Fluid Mechanics

with Engineering Applications

Tenth Edition



E. John Finnemore

Joseph B. Franzini

INTERNATIONAL EDITION

The McGraw-Hill Series in Civil and Environmental Engineering

ENGINEERING ECONOMY

Blank and Tarquin: *Engineering Economy*
Humphreys: *Jelen's Cost and Optimization Engineering*
Riggs, Bedworth, Randhawa: *Engineering Economics*
Steiner: *Engineering Economic Principles*

ENGINEERING MATH AND STATISTICS

Bannerjee: *The Boundary Element Methods in Engineering*
Barnes: *Statistical Analysis for Engineers and Scientists: A Computer-Based Approach (IBM)*
Ledbed: *Formulas for Structural Dynamics*
Milton and Arnold: *Introduction to Probability and Statistics: Principles and Applications for Engineering and the Computing Sciences*
Reddy: *Introduction to the Finite Element Method*
Rosenkrantz: *Introduction to Probability and Statistics for Scientists and Engineers*
Zienkiewicz and Taylor: *The Finite Element Method: Basic Concepts and Linear Applications*

FLUID MECHANICS

Çengel and Turner: *Fundamentals of Thermal-Fluid Sciences*
Finnemore and Franzini: *Fluid Mechanics with Engineering Applications*
Streeter, Bedford, Wylie: *Fluid Mechanics*
White: *Fluid Mechanics*

GEOTECHNICAL ENGINEERING

Atkinson: *Introduction to the Mechanics of Soils and Foundations*
Bowles: *Foundation Analysis and Design*
Bowles: *Engineering Properties of Soils and Their Measurement*

NUMERICAL METHODS

Chapra and Canale: *Numerical Methods for Engineers*
Heath: *Scientific Computing: An Introductory Survey*

STRUCTURES

Gaylord and Stallmeyer: *Design of Steel Structures*
Laursen: *Structural Analysis*
Leet and Bernal: *Reinforced Concrete Design*
Leet and Uang: *Fundamentals of Structural Analysis*
Leonard: *Tension Structures: Behavior and Analysis*
Lin and Cai: *Probabilistic Structural Dynamics: Advanced Theory and Applications*
Nilson: *Design of Concrete Structures*
Nowak and Collins: *Reliability of Structures*
Taly: *Design of Modern Highway Bridges*
Taly: *Reinforced Masonry Structure Design*

SURVEYING

Anderson and Mikhail: *Surveying: Theory and Practice*
Wolf and DeWitt: *Elements of Photogrammetry (with Applications in GIS)*

STATICS, DYNAMICS, AND MECHANICS OF MATERIALS

Barber: *Intermediate Mechanics of Materials*
Beer and Johnston: *Vector Mechanics for Engineers: Statics*
Beer and Johnston: *Vector Mechanics for Engineers: Dynamics*
Beer and Johnston: *Vector Mechanics for Engineers: Statics and Dynamics*
Beer and Johnston: *Mechanics of Materials*
Young: *Roark's Formulas for Stress and Strain*

CONSTRUCTION ENGINEERING AND PROJECT MANAGEMENT

Raymond E. Levitt, *Stanford University,* Consulting Editor

Barrie and Paulson: *Professional Construction Management*
Bockrath: *Contracts and the Legal Environment for Engineers and Architects*
Callahan, Quackenbush, Rowlings: *Construction Project Scheduling*
Griffis and Farr: *Construction Planning for Engineers*
Hinze: *Construction Contracts*
Oberlender: *Project Management for Engineering and Construction*
Peurifoy, Ledbetter, Schexnayder: *Construction Planning, Equipment, and Methods*
Peurifoy and Oberlender: *Estimating Construction Costs*

TRANSPORTATION ENGINEERING

Edward K. Morlok, *University of Pennsylvania,* Consulting Editor

Banks: *Introduction to Transportation Engineering*
Horonjeff and McKelvey: *Planning and Design of Airports*
Kanafani: *Transportation Demand Analysis*
Meyer and Miller: *Urban Transportation Planning*
Wells: *Airport Planning and Management*

WATER RESOURCES AND ENVIRONMENTAL ENGINEERING

George Tchobanoglous, *University of California, Davis,* Consulting Editor

Bailey and Ollis: *Biochemical Engineering Fundamentals*
Benjamin: *Water Chemistry*
Bishop: *Pollution Prevention: Fundamentals and Practice*
Canter: *Environmental Impact Assessment*
Chanlett: *Environmental Protection*
Chapra: *Surface Water Quality Modeling*
Chow, Maidment, Mays: *Applied Hydrology*
Crites and Tchobanoglous: *Small and Decentralized Wastewater Management Systems*
Davis and Cornwell: *Introduction to Environmental Engineering*
deNevers: *Air Pollution Control Engineering*
Eckenfelder: *Industrial Water Pollution Control*
Eweis, Ergas, Chang, Schroeder: *Bioremediation Principles*
Freeman: *Hazardous Waste Minimization*
LaGrega, Buckingham, Evans: *Hazardous Waste Management*
Linsley, Franzini, Freyberg, Tchobanoglous: *Water Resources Engineering*
McGhee: *Water Supply and Sewage*
Metcalf & Eddy, Inc.: *Wastewater Engineering: Collection and Pumping of Wastewater*
Metcalf & Eddy, Inc.: *Wastewater Engineering: Treatment and Reuse*
Peavy, Rowe, Tchobanoglous: *Environmental Engineering*
Rittmann and McCarty: *Environmental Biotechnology: Principles and Applications*
Rubin: *Introduction to Engineering and the Environment*
Sawyer, McCarty, Parkin: *Chemistry for Environmental Engineering*
Sturm: *Open Channel Hydraulics*
Tchobanoglous, Theisen, Vigil: *Integrated Solid Waste Management: Engineering Principles and Management Issues*
Wentz: *Safety, Health, and Environmental Protection*

OTHERS TITLES OF INTEREST

Budynas: *Advanced Strength and Applied Stress Analysis*
Dally and Riley: *Experimental Stress Analysis*
Ugural: *Stresses in Plates and Shells*

Fluid Mechanics with Engineering Applications

TENTH EDITION

E. John Finnemore
Professor of Civil Engineering
Santa Clara University

Joseph B. Franzini
Professor Emeritus of Civil Engineering
Stanford University

UNIVERSIDAD POLITÉCNICA SALESIANA
BIBLIOTECA CUENCA
CL004531

Boston Burr Ridge, IL Dubuque, IA Madison, WI New York San Francisco St. Louis
Bangkok Bogotá Caracas Kuala Lumpur Lisbon London Madrid Mexico City
Milan Montreal New Delhi Santiago Seoul Singapore Sydney Taipei Toronto

McGraw-Hill Higher Education
A Division of The McGraw-Hill Companies

FLUID MECHANICS WITH ENGINEERING APPLICATIONS, TENTH EDITION

Published by McGraw-Hill, a business unit of The McGraw-Hill Companies, Inc., 1221 Avenue of the Americas, New York, NY 10020. Copyright © 2002, 1997, 1985, 1977, 1965, 1954 by The McGraw-Hill Companies, Inc. All rights reserved. Formerly published under the title of *Hydraulics,* copyright © 1937, 1925, 1919, 1916 by McGraw-Hill, Inc. All rights reserved. Copyright renewed 1953, 1965 by R. L. Daugherty. Copyright renewed 1982 by Marguerite R. Daugherty. No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior written consent of The McGraw-Hill Companies, Inc., including, but not limited to, in any network or other electronic storage or transmission, or broadcast for distance learning.

Some ancillaries, including electronic and print components, may not be available to customers outside the United States.

This book is printed on acid-free paper.

International 1 2 3 4 5 6 7 8 9 0 DOC/DOC 0 9 8 7 6 5 4 3 2 1
Domestic 1 2 3 4 5 6 7 8 9 0 DOC/DOC 0 9 8 7 6 5 4 3 2 1

ISBN 0–07–243202–0
ISBN 0–07–112196–X (ISE)

General manager: *Thomas E. Casson*
Publisher: *Elizabeth A. Jones*
Sponsoring editor: *Suzanne Jeans*
Developmental editor: *Amy Hill*
Executive marketing manager: *John Wannemacher*
Senior project manager: *Gloria G. Schiesl*
Production supervisor: *Enboge Chong*
Coordinator of freelance design: *Rick D. Noel*
Interior design: *Kathleen Theis*
Cover design: *Ellen Pettengell*
Senior photo research coordinator: *Lori Hancock*
Photo research: *Alexandra Truitt & Jerry Marshall, www.pictureresearching.com*
Supplement producer: *Brenda A. Ernzen*
Media technology senior producer: *Phillip Meek*
Compositor: *Interactive Composition Corporation*
Typeface: *10.5/12 Times Ten Roman*
Printer: *R. R. Donnelley & Sons Company/Crawfordsville, IN*
Cover images: Top right and bottom right: © Tony Stone Images, C137DAQ0034-001R Glen Canyon Dam, worker/jets, and T36D091268-004R Kurobe Dam, Toyama, Japan; left: Mossyrock Dam, photo courtesy of Harza Engineering Company, Chicago, Illinois.

Library of Congress Cataloging-in-Publication Data

Finnemore, E. John.
Fluid mechanics with engineering applications / E. John Finnemore, Joseph B. Franzini. — 10th ed.
p. cm. — (McGraw-Hill series in civil and environmental engineering)
Franzini's name appears first on the earlier ed.
Includes bibliographical references and index.
ISBN 0–07–243202–0 — ISBN 0–07–112196–X (ISE)
1. Fluid mechanics. I. Franzini, Joseph B. II. Title. III. Series.

TA357 .F695 2002
620.1'06—dc21 2001044672
CIP

INTERNATIONAL EDITION ISBN 0–07–112196–X
Copyright © 2002. Exclusive rights by The McGraw-Hill Companies, Inc., for manufacture and export. This book cannot be re-exported from the country to which it is sold by McGraw-Hill. The International Edition is not available in North America.

www.mhhe.com

About the Authors

E. John Finnemore is Professor of Civil Engineering at Santa Clara University, California. Born in London, England, he received a B.Sc. (Eng.) degree from London University in 1960, and M.S. and Ph.D. degrees from Stanford University in 1966 and 1970, all in civil engineering. Finnemore worked with consulting civil engineers in England and Canada for five years before starting graduate studies, and for another seven years in California after completing his doctorate. He served one year on the faculty of Pahlavi University in Shiraz, Iran, and he has been a member of the faculty at Santa Clara University since 1979. He has taught courses in fluid mechanics, hydraulic engineering, hydrology, and water resources engineering, and has authored numerous technical articles and reports in several related fields. His research has often involved environmental protection, such as in stormwater management and onsite wastewater disposal. Professor Finnemore has served on governmental review boards and as a consultant to various private concerns. He is a Fellow of the American Society of Civil Engineers and a registered civil engineer in Britain and California. He lives with his wife Gulshan in Cupertino, California.

Joseph B. Franzini is Professor Emeritus of Civil Engineering at Stanford University. Born in Las Vegas, New Mexico, he received B.S. and M.S. degrees from the California Institute of Technology in 1942 and 1943, and a Ph.D. from Stanford University in 1950. All his degrees are in civil engineering. Franzini served on the faculty at Stanford University from 1950 to 1986. There he taught courses in fluid mechanics, hydrology, sedimentation, and water resources, and also did research on a number of topics in those fields. Since retirement from Stanford, he has been active as an engineering consultant and an expert witness. He is coauthor of the authoritative and widely used textbook, *Water Resources Engineering,* and of its predecessor, *Elements of Hydraulic Engineering.* Through the years, Franzini has been active as a consultant to various private organizations and governmental agencies in both the United States and abroad; he was associated with Nolte and Associates, a consulting civil engineering firm in San Jose, California, for over 30 years. He is a Fellow of the American Society of Civil Engineers and a registered civil engineer in California. He lives with his wife Gloria in Palo Alto, California.

