, 465
  triangular, 463
Wetted perimeter, 704
Writing of report, 1347

Downloaded From : www.EasyEngineering.net

Downloaded From : www.EasyEngineering.net

FOR MORE EXCLUSIVE

(Civil, Mechanical, EEE, ECE)

ENGINEERING & GENERAL STUDIES

(Competitive Exams)

TEXT BOOKS, IES GATE PSU's TANCET & GOVT EXAMS
NOTES & ANNA UNIVERSITY STUDY MATERIALS

VISIT

www.EasyEngineering.net

AN EXCLUSIVE WEBSITE FOR ENGINEERING STUDENTS & GRADUATES

EASY ENGINEERING
Apprise Education, Reprise Innovations

**Note : Other Websites/Blogs Owners we requested you, Please do not Copy (or) Republish this Material.

This copy is NOT FOR SALE.

**Disclimers : EasyEngineering does not own this book/materials, neither created nor scanned. we provide the links which is already available on the internet. For any quarries, Disclaimer are requested to kindly contact us. We assured you we will do our best. We DO NOT SUPPORT PIRACY, this copy was provided for students who are financially troubled but deserving to learn.

Thank You and God Bless!

Downloaded From : www.EasyEngineering.net