BIBLIOTECA U.P.S.
620.106/F514f/5322002
Clasificación
No Inventario CL00 4531
Precio $
Fecha Adquisición

BIBLIOTECA U.P.S.
Clasificación
No Inventario
Precio $
Fecha Adquisición

To that great love which encourages humanity in all its noble endeavors

and

to Gulshan and Gloria for their loving support

Brief Contents

Preface xv
List of Symbols xix
List of Abbreviations xxv

Chapter 1 Introduction 1
Chapter 2 Properties of Fluids 13
Chapter 3 Fluid Statics 45
Chapter 4 Basics of Fluid Flow 97
Chapter 5 Energy in Steady Flow 127
Chapter 6 Momentum and Forces in Fluid Flow 185
Chapter 7 Similitude and Dimensional Analysis 232
Chapter 8 Steady Incompressible Flow in Pressure Conduits 255
Chapter 9 Forces on Immersed Bodies 356
Chapter 10 Steady Flow in Open Channels 407
Chapter 11 Fluid Measurements 491
Chapter 12 Unsteady-Flow Problems 546
Chapter 13 Steady Flow of Compressible Fluids 580
Chapter 14 Ideal Flow Mathematics 622
Chapter 15 Hydraulic Machinery—Pumps 647
Chapter 16 Hydraulic Machinery—Turbines 685

Appendixes

A ***Fluid and Geometric Properties 729***
B ***Equations in Fluid Mechanics 740***
C ***Programming and Computer Applications 745***
D ***Examples of Using Solvers 754***
E ***References 764***
F ***Answers to Exercises 769***

Index 777

Contents

Preface xv
List of Symbols xix
List of Abbreviations xxv

Chapter 1 Introduction 1

1.1 Scope of Fluid Mechanics 1
1.2 Historical Sketch of the Development of Fluid Mechanics 2
1.3 The Book, Its Contents, and How to Best Study Fluid Mechanics 3
1.4 Approach to Problem Solving 4
1.5 Dimensions and Units 6

Chapter 2 Properties of Fluids 13

2.1 Distinction Between a Solid and a Fluid 13
2.2 Distinction Between a Gas and a Liquid 13
2.3 Density, Specific Weight, Specific Volume, and Specific Gravity 14
2.4 Compressible and Incompressible Fluids 16
2.5 Compressibility of Liquids 17
2.6 Specific Weight of Liquids 19
2.7 Property Relations for Perfect Gases 22
2.8 Compressibility of Perfect Gases 25
2.9 Standard Atmosphere 27
2.10 Ideal Fluid 29
2.11 Viscosity 29
2.12 Surface Tension 37
2.13 Vapor Pressure of Liquids 40
Problems 41

Chapter 3 Fluid Statics 45

3.1 Pressure at a Point the Same in All Directions 45
3.2 Variation of Pressure in a Static Fluid 46
3.3 Pressure Expressed in Height of Fluid 50
3.4 Absolute and Gage Pressures 53
3.5 Measurement of Pressure 55
3.6 Force on a Plane Area 66
3.7 Center of Pressure 68
3.8 Force on a Curved Surface 77
3.9 Buoyancy and Stability of Submerged and Floating Bodies 81
3.10 Liquid Masses Subjected to Acceleration 88
Problems 92

Chapter 4 Basics of Fluid Flow 97

4.1 Types of Flow 97
4.2 Laminar and Turbulent Flow 98
4.3 Steady Flow and Uniform Flow 101
4.4 Path Lines, Streamlines, and Streak Lines 102
4.5 Flow Rate and Mean Velocity 103
4.6 Fluid System and Control Volume 106
4.7 Equation of Continuity 108
4.8 One-, Two-, and Three-Dimensional Flow 110
4.9 The Flow Net 111
4.10 Use and Limitations of the Flow Net 114
4.11 Frame of Reference in Flow Problems 117
4.12 Velocity and Acceleration in Steady Flow 117
4.13 Velocity and Acceleration in Unsteady Flow 121
Problems 124

Chapter 5 Energy in Steady Flow 127

5.1 Energies of a Flowing Fluid 127
5.2 Equation for Steady Motion of an Ideal Fluid Along a Streamline, and Bernoulli's Theorem 131
5.3 Equation for Steady Motion of a Real Fluid Along a Streamline 135
5.4 Pressure in Fluid Flow 138
5.5 General Energy Equation for Steady Flow of Any Fluid 140
5.6 Energy Equations for Steady Flow of Incompressible Fluids, Bernoulli's Theorem 143
5.7 Energy Equation for Steady Flow of Compressible Fluids 147
5.8 Head 150
5.9 Power Considerations in Fluid Flow 150
5.10 Cavitation 154
5.11 Definition of Hydraulic Grade Line and Energy Line 158
5.12 Loss of Head at Submerged Discharge 160
5.13 Application of Hydraulic Grade Line and Energy Line 161
5.14 Method of Solution of Liquid Flow Problems 165
5.15 Jet Trajectory 169
5.16 Flow in a Curved Path 172
5.17 Forced or Rotational Vortex 173
5.18 Free or Irrotational Vortex 176

Problems 179

Chapter 6 Momentum and Forces in Fluid Flow 185

6.1 Development of the Momentum Principle 185
6.2 Navier-Stokes Equations 188
6.3 Momentum Correction Factor 189
6.4 Applications of the Momentum Principle 190
6.5 Force on Pressure Conduits 193
6.6 Force of a Free Jet on a Stationary Vane or Blade 198
6.7 Moving Vanes: Relation Between Absolute and Relative Velocities 200
6.8 Force of a Jet on One or More Moving Vanes or Blades 201
6.9 Reaction of a Jet 206
6.10 Jet Propulsion 210
6.11 Rotating Machines: Continuity, Relative Velocities, Torque 212
6.12 Head Equivalent of Mechanical Work 219
6.13 Flow Through a Rotating Channel 219
6.14 Reaction with Rotation 220
6.15 Momentum Principle Applied to Propellers and Windmills 222

Problems 226

Chapter 7 Similitude and Dimensional Analysis 232

7.1 Definition and Uses of Similitude 232
7.2 Geometric Similarity 232
7.3 Kinematic Similarity 233
7.4 Dynamic Similarity 234
7.5 Scale Ratios 241
7.6 Comments on Models 243
7.7 Dimensional Analysis 245

Problems 252

Chapter 8 Steady Incompressible Flow in Pressure Conduits 255

8.1 Laminar and Turbulent Flow 255
8.2 Critical Reynolds Number 256
8.3 Hydraulic Radius, Hydraulic Diameter 258
8.4 Friction Head Loss in Conduits of Constant Cross Section 258
8.5 Friction in Circular Conduits 261
8.6 Friction in Noncircular Conduits 263
8.7 Laminar Flow in Circular Pipes 264
8.8 Entrance Conditions in Laminar Flow 265
8.9 Turbulent Flow 268
8.10 Viscous Sublayer in Turbulent Flow 271

8.11 Velocity Profile in Turbulent Flow 276
8.12 Pipe Roughness 280
8.13 Chart for Friction Factor 282
8.14 Single-Pipe Flow: Solution Basics 285
8.15 Single-Pipe Flow: Solution by Trials 287
8.16 Single-Pipe Flow: Direct Solutions 293
8.17 Single-Pipe Flow: Automated Solutions 296
8.18 Empirical Equations for Single-Pipe Flow 298
8.19 Nonrigorous Head-Loss Equations 300
8.20 Minor Losses in Turbulent Flow 301
8.21 Loss of Head at Entrance 302
8.22 Loss of Head at Submerged Discharge 303
8.23 Loss Due to Contraction 305
8.24 Loss Due to Expansion 307
8.25 Loss in Pipe Fittings 312
8.26 Loss in Bends and Elbows 312
8.27 Single-Pipe Flow with Minor Losses 315
8.28 Pipeline with Pump or Turbine 321
8.29 Branching Pipes 326
8.30 Pipes in Series 333
8.31 Pipes in Parallel 336
8.32 Pipe Networks 339
8.33 Further Topics in Pipe Flow 343

Problems 344

Chapter 9 Forces on Immersed Bodies 356

9.1 Introduction 356
9.2 Friction Drag of Boundary Layer—Incompressible Flow 358
9.3 Laminar Boundary Layer for Incompressible Flow Along a Smooth Flat Plate 360
9.4 Turbulent Boundary Layer for Incompressible Flow Along a Smooth Flat Plate 365
9.5 Friction Drag for Incompressible Flow Along a Smooth Flat Plate with a Transition Regime 369
9.6 Boundary-Layer Separation and Pressure Drag 372
9.7 Drag on Three-Dimensional Bodies (Incompressible Flow) 374
9.8 Drag on Two-Dimensional Bodies (Incompressible Flow) 382
9.9 Lift and Circulation 385
9.10 Ideal Flow About a Cylinder 387
9.11 Lift of an Airfoil 390
9.12 Induced Drag on Airfoil of Finite Length 392
9.13 Lift and Drag Diagrams 395
9.14 Effects of Compressibility on Drag and Lift 399
9.15 Concluding Remarks 401

Problems 402

Chapter 10 Steady Flow in Open Channels 407

10.1 Open Channels 407
10.2 Uniform Flow 409
10.3 Solution of Uniform Flow Problems 414
10.4 Velocity Distribution in Open Channels 419
10.5 "Wide and Shallow" Flow 421
10.6 Most Efficient Cross Section 422
10.7 Circular Sections Not Flowing Full 426
10.8 Laminar Flow in Open Channels 429
10.9 Specific Energy and Alternate Depths of Flow in Rectangular Channels 431
10.10 Subcritical and Supercritical Flow 436
10.11 Critical Depth in Nonrectangular Channels 438
10.12 Occurrence of Critical Depth 441
10.13 Humps and Contractions 442
10.14 Nonuniform, or Varied, Flow 448
10.15 Energy Equation for Gradually Varied Flow 449
10.16 Water-Surface Profiles in Gradually Varied Flow (Rectangular Channels) 452
10.17 Examples of Water-Surface Profiles 456

10.18 The Hydraulic Jump 460
10.19 Location of Hydraulic Jump 465
10.20 Velocity of Gravity Waves 468
10.21 Flow Around Channel Bends 471
10.22 Transitions 474
10.23 Hydraulics of Culverts 476
10.24 Further Topics in Open-Channel Flow 480
Problems 481

Chapter 11 Fluid Measurements 491

11.1 Measurement of Fluid Properties 491
11.2 Measurement of Static Pressure 495
11.3 Measurement of Velocity with Pitot Tubes 496
11.4 Measurement of Velocity by Other Methods 500
11.5 Measurement of Discharge 503
11.6 Orifices, Nozzles, and Tubes 505
11.7 Venturi Meter 515
11.8 Flow Nozzle 519
11.9 Orifice Meter 522
11.10 Flow Measurement of Compressible Fluids 524
11.11 Thin-Plate Weirs 527
11.12 Streamlined Weirs and Free Overfall 533
11.13 Overflow Spillway 536
11.14 Sluice Gate 538
11.15 Measurement of Liquid-Surface Elevation 540
11.16 Other Methods of Measuring Discharge 540
Problems 541

Chapter 12 Unsteady-Flow Problems 546

12.1 Introduction 546
12.2 Discharge with Varying Head 546
12.3 Unsteady Flow of Incompressible Fluids in Pipes 550
12.4 Approach to Steady Flow 554
12.5 Velocity of Pressure Wave in Pipes 558
12.6 Water Hammer 559
12.7 Surge Tanks 569
Problems 574

Chapter 13 Steady Flow of Compressible Fluids 580

13.1 Thermodynamic Considerations 580
13.2 Fundamental Equations Applicable to the Flow of Compressible Fluids 584
13.3 Speed of Sound 585
13.4 Adiabatic Flow (With or Without Friction) 588
13.5 Stagnation Properties 589
13.6 Isentropic Flow 593
13.7 Effect of Area Variation on One-Dimensional Compressible Flow 594
13.8 Compressible Flow Through a Converging Nozzle 596
13.9 Isentropic Flow Through a Converging-Diverging Nozzle 600
13.10 One-Dimensional Shock Wave 603
13.11 The Oblique Shock Wave 607
13.12 Isothermal Flow 609
13.13 Isothermal Flow in a Constant-Area Duct 610
13.14 Adiabatic Flow in a Constant-Area Duct 614
13.15 Comparison of Flow Types 618
13.16 Concluding Remarks 619
Problems 619

Chapter 14 Ideal Flow Mathematics 622

14.1 Differential Equation of Continuity 622
14.2 Irrotational Flow 625
14.3 Circulation and Vorticity 627
14.4 The Stream Function 629
14.5 Basic Flow Fields 631
14.6 Velocity Potential 635

14.7 Orthogonality of Streamlines and Equipotential Lines 636
14.8 Flow Through Porous Media 639
Problems 642

Chapter 15 Hydraulic Machinery—Pumps 647

15.1 Description of Centrifugal and Axial-Flow Pumps 647
15.2 Head Developed by a Pump 651
15.3 Pump Efficiency 652
15.4 Similarity Laws for Pumps 652
15.5 Performance Characteristics of Pumps at Constant Speed 655
15.6 Performance Characteristics at Different Speeds and Sizes 658
15.7 Operating Point of a Pump 660
15.8 Specific Speed of Pumps 662
15.9 Peripheral-Velocity Factor 665
15.10 Cavitation in Pumps 666
15.11 Viscosity Effect 671
15.12 Selection of Pumps 671
15.13 Pumps Operating in Series and in Parallel 675
15.14 Pump Installations 677
Problems 679

Chapter 16 Hydraulic Machinery—Turbines 685

16.1 Hydraulic Turbines 685
16.2 Impulse Turbines 686
16.3 Action of the Impulse Turbine 689
16.4 Head on an Impulse Turbine and Efficiency 691
16.5 Nozzles for Impulse Turbines 695
16.6 Reaction Turbines 697
16.7 Action of the Reaction Turbine 701
16.8 Draft Tubes and Effective Head on Reaction Turbines 702
16.9 Efficiency of Turbines 706
16.10 Similarity Laws for Reaction Turbines 708
16.11 Peripheral-Velocity Factor and Specific Speed of Turbines 711
16.12 Cavitation in Turbines 713
16.13 Selection of Turbines 717
16.14 Pump Turbine 719
16.15 Turbine Installations 720
Problems 722

Appendixes

A ***Fluid and Geometric Properties 729***
B ***Equations in Fluid Mechanics 740***
C ***Programming and Computer Applications 745***
D ***Examples of Using Solvers 754***
E ***References 764***
F ***Answers to Exercises 769***

Index 777

Preface

Philosophy and History

This tenth edition of the classic textbook, *Fluid Mechanics with Engineering Applications,* continues and improves on its tradition of explaining the physical phenomena of fluid mechanics and applying its basic principles in the simplest and clearest possible manner without the use of complicated mathematics. It focuses on civil, environmental, and agricultural engineering problems, although mechanical and aerospace engineering topics are also strongly represented. The book is written as a text for a first course in fluid mechanics for engineering students, with sufficient breadth of coverage that it can serve in a number of ways for a second course if desired.

Thousands of engineering students and practitioners throughout the world have used this book for over 85 years; it is widely distributed as an International Edition, and translations into Spanish and Korean are available. The book is now in its third generation of authorship. Though this tenth edition is very different from the first edition, it retains the same basic philosophy and presentation of fluid mechanics as an engineering subject that Robert L. Daugherty originally developed over his many years of teaching at Cornell University, Rensselaer Polytechnic Institute, and the California Institute of Technology. The first edition that Professor Daugherty authored was published in 1916 with the title *Hydraulics.* He revised the book four times. On the fifth edition (fourth revision) Dr. Alfred C. Ingersoll assisted him, and they changed the title of the book to *Fluid Mechanics with Engineering Applications.* The sixth and seventh editions were entirely the work of Professor Franzini. A student of Daugherty's at Caltech, Franzini had received his first exposure to the subject of fluid mechanics from the fourth edition of the book. Professor Franzini enlisted the services of Professor Finnemore, a former student of Franzini's at Stanford, to assist him with the eighth and ninth editions. This tenth edition is the work of Dr. Finnemore, with the exception of Chapters 15 and 16, which Dr. Franzini revised.

The Book, Its Organization

We feel it is most important that the engineering student clearly visualize the physical situation under consideration. Throughout the book, therefore, we place considerable emphasis on physical phenomena of fluid mechanics. We stress the governing principles, the assumptions made in their development, and their limits of applicability, and show how we can apply the principles to the solution of practical engineering problems. The emphasis is on teachability for the instructor and on clarity for both the instructor and the student, so that they can readily grasp basic principles and applications. Numerous worked sample problems are

presented to demonstrate the application of basic principles. These sample problems also help to clarify the text. Drill exercises with answers provided follow most sections to help students rapidly reinforce their understanding of the subjects and concepts. The end-of-chapter problems presented for assignment purposes were carefully selected to provide the student with a thorough workout in the application of basic principles. Only by working numerous exercises and problems will students experience the evolution so necessary to the learning process. We recommend ways to study fluid mechanics and to approach problem solving in Chapter 1.

The book is essentially "self-contained." The treatment is such that an instructor generally need not resort to another reference to answer any question that a student might normally be expected to ask. This has required more detailed discussion than that needed for a more superficial presentation of certain topics. A list of selected references is provided at the end of the book to serve as a guide for those students who wish to probe deeper into the various fields of fluid mechanics. The appendix section contains information on physical properties of fluids and other useful tables, Chapter 1 contains information on dimensions and units, and, for convenient reference, the insides of the covers contain conversion factors and important quantities and definitions.

Even though we use British Gravitational (BG) units (feet, slugs, seconds, pounds) as the primary system of units, we give the corresponding SI units in the text. We provide sample problems, and exercises and problems in BG and in SI units in near equal numbers. We have made every effort to ease the changeover from BG units to SI units; Chapter 1 includes a discussion of unit systems and conversion of units. We encourage instructors to assign problems in each system so that students become conversant with both.

Improvements to This Edition

Probably the most noticeable improvement made throughout this edition will be our addition of many figures (over 110), to help present exercises and problems, and to help explain solutions of sample problems. Also, throughout we have made the use of programming and computers optional, we have included more ways to solve trial-and-error problems, and we have added more cross-references.

In this revision, we have given special attention to the first eight chapters. There we have improved understandability by simplifying and clarifying text and sample problems that were more involved, and by thoroughly modernizing the language, as well as adding figures. Of the exercises and problems in these chapters, 40% are now new or changed from the previous edition.

Chapter 5 is strongly revised, with the basic derivation of Bernoulli's equation moved to a very early position, alternate forms of the equation added, and the assumptions on which it is based clarified. A new, clear distinction is made between wall (or pipe) friction head loss and total head loss in pipes, and this is carried forward into subsequent chapters. How cavitation causes damage is better explained with the aid of a new microphotograph of an imploding bubble.

New features in other chapters include: information about computational fluid dynamics, with a supporting figure and photograph; various aspects of single-pipe flow are now separated out into different sections; a treatment of submerged discharge into moving water; information about conveyance in open channels; a clarified treatment of optimal hydraulic efficiency of channel flow; a table summarizing damming action; descriptions of methods of measuring fluid velocity using laser light; data on the hydraulic conductivities of major geologic deposits; and a discussion of affinity laws for pumps. We have increased the total number of exercises and problems in the book to 1354.

There are two new appendices. One summarizes the characteristics and properties of the main types of equations used in fluid mechanics. The other provides examples of using equation solvers and polynomial solvers, on HP48G calculators and in Excel and Mathcad, to solve selected sample problems. In addition, Appendix C, on programming and computer applications, is upgraded by the addition of many examples of applications software that model flow systems, components, processes, and flow fields.

Use of the Book, Course Planning

An excellent, brief first course in fluid mechanics could consist of Chapters 1 through 7 and the first half of Chapter 8; however, one might wish to include parts of Chapters 11 (Fluid Measurements) and 14 (Ideal Flow Mathematics) in a first course. Schools having stringent requirements in fluid mechanics might wish to cover the entire text in their course or courses required of all engineers. At other schools only partial coverage of the text might suffice for the course required of all engineers, and they might cover other portions of the text in a second course for students in a particular branch of engineering. Thus civil, environmental, and agricultural engineers would emphasize Chapter 10 and perhaps Chapter 12 in a second course, while mechanical engineers would probably include Chapters 9 and 13 in a second course. A number of schools have used the book for courses in hydraulic machinery.

For instructors only, a companion Solutions Manual is available from McGraw-Hill that contains typed and carefully explained solutions to all the exercises and end-of-chapter problems in the book; for convenience, the problem statements and problem figures are repeated with the solutions. The manual contains suggestions on how to use it most effectively to select problems for assignment, and a Problem Selection Guide for each chapter categorizes the problems by their difficulty, length, units used, and any special features.

Acknowledgments

We appreciate the many comments and suggestions that we have received from users of the book throughout the years, and from numerous anonymous indepth reviews arranged by McGraw-Hill. In particular, we thank the following reviewers for this tenth edition: Kenneth Edwards, Ohio University; Joel Melville, Auburn University; A. R. Rao, Purdue University; Henry Santeford,

Michigan Technological University; Yiannis Ventikos, Georgia Institute of Technology; Vaughan Voller, University of Minnesota; and Mark Widdowson, Virginia Polytechnic Institute and State University. They have all influenced the content and mode of presentation of the material. Further comments and suggestions for future editions of the book are always welcome.

We are very grateful for the care, assistance, and guidance that many people at McGraw-Hill and its subcontractors have provided to us in the preparation of this edition. Particularly, we appreciate the startup support that our developmental editor, Eric Munson, gave us, and the unflagging cooperation and patience of our production manager, Gloria Schiesl.

E. John Finnemore
Joseph B. Franzini

List of Symbols

The following table lists the letter symbols generally used throughout the text. Because there are so many more concepts than there are English and suitable Greek letters, certain conflicts are unavoidable. However, where we have used the same letter for different concepts, the topics are so far removed from each other that no confusion should result. Occasionally we will use a particular letter in one special case only, but we will clearly indicate this local deviation from the table, and will not use it elsewhere. We give the customary units of measurement for each item in the British Gravitational (BG) system, with the corresponding SI unit in parentheses or brackets.

For the most part, we have attempted to adhere to generally accepted symbols, but not always.

A = any area, ft^2 (m^2)
= cross-sectional area of a stream normal to the velocity, ft^2 (m^2)
= area in turbines or pumps normal to the direction of the absolute velocity of the fluid, ft^2 (m^2)
A_c = circumferential flow area, ft^2 (m^2)
A_s = area of a liquid surface as in a tank or reservoir, ft^2 or acre (m^2 or hectare)
a = area in turbines or pumps normal to the relative velocity of the fluid, ft^2 (m^2)
= linear acceleration, ft/sec^2 (m/s^2)
B = any width, ft (m)
= width of open channel at water surface, ft (m)
= width of turbine runner or pump impeller at periphery, ft (m)
b = bottom width of open channel, ft (m)
$\mathbf{C}$ = cavitation number = $(p - p_v)/(\frac{1}{2}\rho V^2)$ [dimensionless]
C = any coefficient [dimensionless]
= Chézy coefficient [$ft^{1/2}sec^{-1}$ ($m^{1/2}s^{-1}$)]
C_c = coefficient of contraction
C_d = coefficient of discharge
C_v = coefficient of velocity
} for orifices, tubes, and nozzles [all dimensionless]
C_D = drag coefficient [dimensionless]
C_f = average friction-drag coefficient for total surface [dimensionless]
C_{HW} = Hazen-Williams pipe roughness coefficient, $ft^{0.37}$/sec ($m^{0.37}$/s)
C_L = lift coefficient [dimensionless]
C_p = pressure coefficient = $\Delta p/(\frac{1}{2}\rho V^2)$ [dimensionless]
c = specific heat of liquid, Btu/(slug·°R) [cal/(g·K) or N·m/(kg·K)]
= wave velocity (celerity), fps (m/s)
= sonic (i.e., acoustic) velocity (celerity), fps (m/s)

c_f = local friction-drag coefficient [dimensionless]
c_j = velocity (celerity) of pressure wave in elastic fluid inside an elastic pipe, ft/sec (m/s)
c_p = specific heat of gas at constant pressure, ft·lb/(slug·°R) [N·m/(kg·K)]
c_v = specific heat of gas at constant volume, ft·lb/(slug·°R) [N·m/(kg·K)]
D = diameter of pipe, turbine runner, or pump impeller, ft or in (m or mm)
$D''V$ = product of pipe diameter in inches and mean flow velocity in fps
$\mathbf{E}$ = Euler number = $V/\sqrt{2\Delta p/\rho}$ [dimensionless]
E = specific energy in open channels = $y + V^2/2g$, ft (m)
= linear modulus of elasticity, psi (N/m^2)
E_j = "joint" volume modulus of elasticity for elastic fluid in an elastic pipe, psi (N/m^2)
E_v = volume modulus of elasticity, psi (N/m^2)
e = height of surface roughness projections, ft (mm)
= 2.71828182846 . . .
$\mathbf{F}$ = Froude number = $V/\sqrt{gL}$ [dimensionless]
F = any force, lb (N)
F_D = drag force, lb (N)
F_L = lift force, lb (N)
f = friction factor for pipe flow [dimensionless]
G = weight flow rate = $dW/dt = \dot{m}g = \gamma Q$, lb/sec (N/s)
g = acceleration due to gravity = 32.1740 ft/sec^2 (9.80665 m/s^2) (standard)
= 32.2 ft/sec^2 (9.81 m/s^2) for usual computation
H = total energy head = $p/\gamma + z + V^2/2g$, ft (m)
= head on weir or spillway, ft (m)
h = any head, ft (m)
= enthalpy (energy) per unit mass of gas = $i + p/\rho$, ft·lb/slug (N·m/kg)
h' = minor head loss, ft (m)
h_a = accelerative head = $(L/g)(dV/dt)$, ft (m)
h_c = depth to centroid of area, ft (m)
h_f = head loss due to wall or pipe friction, ft (m)
h_L = total head loss due to all causes, ft (m)
h_M = energy *added* to a flow by a machine per unit weight of flowing fluid, ft·lb/lb (N·m/N)
h_O = stagnation (or total) enthalpy of a gas = $h + \frac{1}{2}V^2$, ft·lb/slug (N·m/kg)
h_p = depth to center of pressure, ft (m)
= head added to a flow by a pump, ft (m)
h_t = head removed from a flow by a turbine, ft (m)
I = moment of inertia of area, ft^4 or in^4 (m^4 or mm^4)
= internal thermal energy per unit weight = i/g, ft·lb/lb (N·m/N)

I_c = moment of inertia about centroidal axis, ft^4 or in^4 (m^4 or mm^4)
i = internal thermal energy per unit mass = gI, ft·lb/slug (N·m/kg)
K = any constant [dimensionless]
k = any loss coefficient [dimensionless]
= specific heat ratio = c_p/c_v [dimensionless]
L = length, ft (m)
L_r = $1/\lambda$ = scale ratio = L_p/L_m [dimensionless]
ℓ = mixing length, ft or in (m or mm)
$\mathbf{M}$ = Mach number = V/c [dimensionless]
M = molar mass, slugs/slug-mol (kg/kg-mol)
m = mass = W/g, slugs (kg)
$\dot{m}$ = mass flow rate = $dm/dt = \rho Q$, slugs/sec (kg/s)
$\mathbf{N}$ = any dimensionless number
N_s = specific speed = $n_e\sqrt{\text{gpm}}/h^{3/4}$ for pumps
= specific speed = $n_e\sqrt{\text{bhp}}/h^{5/4}$ for turbines } [dimensionless]
NPSH = net positive suction head, ft (m)
n = an exponent or any number in general
= Manning coefficient of roughness, sec/ft$^{1/3}$ (s/m$^{1/3}$)
= revolutions per minute, min^{-1}
n_e = rotative speed of hydraulic machine at maximum efficiency, rev/min
P = power, ft·lb/sec (N·m/s)
= height of weir or spillway crest above channel bottom, ft (m)
= wetted perimeter, ft (m)
p = fluid pressure, lb/ft^2 or psi (N/m^2 = Pa)
p_{atm} = atmospheric pressure, psia (N/m^2 abs)
p_b = back pressure in gas flow, psf or psi (Pa)
p_O = stagnation pressure, psf or psi (Pa)
p_v = vapor pressure, psia (N/m^2 abs)
Q = volume rate of flow (discharge rate), cfs (m^3/s)
Q_H = heat *added* to a flow per unit weight of fluid, ft·lb/lb (N·m/N)
q = volume rate of flow per unit width of rectangular channel, cfs/ft = ft^2/sec (m^2/s)
q_H = heat transferred per unit mass of fluid, ft·lb/slug (N·m/kg)
$\mathbf{R}$ = Reynolds number = $LV\rho/\mu = LV/\nu$ [dimensionless]
R = gas constant, ft·lb/(slug·°R) or N·m/(kg·K)
R_h = hydraulic radius = A/P, ft (m)
R_m = manometer reading, ft or in (m or mm)
R_0 = universal gas constant = 49,709 ft·lb/(slug-mol·°R) [8312 N·m/(kg-mol·K)]
r = any radius, ft or in (m or mm)
r_0 = radius of pipe, ft or in (m or mm)
S = slope of energy grade line = h_L/L
S_c = critical slope of open channel flow
S_0 = slope of channel bed
S_w = slope of water surface } [dimensionless]

s = specific gravity of a fluid = ratio of its density to that of a standard fluid (water, air, or hydrogen) [dimensionless]
T = temperature, °F or °R (°C or K)
= period of time for travel of a pressure wave, sec (s)
= torque, ft·lb (N·m)
T_O = stagnation temperature of a gas = $T + \frac{1}{2}V^2/c_p$, °F or °R (°C or K)
T_r = travel time (pulse interval) of a pressure wave, sec (s)
t = time, sec (s)
= thickness, ft or in (m or mm)
t_c = time for complete or partial closure of a valve, sec (s)
U, U_0 = uniform velocity of fluid, fps (m/s)
u = velocity of a solid body, fps (m/s)
= tangential velocity of a point on a rotating body = $r\omega$, fps (m/s)
= local velocity of fluid, fps (m/s)
u' = turbulent velocity fluctuation in the direction of flow, fps (m/s)
u_* = shear stress velocity or friction velocity = $\sqrt{\tau_0/\rho}$, ft/sec (m/s)
V = mean velocity of fluid, fps (m/s)
= absolute velocity of fluid in hydraulic machines, fps (m/s)
V_c = critical mean velocity of open channel flow, fps (m/s)
V_j = jet velocity, fps (m/s)
V_m = meridional velocity, fps (m/s)
V_r = radial component of velocity = $V\sin\alpha = v\sin\beta$, fps (m/s)
V_u = tangential component of velocity = $V\cos\alpha = u + v\cos\beta$, fps (m/s)
$\forall$ = any volume, ft^3 (m^3)
v = relative velocity of fluid in hydraulic machines, fps (m/s)
= specific volume = $1/\rho$, ft^3/slug (m^3/kg)
v_r = radial component of relative velocity = $v\sin\beta$, fps (m/s)
v_u = tangential component of relative velocity = $v\cos\beta$, fps (m/s)
v' = turbulent velocity fluctuation normal to the direction of flow, fps (m/s)
u, v, w = components of velocity in x, y, z, directions, fps (m/s)
W = Weber number = $V/\sqrt{\sigma/\rho L}$ [dimensionless]
W = total weight, lb (N)
x = a distance, usually parallel to flow, ft (m)
x_c = distance from leading edge to point where boundary layer becomes turbulent, ft (m)
y = a distance along a plane in hydrostatics, ft (m)
= total depth of open channel flow, ft (m)
y_c = critical depth of open channel flow, ft (m)
= distance to centroid, ft (m)
y_h = hydraulic (mean) depth = A/B, ft (m)
y_0 = depth for uniform flow in open channel (normal depth), ft (m)
y_p = distance to center of pressure, ft (m)
z = elevation above any arbitrary datum plane, ft (m)

α (alpha) = an angle; between V and u in rotating machinery, measured between their positive directions
= kinetic energy correction factor [dimensionless]
β (beta) = an angle; between v and u in rotating machinery, measured between their positive directions
= momentum correction factor [dimensionless]
Γ (gamma) = circulation, ft^2/sec (m^2/s)
γ (gamma) = specific weight, lb/ft^3 (N/m^3)
δ (delta) = thickness of boundary layer, in (mm)
δ_v = thickness of viscous sublayer in turbulent flow, in (mm)
δ_t = thickness of transition boundary layer in turbulent flow, in (mm)
ε (epsilon) = kinematic eddy viscosity, ft^2/sec (m^2/s)
η (eta) = eddy viscosity, lb·sec/ft^2 (N·s/m^2)
= efficiency of hydraulic machine
θ (theta) = any angle
λ (lambda) = model ratio or model scale = 1/(scale ratio) = L_m/L_p [dimensionless]
μ (mu) = absolute or dynamic viscosity, lb·sec/ft^2 (N·s/m^2)
ν (nu) = kinematic viscosity = μ/ρ, ft^2/sec (m^2/s)
ξ (xi) = vorticity, sec^{-1} (s^{-1})
Π (pi) = dimensionless parameter
π = 3.14159265359. . .
ρ (rho) = density, mass per unit volume = γ/g, slug/ft^3 (kg/m^3)
ρ_O = stagnation density of a gas, slug/ft^3 (kg/m^3)
Σ (sigma) = summation
σ (sigma) = surface tension, lb/ft (N/m)
= cavitation parameter in turbomachines [dimensionless]
= submergence of weir = h_d/h_u [dimensionless]
σ_c = critical cavitation parameter in turbomachines [dimensionless]
τ (tau) = shear stress, lb/ft^2 (N/m^2)
τ_0 = shear stress at a wall or boundary, lb/ft^2 (N/m^2)
ϕ (phi) = any function
= velocity potential, ft^2/sec (m^2/s) for two-dimensional flow
= peripheral-velocity factor = $u_{\text{periph}}/\sqrt{2gh}$ [dimensionless]
ϕ_e = peripheral-velocity factor at point of maximum efficiency [dimensionless]
ψ (psi) = stream function, ft^2/sec (m^2/s) for two-dimensional flow
ω (omega) = angular velocity = $u/r = 2\pi n/60$, rad/sec (rad/s)

Values at specific points will be indicated by suitable subscripts. In the use of subscripts 1 and 2, the fluid is always assumed to flow from 1 to 2.

List of Abbreviations

abs = absolute
atm = atmospheric, atmospheres
avg = average
bhp = brake (or shaft) horsepower
Btu = British Thermal Unit
°C = degree celsius
cal = calorie
cfm = cubic feet per minute
cfs = cubic feet per second
cm = centimeter
d = day or days (SI)
°F = degree fahrenheit
fpm = feet per minute
fps = feet per second
ft = foot or feet
g = gram or grams
gal = gallon
gpd = (U.S.) gallons per day
gpm = (U.S.) gallons per minute
h = hour or hours (SI)
ha = hectare
hp = horsepower
hr = hour or hours (BG)
Hz = hertz (cycles per second)
in = inch or inches
J = joules = N·m = W·s
K = kelvin (unit of temperature)
kg = kilograms = 10^3 grams
kgf = kilogram force
kgm = kilogram mass
km = kilometer
L = liter
lb = pounds of force (*not* lbs)
lbf = pound force
lbm = pound mass
ln = $\log_e$
log = $\log_{10}$
m = meter or meters
mb = millibars = 10^{-3} bar
mb abs = millibars, absolute
mgd = million (U.S.) gallons per day
min = minute or minutes (BG and SI)
mL = milliliter
mm = millimeters = 10^{-3} meter
mol = mole
mph = miles per hour
N = newton or newtons
= $kg \cdot m/s^2$
N/m^2 abs = newtons per square meter, absolute
oz = ounce
P = poise = 0.10 $N \cdot s/m^2$
Pa = pascal = N/m^2
pcf = pounds per cubic foot
psf = pounds per square foot
psfa = pounds per square foot, absolute
psfg = pounds per square foot, gage
psi = pounds per square inch
psia = pounds per square inch, absolute
psig = pounds per square inch, gage
°R = degree rankine
rev = revolutions
rpm = revolutions per minute
rps = revolutions per second
s = second or seconds (SI)
sec = second or seconds (BG)
St = stoke = cm^2/s
W = watt or watts = J/s
y = year or years (SI)
yr = year or years (BG)

Chapter 1
Introduction

In preparing this tenth edition of *Fluid Mechanics with Engineering Applications* we have strived to present the material in such a way that you, the student, can readily learn the fundamentals of fluid mechanics and see how those fundamentals can be applied to practical engineering problems. Only by understanding the text and working many problems can students master the application of the fundamentals.

1.1 Scope of Fluid Mechanics

Undoubtedly you have observed the movement of clouds in the atmosphere, the flight of birds through the air, the flow of water in streams, and the breaking of waves at the seashore. Fluid mechanics phenomena are involved in all of these. Fluids include gases and liquids, with air and water as the most prevalent. Some of the many other aspects of our lives that involve fluid mechanics are flow in pipelines and channels, movements of air and blood in the body, air resistance or drag, wind loading on buildings, motion of projectiles, jets, shock waves, lubrication, combustion, irrigation, sedimentation, and meteorology and oceanography. The motions of moisture through soils and oil through geologic formations are other applications. A knowledge of fluid mechanics is required to properly design water supply systems, wastewater treatment facilities, dam spillways, valves, flow meters, hydraulic shock absorbers and brakes, automatic transmissions, aircraft, ships, submarines, breakwaters, marinas, rockets, computer disk drives, windmills, turbines, pumps, heating and air conditioning systems, bearings, artificial organs, and even sports items like golf balls, yachts, race cars, and hang gliders. It is clear that everybody's life is affected by fluid mechanics in a variety of ways. All engineers should have at least a basic knowledge of fluid phenomena.

Fluid mechanics is the science of the mechanics of liquids and gases, and is based on the same fundamental principles that are employed in the mechanics of solids. The mechanics of fluids is a more complicated subject than the mechanics of solids, however, because with solids one deals with separate and tangible elements, while with fluids there are no separate elements to be distinguished.

Fluid mechanics can be divided into three branches: ***fluid statics*** is the study of the mechanics of fluids at rest; ***kinematics*** deals with velocities and streamlines without considering forces or energy; and ***fluid dynamics*** is concerned with the relations between velocities and accelerations and the forces exerted by or upon fluids in motion.

Classical ***hydrodynamics*** is largely a subject in mathematics, since it deals with an imaginary ideal fluid that is completely frictionless. The results of such studies, without consideration of all the properties of real fluids, are of limited practical value. Consequently, in the past, engineers turned to experiments, and from these developed empirical formulas that supplied answers to practical problems. When dealing with liquids, this subject is called ***hydraulics.***

Empirical hydraulics was confined largely to water and was limited in scope. With developments in aeronautics, chemical engineering, and the petroleum industry, the need arose for a broader treatment. This has led to the combining of classical hydrodynamics (ideal fluids) with the study of real fluids, both liquids (hydraulics) and gases, and this combination we call ***fluid mechanics.*** In modern fluid mechanics the basic principles of hydrodynamics are combined with experimental data. The experimental data can be used to verify theory or to provide information supplementary to mathematical analysis. The end product is a unified body of basic principles of fluid mechanics that we can apply to the solution of fluid-flow problems of engineering significance. With the advent of the computer, during the past 25 years the entirely new field of ***computational fluid dynamics*** has developed. Various numerical methods such as finite differences, finite elements, boundary elements, and analytic elements are now used to solve advanced problems in fluid mechanics.

1.2 HISTORICAL SKETCH OF THE DEVELOPMENT OF FLUID MECHANICS[1]

From time to time we discover more about the knowledge that ancient civilizations had about fluids, particularly in the areas of irrigation channels and sailing ships. The Romans are well known for their aqueducts and baths, many of which were built in the fourth century B.C., with some still operating today. The Greeks are known to have made quantified measurements, the best known being those of Archimedes who discovered and formulated the principles of buoyancy in the third century B.C.

We know of no basic improvements to the understanding of flow until Leonardo da Vinci (1452–1519), who performed experiments, investigated, and speculated on waves and jets, eddies and streamlining, and even on flying. He contributed to the one-dimensional equation for conservation of mass.

Isaac Newton (1642–1727), by formulating his laws of motion and his law of viscosity, in addition to developing the calculus, paved the way for many great developments in fluid mechanics. Using Newton's laws of motion, numerous 18th-century mathematicians solved many frictionless (zero-viscosity) flow

[1] See also Rouse, H., and S. Ince, *History of Hydraulics*, Dover, New York, 1963.

problems. However, most flows are dominated by viscous effects, so engineers of the 17th and 18th centuries found the inviscid flow solutions unsuitable, and by experimentation they developed empirical equations, thus establishing the science of hydraulics.

Late in the 19th century the importance of dimensionless numbers and their relationship to turbulence was recognized, and dimensional analysis was born. In 1904 Ludwig Prandtl published a key paper, proposing that the flow fields of low-viscosity fluids be divided into two zones, namely a thin, viscosity-dominated ***boundary layer*** near solid surfaces, and an effectively inviscid outer zone away from the boundaries. This concept explained many former paradoxes, and enabled subsequent engineers to analyze far more complex flows. However, we still have no complete theory for the nature of turbulence, and so modern fluid mechanics continues to be a combination of experimental results and theory.

1.3 THE BOOK, ITS CONTENTS, AND HOW TO BEST STUDY FLUID MECHANICS

In this introductory chapter we attempt to give you some insight into what fluid mechanics is all about. In the previous sections we discussed the scope of fluid mechanics and the historical development of the subject. This and the next section explain how to best use this book to study fluid mechanics. The last section of this chapter discusses the importance of dimensions and units.

You can get a feel for the contents of the book and the variety of topics it covers by reviewing the Contents at the front of the text. Most of the subject titles are self-explanatory. The powerful analytical techniques of similitude and dimensional analysis build on the knowledge of dimensions to be reviewed in Sec. 1.5.

Because problem solving is such an important part of the study of fluid mechanics, before beginning you should make yourself very familiar with the supporting resources available. You will often be expected to know where to find such information, without any direct reference. For convenience, many unit conversions and related data have been collected on the inside covers of the book and the facing pages. Many data on material properties, often needed, are collected into the figures and tables of Appendix A; but some are also in the chapters, such as Fig. 2.1. The lists of symbols and abbreviations preceding this chapter are also a useful resource. Appendix B summarizes important information about equations, which form a key part of the language of fluid mechanics. As you progress you will increasingly realize how helpful programming procedures can be in solving many fluid mechanics problems, such as flow in pipes and pipe networks, water surface profiles in open channels and culverts, and unsteady flow problems. The most convenient of these procedures are in mathematics software packages such as Mathcad, in spreadsheets like Excel, and in equation solvers on programmable scientific calculators; use of these is described particularly in Chaps. 8 and 10, with some example applications in Appendix D. The major different types of programming procedures available are described in Appendix C, and problems for which these might be helpful are indicated by a preceding the problem number. Note, however,

that in all instances the use of programming and computers is optional. To help you broaden your horizons by reading books on subjects related to those in this text, a list of such references is provided in Appendix E.

Throughout the book we strive to develop basic concepts in a logical manner so that you can readily read the material and understand it. Material is divided into "building blocks" within separate sections of the chapters. Once the basic concepts are developed, we often provide sample problems to illustrate applications of the concepts; then we usually provide exercises, which you should perform as needed to reinforce your understanding. The exercises normally address only material in the preceding section, and are generally quite straightforward. They are drill exercises, to familiarize you with the subject and concepts, and answers to the exercises are provided at the back of the book (Appendix F).

At the end of each chapter we have placed summary problems. These are intended to be more like real-world or examination problems, where it is not indicated which section(s) they address. In some instances they may require the application of concepts from a number of sections or even chapters. You will find it a great advantage to have developed your familiarity with the concepts by doing drill exercises before tackling the end-of-chapter problems. Although answers to the exercises are given, answers to the end-of-chapter problems are not. One reason is that many problems in fluid mechanics require trial-and-error solution methods, and having answers reduces learning of such methods. Another is that as you progress in competence, you need to rely on yourself more and learn ways to check yourself; real-world problems do not come with accompanying answers.

As we stressed at the outset, there are two major keys to success in mastering fluid mechanics. The first of these is learning the fundamentals, and this requires that you read and understand the text. There are many phenomena and situations that must be described in words and figures, and that equations and numbers alone cannot explain. So be sure you adequately read (and reread) the text.

The second key to success is working many problems. In this text we stress the application of basic principles to the solution of practical engineering problems. *Only by working many problems can you truly understand the basic principles and how to apply them.* We feel this is very important! Because of this importance, we next include some suggestions that will aid you in solving problems.

1.4 APPROACH TO PROBLEM SOLVING

The following are four important steps to becoming a master of the assigned material:

1. Study the material of the section(s) to be covered next *before* it is covered in class. This way you will get so much more out of the review in class; also, you will be able to ask (and answer) perceptive questions!
2. Study the sample problems, and be sure you can work them yourself *without referring to the text,* i.e., "closed book."

3. Do enough of the drill Exercises, answers unseen, until you are confident of your familiarity with the basic material and procedures.
4. Do the (more challenging) homework Problems you have been assigned.

If you get stuck on any of these steps, this suggests you have not sufficiently mastered the previous step(s). Review those yourself before seeking help. Mastering the material by yourself will build your self-confidence, but of course you should always seek help if unable to master it alone.

In writing solutions to problems, for steps 3 and 4 these substeps are often recommended, and are an excellent guide:

a. Thoroughly read and ponder the problem statement for a few moments before writing anything on paper. Identify the simplest approach.
b. Summarize information to be used, both that given and that obtained elsewhere; and summarize quantities to be found. Define any unusual symbols.
c. Draw a neat figure or figures, fully labeled, of the situation to be analyzed.
d. State all assumptions you consider necessary.
e. Reference all principles, equations, tables, etc., that you will use. (Remember all the available supporting resources, mentioned in Sec. 1.3.)
f. Solve the problem as far as possible algebraically (in terms of symbols) before inserting numbers.
g. Check the dimensions of the various terms for consistency (per Sec. 1.5).
h. Insert numerical values[2] for the variables at the last possible stage, using a consistent (SI or BG) set of units (per Sec. 1.5). Evaluate a numerical answer, with units, and report it to an appropriate precision. (This should be no more precise, as a percentage, than that of the least precise inserted value; and however precise the inserted values may be, a common practical rule in engineering is to report results to three significant figures, or four figures if they begin with a "1," which yields a maximum error of 0.5%.) Do not round off values in your calculator, only do so when presenting your answer.
i. Check your answer for reasonableness and accuracy by comparing it with expected results and by whatever other methods you can devise.
j. Check that any assumptions you made initially are satisfied or appropriate. Note any limitations that apply.

We suggest you do not attempt more advanced problems until you have mastered the less advanced ones. *Demonstrate* this mastery to yourself by achieving correct answers without referring to the text, i.e., "closed book."

To confirm that you have sufficiently mastered problem solving, practice working problems closed book with a time limit. This can be quite challenging, so doing this regularly can be helpful. You can reserve one or two of the homework problems in each assignment for this purpose.

[2] Given values (only) are to be assumed fully accurate, regardless of the number of significant figures.

Form a study group early on in the course with one or more study partners. It is very time effective to quiz one another about the categories that problems fall into, and about the procedures that should be used to solve them (without always doing all the calculations).

Not only do you need to learn and understand the material, but also you need to *know how and when to use it!* Seek and build understanding of *applications* for your knowledge, particularly to problems that are not straightforward. It is for non-straightforward problems that we need well-trained engineers.

Understanding is particularly demonstrated by successful application of the principles to situations different from those you have met before. So getting the correct answers to a few "plug and chug" exercises does not alone indicate understanding. Also you should know that *feeling* you are prepared is not reliable. You should *prove* it to yourself by (a) correctly solving problems closed book under a time limit and (b) by correctly answering questions on the material.

Although the preceding emphasizes analysis, which can involve algebra, trial-and-error methods, graphical methods, and calculus, other problem-solving methods such as computer and experimental techniques can be used, and should be mastered to a reasonable extent. Become familiar with the use of computers to solve problems by iterative procedures, to perform repetitive numerical evaluations, to perform numerical integration, etc. Also, programmable calculators are becoming very powerful, with root finders to solve implicit equations and with many integration and graphing capabilities. Familiarity with these will greatly add to your effectiveness in fluid mechanics and as an engineer in general. Chapter 7 provides guidance on planning flow experiments and model tests. Take every opportunity to learn about practical issues in the laboratory and on field trips; never forget, as the title of this book reminds us, that all this theory and analysis is for application to the real world.

Problems in the real world of course are usually not like those in our textbooks. So next you will need to develop your abilities to *recognize* problems in our environment, and to clearly *define* (or formulate) them, before beginning any analysis. Often you will find that various methods of solution can be used, and experience will help you select the most appropriate. In the real world the numerical results of analyzing a problem are not the ultimate goal; for those results then need to be interpreted in terms of the physical problem, and then recommendations need to be made for action.

Remain conscious of your goal, to become a capable and responsible engineer, and remain conscious of your path to that goal, which involves the many steps we have outlined here.

1.5 DIMENSIONS AND UNITS

To properly define a physical property or a fluid phenomenon, one must express the property or phenomenon in terms of some set of units. For example, the diameter of a pipe might be 160 millimeters and the average flow velocity 8 meters per second.[3] A different set of units might have been used, such as a diameter of

[3] This book uses the American spelling *meter,* although the official spelling is *metre.*

0.16 meter and a velocity of 800 centimeters per second. Or, the diameter and velocity might have been expressed in English (U.S. Customary) or other units. In this book we use two systems of units: the British Gravitational (BG) system when dealing with English units, and the SI (Système Internationale d'Unités) when dealing with metric units. The SI was adopted in 1960 at the Eleventh General International Conference on Weights and Measures, at which the United States was represented. As of the year 2000, nearly every major country in the world, except the United States, was using the SI; it appears likely that the United States will officialy adopt the SI within a few years. Because of the imminence of metrification in the United States, the need to be able to readily interact with the many users of SI units, and because English units have been used in the technical literature for so many years, it is essential that the engineer be familiar with *both* the systems, BG and SI, used in this book.

In fluid mechanics the basic dimensions are length (L), mass (M), time (T), force (F), and temperature (θ). In order to satisfy Newton's second law, $F = ma = MLT^{-2}$, where acceleration a is expressed by its basic dimensions as LT^{-2}, we note that units for only three of the first four of these dimensions can be assigned arbitrarily; the fourth unit must agree with the other three, and is therefore known as a ***derived unit***. In the two systems of units used in this book, the commonly used units for the five basic dimensions mentioned are:

Dimension	BG unit	SI unit
Length (L)	Foot (ft)	Meter (m)
Mass (M)	Slug (= lb·sec^2/ft)	Kilogram (kg)
Time (T)	Second (sec)	Second (s)
Force (F)	Pound (lb)	Newton (N) (= kg·m/s^2)
Temperature (θ)		
Absolute	Rankine (°R)	Kelvin (K)
Ordinary	Fahrenheit (°F)	Celsius (°C)

Derived units in blue.

The SI employs L, M, and T and derives F from MLT^{-2}. Force in the SI is defined by the ***newton,*** the force required to accelerate one kilogram of mass at a rate of one meter per second per second; that is,

$$1\ \text{N} = (1\ \text{kg})(1\ \text{m/s}^2)$$

On the other hand, the British Gravitational system, also sometimes known as the U.S. Customary (USC) system, employs L, F, and T, and derives M from $F/a = FL^{-1}T^2$. The BG unit of mass, the ***slug,*** is therefore defined as that mass that accelerates at one foot per second per second when acted upon by a force of one pound; that is,

$$1\ \text{slug} = (1\ \text{lb})/(1\ \text{ft/sec}^2) = 1\ \text{lb·sec}^2/\text{ft}$$

or

$$1\ \text{lb} = (1\ \text{slug})(1\ \text{ft/sec}^2)$$

When working in the BG system, it often pays to keep mass expressed in basic units (lb·sec^2/ft or weight/gravitational acceleration) for as long as possible.

We see that the definition of mass in the BG system depends on the definition of one ***pound,*** which is the force of gravity acting on (or weight of) a platinum standard whose mass is 0.453 592 43 kg. ***Weight*** is the gravitational attraction force F between two bodies, of masses m_1 and m_2, given by Newton's Law of Gravitation as

$$F = G\frac{m_1 m_2}{r^2}$$

where G is the universal constant of gravitation and r is the distance between the centers of the two masses. If m_1 is the mass m of an object on the earth's surface and m_2 is the mass M of the earth then r is the radius of the earth, so that

$$F = m\left(\frac{GM}{r^2}\right)$$

and the weight of the object is

$$W = mg$$

where the gravitational acceleration $g = GM/r^2$. Clearly g varies slightly with altitude and latitude on earth, since the earth is not truly spherical, while in space and on other planets it is much different. Furthermore, the preceding does not take into account the earth's rotation, which by centrifugal action reduces the apparent weight of an object by at most 0.35% at the equator. Because the force (weight) depends on the value of g, which in turn varies with location, a system such as the BG system based on length (L), force (F), and time (T) is referred to as a ***gravitational system.*** On the other hand, systems like the SI, which are based on length (L), mass (M), and time (T), are ***absolute*** because they are *independent* of the gravitational acceleration g.

A partial list of derived quantities encountered in fluid mechanics and their commonly used dimensions in terms of L, M, T, and F is:

Quantity	Commonly used dimensions	BG unit	SI unit
Acceleration (a)	LT^{-2}	ft/sec^2	m/s^2
Area (A)	L^2	ft^2	m^2
Density (ρ)	ML^{-3}	slug/ft^3	kg/m^3
Energy, work or quantity of heat	FL	ft·lb	N·m = J
Flowrate (Q)	L^3T^{-1}	ft^3/sec (cfs)	m^3/s
Frequency	T^{-1}	cycle/sec (sec^{-1})	Hz (hertz, s^{-1})
Kinematic viscosity (ν)	L^2T^{-1}	ft^2/sec	m^2/s
Power	FLT^{-1}	ft·lb/sec	N·m/s = W
Pressure (p)	FL^{-2}	lb/in^2 (psi)	N/m^2 = Pa
Specific weight (γ)	FL^{-3}	lb/ft^3 (pcf)	N/m^3
Velocity (V)	LT^{-1}	ft/sec (fps)	m/s
Viscosity (μ)	FTL^{-2}	lb·sec/ft^2	N·s/m^2
Volume ($\forall$)	L^3	ft^3	m^3

UNIVERSIDAD POLITECNICA SALESIANA
BIBLIOTECA

Using the identity $F = MLT^{-2}$, all dimensions containing an F could have been expressed using an M instead, and vice versa. Other derived quantities will be dealt with when they are encountered in the text, and particularly in Chap. 7. Radians do not have dimensions, because they are defined as an arc length divided by a radius.

On the earth's surface the variation in g is small, and, by international agreement, standard gravitational acceleration at sea level is 32.1740 ft/sec^2 or 9.806 65 m/s^2 (for problem solving we usually use 32.2 ft/sec^2 or 9.81 m/s^2). So variations in g are generally not considered in this text as long as we are analyzing problems on the earth's surface. Fluid problems for other locations, such as on the moon, where g is quite different from that on earth, can be handled by the methods presented in this text if proper consideration is given to the value of g.

For unit mass (1 slug or 1 kg) on the earth's surface, we note that

In BG units: $W = mg = (1\text{ slug})(32.2\text{ ft/sec}^2) = 32.2\text{ lb}$;
In SI units: $W = mg = (1\text{ kg})(9.81\text{ m/s}^2) = 9.81\text{ N}$.

Other systems of units used elsewhere include the English Engineering (EE) system, the Absolute Metric (cgs) system, and the mks metric system. The EE system uses pound force (lbf) and pound mass (lbm), and the mks metric system uses kilogram force (kgf) and kilogram mass (kgm). As a result, both of these are said to be ***inconsistent systems,*** because unit force does not cause unit mass to undergo unit acceleration; they require an additional proportionality constant or conversion factor. The SI and BG systems used in this book are ***consistent systems*** having conversion factors with a magnitude of one. Although the cgs metric system is both consistent and nongravitational, it is little used for engineering applications because its unit of force, the dyne, is so small; 1 dyne = (1 g)(1 cm/s^2) = 10^{-5} N.

Do not be confused by popular usage of kilograms to measure weight (force). When European shoppers buy a kilo of sugar, say, in our terms they are buying sugar with a mass of 1 kg, in effect defining a force of 1 kg (1 kgf) = (1 kgm)(9.81 m/s^2), which is equivalent to 9.81 N. Because a 1-lb weight has a mass of about 0.4536 kg, the shoppers' conversion factor is 1.0/0.4536 = 2.205 lb/kgf. In engineering we are careful to distinguish between mass and weight, reserving kg for mass and using newtons for force in the SI system.

In this book we shall use the abbreviation kg for kilogram mass, and lb for pound force. The abbreviation lb for pound is taken from the Latin *libra,* plural *librae,* so the correct plural abbreviation is lb not lbs. The units second, minute, hour, day, and year are correctly abbreviated as s, min, h, d, and y in the SI system, and although in the BG system they should be abbreviated as sec, min, hr, day, and yr, it is common to use the SI abbreviations for both systems. There are many "nonstandard" or traditional abbreviations used by engineers, such as fps for ft/sec, gpm for gal/min, and cfs for ft^3/sec (also sometimes referred to as the second-foot and the cusec). The more common of these are included in the list just preceding this chapter. Acres, tons, and slugs are not abbreviated. When units are named after people, like the newton (N), joule (J), and pascal (Pa), they are capitalized when abbreviated but not capitalized when spelled out.

The abbreviation capital L for liter is a special case, used to avoid confusion with one (1). Also note that in the SI the unit for absolute temperature measurement is the degree kelvin, which is abbreviated K *without* a degree (°) symbol.

The British or imperial gallon is, within 0.1%, equal to 1.2 U.S. gallons. Where the kind of gallons is not specified, in this book assume them to be U.S. gallons.

When dealing with unusually large or very small numbers, a series of prefixes has been adopted for use with SI units. The most commonly used prefixes are given for convenient reference facing the inside front and back covers of this book. Hence Mg (megagram) represents 10^6 grams, mm (millimeter) represents 10^{-3} meters, and kN (kilonewton) represents 10^3 newtons, for example. Note that multiples of 10^3 are preferred in engineering usage; other multiples like cm are to be avoided if possible. Also, in the SI it is conventional to separate sequences of digits into groups of three by spaces rather than by commas, as was done earlier for the mass of the standard pound. Thus 10 cubic meters of water weigh 98 100 N, or 98.1 kN.

Often we need to convert quantities from BG units into SI units, and vice versa. Because time units are the same in both systems, we only need to convert units of length, and force or mass, from which all other units can then be derived. For length, by definition, one foot is *exactly* 0.3048 meters, and so an inch is *exactly* 25.4 mm. For force, using $W = mg$ and definitions given earlier, 1 lb = (0.453 592 43 kg)(9.806 65 m/s^2), or about 4.448 N. For mass, 1 slug = (1 lb)/(1 ft/sec^2) is about equal to (4.448 N)/(0.3048 m/s^2) = 14.59 kg. Conversion factors for many other units, derived from these three basic ones, are given for convenience in tables on the insides of the front cover (BG to SI) and back cover (SI to BG) of the book; *exact* conversion factors are indicated by an asterisk (*). These tables include conversions of units *within* the BG system and *within* the SI. On the facing pages we give some definitions, other useful conversions, and relations between the four principal temperature scales.

In the SI, lengths are commonly expressed in millimeters (mm), centimeters (cm; try to avoid), meters (m), or kilometers (km), depending on the distance being measured. A meter is about 39 inches or 3.3 ft and a kilometer is approximately five-eighths of a mile. Areas are usually expressed in square centimeters (cm^2), square meters (m^2), or ***hectares*** (100 m × 100 m = 10^4 m^2), depending on the area being measured. The hectare, used for measuring large areas, is equivalent to about 2.5 acres. A newton is equivalent to almost 0.225 lb. The SI unit of stress (or pressure), newton per square meter (N/m^2), is known as the ***pascal*** (Pa), and is equivalent to about 0.021 lb/ft^2 or 0.000 15 lb/in^2. In SI units energy, work, or quantity of heat are ordinarily expressed in joules (J). A ***joule***[4] is equal to a newton-meter, i.e., J = N·m. The unit of power is the ***watt*** (W), which is equivalent to a joule per second, i.e., W = J/s = N·m/s.

[4] Joule is pronounced (jo͞ol), to rhyme with cool.

When we have to work with less usual units, like centipoise (for viscosity) or ergs (for energy), it is best to convert them into SI or BG units as soon as possible.

SAMPLE PROBLEM 1.1 Bernoulli's equation for the flow of an ideal fluid, which is discussed in Chap. 5, can be written

$$\frac{p}{\gamma} + z + \frac{V^2}{2g} = \text{constant} \tag{5.7}$$

where p = pressure, γ = specific weight, z = elevation, V = mean flow velocity, and g = acceleration of gravity. Demonstrate that this equation is dimensionally homogeneous, i.e., that all terms have the same dimensions.

Solution

Term 1: Dimensions of $\dfrac{p}{\gamma} = \dfrac{F/L^2}{F/L^3} = L$

Term 2: Dimensions of $z = L$

Term 3: Dimensions of $\dfrac{V^2}{2g} = \dfrac{(L/T)^2}{L/T^2} = L$

So all the terms have the same dimensions, L, which must also be the dimensions of the constant at the right-hand side of Eq. (5.7).

SAMPLE PROBLEM 1.2 Convert 200 Btu to (*a*) BG, (*b*) SI, and (*c*) cgs metric units of energy.

Solution

From inside the front cover:

$$1\text{ Btu} = 778\text{ ft·lb}, \quad 1\text{ ft·lb} = 1.356\text{ N·m} = 1.356\text{ J}, \quad 1\text{ N} = 10^5\text{ dyne}.$$

(*a*) For BG units: $200\text{ Btu} = 200\text{ Btu}\left(\dfrac{778\text{ ft·lb}}{1\text{ Btu}}\right) = 155{,}600\text{ ft·lb}.$

(*b*) For SI units: $155{,}600\text{ ft·lb} = 155{,}600\text{ ft·lb}\left(\dfrac{1.356\text{ N·m}}{1\text{ ft·lb}}\right)$

$$= 210\,994\text{ N·m} = 211\text{ kN·m} = 211\text{ kJ}.$$

(*c*) For cgs units: $210\,994\text{ N·m} = 211 \times 10^3\text{ N·m}\left(\dfrac{10^5\text{ dyne}}{1\text{ N}}\right)\left(\dfrac{10^2\text{ cm}}{\text{m}}\right)$

$$= 211 \times 10^{10}\text{ dyne·cm} = 211 \times 10^{10}\text{ erg}.$$

EXERCISES

1.5.1 Demonstrate that Eq. (6.5) is dimensionally homogeneous.

1.5.2 Demonstrate that Eq. (10.57) is dimensionally homogeneous. Note that C_d is dimensionless.

1.5.3 Demonstrate that Eq. (11.2) is dimensionally homogeneous. Note that $\forall$ is a volume, h_f has the dimensions of length, and ν is kinematic viscosity.

1.5.4 Demonstrate that Eq. (12.4) is dimensionally homogeneous.

1.5.5 Demonstrate that Eq. (13.45) is dimensionally homogeneous. Note that k is dimensionless.

1.5.6 Using information from inside the cover of this book, determine the weight of a U.S. gallon of water in the following units: (*a*) pounds; (*b*) newtons; (*c*) dynes.

1.5.7 Using information from inside the cover of this book, determine the weight of one liter of water at 5°C in the following units: (*a*) pounds; (*b*) newtons; (*c*) dynes.

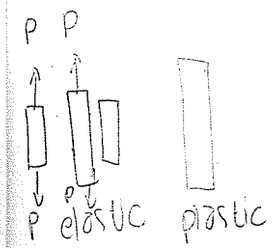
1.5.8 Using information from inside the cover of this book, convert 25 million U.S. gallons per day (mgd) into (*a*) BG and (*b*) SI units.

1.5.9 Using information from inside the cover of this book, convert 100 km/h into (*a*) SI and (*b*) BG units.

CHAPTER 2
Properties of Fluids

In this chapter we discuss a number of fundamental properties of fluids. An understanding of these properties is essential for us to apply basic principles of fluid mechanics to the solution of practical problems.

2.1 DISTINCTION BETWEEN A SOLID AND A FLUID



The molecules of a ***solid*** are usually closer together than those of a ***fluid.*** The attractive forces between the molecules of a solid are so large that a solid tends to retain its shape. This is not the case for a fluid, where the attractive forces between the molecules are smaller. An ideal elastic solid will deform under load and, once the load is removed, will return to its original state. Some solids are plastic. These deform under the action of a sufficient load and deformation continues as long as a load is applied, providing the material does not rupture. Deformation ceases when the load is removed, but the plastic solid does not return to its original state.

The intermolecular cohesive forces in a fluid are not great enough to hold the various elements of the fluid together. Hence a fluid will flow under the action of the slightest stress and flow will continue as long as the stress is present.

2.2 DISTINCTION BETWEEN A GAS AND A LIQUID



A fluid may be either a ***gas*** or a ***liquid.*** The molecules of a gas are much farther apart than those of a liquid. Hence a gas is very compressible, and when all external pressure is removed, it tends to expand indefinitely. A gas is therefore in equilibrium only when it is completely enclosed. A liquid is relatively incompressible, and if all pressure, except that of its own vapor pressure, is removed, the cohesion between molecules holds them together, so that the liquid does not expand indefinitely. Therefore a liquid may have a free surface, i.e., a surface from which all pressure is removed, except that of its own vapor.

A ***vapor*** is a gas whose temperature and pressure are such that it is very near the liquid phase. Thus steam is considered a vapor because its state is

normally not far from that of water. A gas may be defined as a highly superheated vapor; that is, its state is far removed from the liquid phase. Thus air is considered a gas because its state is normally very far from that of liquid air.

The volume of a gas or vapor is greatly affected by changes in pressure or temperature or both. It is usually necessary, therefore, to take account of changes in volume and temperature in dealing with gases or vapors. Whenever significant temperature or phase changes are involved in dealing with vapors and gases, the subject is largely dependent on heat phenomena (***thermodynamics***). Thus fluid mechanics and thermodynamics are interrelated.

2.3 DENSITY, SPECIFIC WEIGHT, SPECIFIC VOLUME, AND SPECIFIC GRAVITY

The ***density*** ρ (rho),[1] or more strictly, ***mass density,*** of a fluid is its *mass* per unit volume, while the ***specific weight*** γ (gamma) is its *weight* per unit volume. In the British Gravitational (BG) system (Sec. 1.5) density ρ will be in slugs per cubic foot (kg/m^3 in SI units), which can also be expressed as units of lb·sec^2/ft^4 (N·s^2/m^4 in SI units) (Sec. 1.5 and inside covers).

Specific weight γ represents the force exerted by gravity on a unit volume of fluid, and therefore must have the units of force per unit volume, such as pounds per cubic foot (N/m^3 in SI units).

Density and specific weight of a fluid are related as:

$$\rho = \frac{\gamma}{g} \quad \text{or} \quad \gamma = \rho g \tag{2.1}$$

Since the physical equations are dimensionally homogeneous, the dimensions of density are

$$\text{Dimensions of } \rho = \frac{\text{dimensions of } \gamma}{\text{dimensions of } g} = \frac{\text{lb/ft}^3}{\text{ft/sec}^2} = \frac{\text{lb·sec}^2}{\text{ft}^4} = \frac{\text{mass}}{\text{volume}} = \frac{\text{slugs}}{\text{ft}^3}$$

In SI units

$$\text{Dimensions of } \rho = \frac{\text{dimensions of } \gamma}{\text{dimensions of } g} = \frac{\text{N/m}^3}{\text{m/s}^2} = \frac{\text{N·s}^2}{\text{m}^4} = \frac{\text{mass}}{\text{volume}} = \frac{\text{kg}}{\text{m}^3}$$

Note that density ρ is absolute, since it depends on mass, which is independent of location. Specific weight γ, on the other hand, is not absolute, since it depends on the value of the gravitational acceleration g, which varies with location, primarily latitude and elevation above mean sea level.

Densities and specific weights of fluids vary with temperature. Appendix A provides commonly needed temperature variations of these quantities for water

[1] The names of Greek letters are given in the List of Symbols on page xix.

and air. It also contains densities and specific weights of common gases at standard atmospheric pressure and temperature. We shall discuss the specific weight of liquids further in Sec. 2.6.

Specific volume v is the volume occupied by a unit mass of fluid.[2] We commonly apply it to gases, and usually express it in cubic feet per slug (m^3/kg in SI units). Specific volume is the reciprocal of density. Thus

$$v = \frac{1}{\rho} \tag{2.2}$$

Specific gravity s of a liquid is the dimensionless ratio

$$s_{\text{liquid}} = \frac{\rho_{\text{liquid}}}{\rho_{\text{water at standard temperature}}}$$

Physicists use 4°C (39.2°F) as the standard, but engineers often use 60°F (15.56°C). In the metric system the density of water at 4°C is 1.00 g/cm^3 (or 1.00 g/mL),[3] equivalent to 1000 kg/m^3, and hence the specific gravity (which is dimensionless) of a liquid has the same numerical value as its density expressed in g/mL or Mg/m^3. Appendix A contains information on specific gravities and densities of various liquids at standard atmospheric pressure.

The specific gravity of a gas is the ratio of its density to that of either hydrogen or air at some specified temperature and pressure, but there is no general agreement on these standards, and so we must explicitly state them in any given case.

Since the density of a fluid varies with temperature, we must determine and specify specific gravities at particular temperatures.

SAMPLE PROBLEM 2.1 The specific weight of water at ordinary pressure and temperature is 62.4 lb/ft^3. The specific gravity of mercury is 13.56. Compute the density of water and the specific weight and density of mercury.

Solution

$$\rho_{\text{water}} = \frac{\gamma_{\text{water}}}{g} = \frac{62.4 \text{ lb/ft}^3}{32.2 \text{ ft/sec}^2} = 1.938 \text{ slugs/ft}^3 \qquad \textit{ANS}$$

$$\gamma_{\text{mercury}} = s_{\text{mercury}}\gamma_{\text{water}} = 13.56(62.4) = 846 \text{ lb/ft}^3 \qquad \textit{ANS}$$

$$\rho_{\text{mercury}} = s_{\text{mercury}}\rho_{\text{water}} = 13.56(1.938) = 26.3 \text{ slugs/ft}^3 \qquad \textit{ANS}$$

[2] Note that in this book we use a "rounded" lower case v (vee), to help distinguish it from a capital V and from the Greek ν (nu).

[3] One cubic centimeter (cm^3) is equivalent to one milliliter (mL).

SAMPLE PROBLEM 2.2 The specific weight of water at ordinary pressure and temperature is 9.81 kN/m³. The specific gravity of mercury is 13.56. Compute the density of water and the specific weight and density of mercury.

Solution

$$\rho_{water} = \frac{9.81\ \text{kN/m}^3}{9.81\ \text{m/s}^2} = 1.00\ \text{Mg/m}^3 = 1.00\ \text{g/mL} \qquad \textit{ANS}$$

$$\gamma_{mercury} = s_{mercury}\gamma_{water} = 13.56(9.81) = 133.0\ \text{kN/m}^3 \qquad \textit{ANS}$$

$$\rho_{mercury} = s_{mercury}\rho_{water} = 13.56(1.00) = 13.56\ \text{Mg/m}^3 \qquad \textit{ANS}$$

EXERCISES

2.3.1 If the specific weight of a liquid is 52 lb/ft³, what is its density?

2.3.2 If the specific weight of a liquid is 8.1 kN/m³, what is its density?

2.3.3 If the specific volume of a gas is 375 ft³/slug, what is its specific weight in lb/ft³?

2.3.4 If the specific volume of a gas is 0.70 m³/kg, what is its specific weight in N/m³?

2.3.5 A certain gas weighs 16.0 N/m³ at a certain temperature and pressure. What are the values of its density, specific volume, and specific gravity relative to air weighing 12.0 N/m³?

2.3.6 The specific weight of glycerin is 78.6 lb/ft³. Compute its density and specific gravity. What is its specific weight in kN/m³?

2.3.7 If a certain gasoline weighs 43 lb/ft³, what are the values of its density, specific volume, and specific gravity relative to water at 60°F? Use Appendix A.

2.4 COMPRESSIBLE AND INCOMPRESSIBLE FLUIDS

Fluid mechanics deals with both incompressible and compressible fluids, that is, with liquids and gases of either constant or variable density. Although there is no such thing in reality as an incompressible fluid, we use this term where the change in density with pressure is so small as to be negligible. This is usually the case with liquids. We may also consider gases to be incompressible when the pressure variation is small compared with the absolute pressure.

Ordinarily we consider liquids to be incompressible fluids, yet sound waves, which are really pressure waves, travel through them. This is evidence of the elasticity of liquids. In problems involving water hammer (Sec. 12.6) we must consider the compressibility of the liquid.

The flow of air in a ventilating system is a case where we may treat a gas as incompressible, for the pressure variation is so small that the change in density is of no importance. But for a gas or steam flowing at high velocity through a long pipeline, the drop in pressure may be so great that we cannot ignore the change in density. For an airplane flying at speeds below 250 mph (100 m/s), we may consider the air to be of constant density. But as an object moving through the air approaches the velocity of sound, which is of the order of 760 mph (1200 km/h) depending on temperature, the pressure and density of the air adjacent to the body become materially different from those of the air at some distance away, and we must then treat the air as a compressible fluid (Chap. 13).

2.5 COMPRESSIBILITY OF LIQUIDS

The compressibility (change in volume due to change in pressure) of a liquid is inversely proportional to its ***volume modulus of elasticity,*** also known as the ***bulk modulus.*** This modulus is defined as

módulo de volumen

$$E_v = -v\frac{dp}{dv} = -\left(\frac{v}{dv}\right)dp$$

where v = specific volume and p = pressure. As v/dv is a dimensionless ratio, the units of E_v and p are identical. The bulk modulus is analogous to the modulus of elasticity for solids; however, for fluids it is defined on a volume basis rather than in terms of the familiar one-dimensional stress–strain relation for solid bodies.

In most engineering problems, the bulk modulus at or near atmospheric pressure is the one of interest. The bulk modulus is a property of the fluid and for liquids is a function of temperature and pressure. A few values of the bulk modulus for water are given in Table 2.1. At any temperature we see that the value of E_v increases continuously with pressure, but at any one pressure the value of E_v is a maximum at about 120°F (50°C). Thus water has a minimum compressibility at about 120°F (50°C).

Note that we often specify applied pressures, such as those in Table 2.1, in absolute terms, because atmospheric pressure varies. The units psia or kN/m² abs indicate absolute pressure, which is the actual pressure on the fluid, relative

TABLE 2.1 **Bulk modulus of water E_v, psi[a]**

	Temperature, °F				
Pressure, psia	**32°**	**68°**	**120°**	**200°**	**300°**
15	293,000	320,000	333,000	308,000	
1,500	300,000	330,000	342,000	319,000	248,000
4,500	317,000	348,000	362,000	338,000	271,000
15,000	380,000	410,000	426,000	405,000	350,000

[a] These values can be transformed to meganewtons per square meter by multiplying them by 0.006895. The values in the first line are for conditions close to normal atmospheric pressure; for a more complete set of values at normal atmospheric pressure, see Table A.1 in Appendix A. The five temperatures are equal to 0, 20, 48.9, 93.3, and 148.9°C, respectively.

to absolute zero. The standard atmospheric pressure at sea level is about 14.7 psia or 101.3 kN/m^2 abs (1013 mb abs) (see Sec. 2.9 and Table A.3). Bars and millibars were previously used in metric systems to express pressure; 1 mb = 100 N/m^2. We measure most pressures relative to the atmosphere, and call them gage pressures. This is explained more fully in Sec. 3.4.

The volume modulus of mild steel is about 26,000,000 psi (170000 MN/m^2). Taking a typical value for the volume modulus of cold water to be 320,000 psi (2200 MN/m^2), we see that water is about 80 times as compressible as steel. The compressibility of liquids covers a wide range. Mercury, for example, is approximately 8% as compressible as water, while the compressibility of nitric acid is nearly six times greater than that of water.

In Table 2.1 we see that at any one temperature the bulk modulus of water does not vary a great deal for a moderate range in pressure. By rearranging the definition of E_v, as an approximation we may use for the case of a fixed mass of liquid at constant temperature

$$\frac{\Delta v}{v} \approx -\frac{\Delta p}{E_v} \tag{2.3a}$$

or

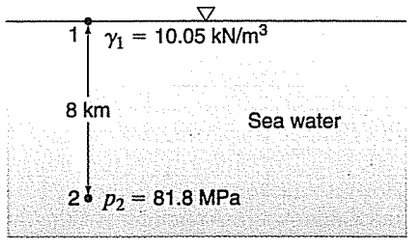
$$\frac{v_2 - v_1}{v_1} \approx -\frac{p_2 - p_1}{E_v} \tag{2.3b}$$

where E_v is the mean value of the modulus for the pressure range and the subscripts 1 and 2 refer to the before and after conditions.

Assuming E_v to have a value of 320,000 psi, we see that increasing the pressure of water by 1000 psi will compress it only $\frac{1}{320}$, or 0.3%, of its original volume. Therefore we find that the usual assumption regarding water as being incompressible is justified.

SAMPLE PROBLEM 2.3 At a depth of 8 km in the ocean the pressure is 81.8 MPa. Assume that the specific weight of seawater at the surface is 10.05 kN/m^3 and that the average volume modulus is 2.34×10^9 N/m^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



(*a*) Eq. (2.2): $v_1 = 1/\rho_1 = g/\gamma_1 = 9.81/10050 = 0.000976$ m^3/kg

Eq. (2.3*a*): $\Delta v = -0.000976(81.8 \times 10^6 - 0)/(2.34 \times 10^9)$

$= -34.1 \times 10^{-6}$ m^3/kg **ANS**

(*b*) Eq. (2.3*b*): $v_2 = v_1 + \Delta v = 0.000942$ m^3/kg **ANS**

(*c*) $\gamma_2 = g/v_2 = 9.81/0.000942 = 10410$ N/m^3 **ANS**

EXERCISES

2.5.1 To two significant figures what is the bulk modulus of water in MN/m^2 at 50°C under a pressure of 30 MN/m^2? Use Table 2.1.

2.5.2 At normal atmospheric conditions, approximately what pressure in psi must be applied to water to reduce its volume by 2%? Use Table 2.1.

2.5.3 Water in a hydraulic press is subjected to a pressure of 4500 psia at 68°F. If the initial pressure is 15 psia, approximately what will be the percentage decrease in specific volume? Use Table 2.1.

2.5.4 At normal atmospheric conditions, approximately what pressure in MPa must be applied to water to reduce its volume by 3%?

2.5.5 A rigid cylinder, inside diameter 15 mm, contains a column of water 500 mm long. What will the column length be if a force of 2 kN is applied to its end by a frictionless plunger? Assume no leakage.

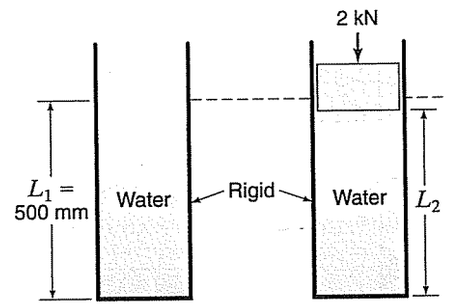


Figure X2.5.5

2.6 SPECIFIC WEIGHT OF LIQUIDS

The specific weights γ of some common liquids at 68°F (20°C) and standard sea-level atmospheric pressure[4] with $g = 32.2$ ft/sec^2 (9.81 m/s^2) are given in Table 2.2. The specific weight of a *liquid* varies only slightly with pressure, depending on the bulk modulus of the liquid (Sec. 2.5); it also depends on temperature, and the variation may be considerable. Since specific weight γ is equal to ρg, the

[4] See Secs. 2.9 and 3.5.

TABLE 2.2 Specific weights γ of common liquids at 68°F (20°C), 14.7 psia (1013 mb abs) with $g = 32.2$ ft/sec^2 (9.81 m/s^2)

	lb/ft^3	kN/m^3
Carbon tetrachloride	99.4	15.6
Ethyl alcohol	49.3	7.76
Gasoline	42	6.6
Glycerin	78.7	12.3
Kerosene	50	7.9
Motor oil	54	8.5
Seawater	63.9	10.03
Water	62.3	9.79

specific weight of a *fluid* depends on the local value of the acceleration of gravity in addition to the variations with temperature and pressure. The variation of the specific weight of water with temperature and pressure, where $g = 32.2$ ft/sec^2 (9.81 m/s^2), is shown in Fig. 2.1. The presence of dissolved air, salts in solution, and suspended matter will increase these values a very slight amount. Ordinarily

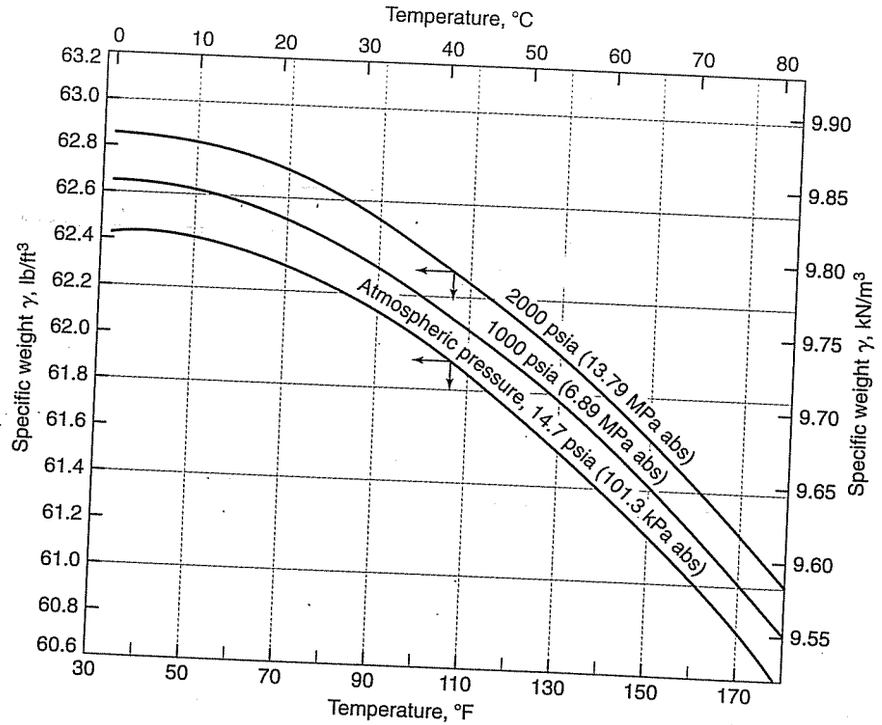
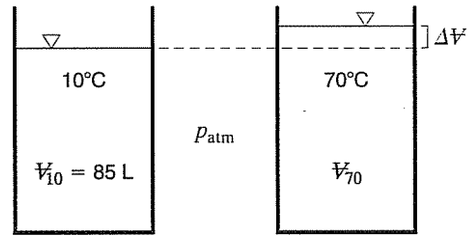


Figure 2.1
Specific weight γ of pure water as a function of temperature and pressure for the condition where $g = 32.2$ ft/sec^2 (9.81 m/s^2).

we assume ocean water to weigh 64.0 lb/ft^3 (10.1 kN/m^3). Unless otherwise specified or implied by a given temperature, the value to use for water in the problems in this book is $\gamma = 62.4$ lb/ft^3 (9.81 kN/m^3). Under extreme conditions the specific weight of water is quite different. For example, at 500°F (260°C) and 6000 psi (42 MN/m^2) the specific weight of water is 51 lb/ft^3 (8.0 kN/m^3).

SAMPLE PROBLEM 2.4 A vessel contains 85 L of water at 10°C and atmospheric pressure. If the water is heated to 70°C, what will be the percentage change in its volume? What weight of water must be removed to maintain the volume at its original value? Use Appendix A.

Solution



Volume, $\forall_{10} = 85 \text{ L} = 0.085 \text{ m}^3$

Table A.1: $\gamma_{10} = 9.804 \text{ kN/m}^3$, $\gamma_{70} = 9.589 \text{ kN/m}^3$

Weight of water, $W = \gamma\forall = \gamma_{10}\forall_{10} = \gamma_{70}\forall_{70}$

i.e., $9.804(0.085) \text{ kN} = 9.589\,\forall_{70}$; $\forall_{70} = 0.08691 \text{ m}^3$

$\Delta\forall = \forall_{70} - \forall_{10} = 0.08691 - 0.08500 = 0.001906 \text{ m}^3$ at γ_{70}

$\Delta\forall/\forall_{10} = 0.001906/0.085 = 2.24\%$ increase ***ANS***

Must remove (at γ_{70}): $W\left(\frac{\Delta\forall}{\forall_{70}}\right) = \gamma_{70}\Delta\forall$

$= (9589 \text{ N/m}^3)(0.001906 \text{ m}^3) = 18.27 \text{ N}$ ***ANS***

EXERCISES

2.6.1 Use Fig. 2.1 to find the approximate specific weight of water in lb/ft^3 under the following conditions: (*a*) at a temperature of 60°C under 101.3 kPa abs pressure; (*b*) at 60°C under a pressure of 13.79 MPa abs.

2.6.2 Use Fig. 2.1 to find the approximate specific weight of water in kN/m^3 under the following conditions: (*a*) at a temperature of 160°F under normal atmospheric pressure; (*b*) at 160°F under a pressure of 2000 psia.

2.6.3 A vessel contains 5.0 ft^3 of water at 40°F and atmospheric pressure. If the water is heated to 80°F, what will be the percentage change in its volume? What weight of water must be removed to maintain the volume at its original value? Use Appendix A.

2.6.4 A cylindrical tank (diameter = 8.00 m and depth = 5.00 m) contains water at 15°C and is brimful. If the water is heated to 60°C, how much water will spill over the edge of the tank? Assume the tank does not expand with the change in temperature. Use Appendix A.

2.7 PROPERTY RELATIONS FOR PERFECT GASES

The various properties of a gas, listed below, are related to one another (see, e.g., Appendix A, Tables A.2 and A.5). They differ for each gas. When the conditions of most real gases are far removed from the liquid phase, these relations closely approximate those of hypothetical ***perfect gases.*** Perfect gases, are here (and often) defined to have constant specific heats[5] and to obey the ***perfect-gas law,***

$$\frac{p}{\rho} = pv = RT \tag{2.4}$$

where p = absolute pressure (Sec. 3.4)
ρ = density (mass per unit volume)
v = specific volume (volume per unit mass, $= 1/\rho$)
R = a gas constant, the value of which depends upon the particular gas
T = absolute temperature in degrees Rankine or Kelvin[6]

For air, the value of R is 1715 ft·lb/(slug·°R) or 287 N·m/(kg·K) (Appendix A, Table A.5); making use of the definitions of a slug and a newton (Sec. 1.5), these units are sometimes given as ft^2/(sec^2·°R) and m^2/(s^2·K), respectively. Since $\gamma = \rho g$, Eq. (2.4) can also be written

$$\gamma = \frac{gp}{RT} \tag{2.5}$$

from which the specific weight of any gas at any temperature and pressure can be computed if R and g are known. Because Eqs. (2.4) and (2.5) relate the various gas properties at a particular state, they are known as ***equations of state*** and as ***property relations.***

In this book we shall assume that all gases are perfect. Perfect gases are sometimes also called ideal gases. Do not confuse a perfect (ideal) gas with an ideal fluid (Sec. 2.10).

[5] Specific heat and other thermodynamic properties of gases are discussed in Sec. 13.1.
[6] Absolute temperature is measured above absolute zero. This occurs on the Fahrenheit scale at −459.67°F (0° Rankine) and on the Celsius scale at −273.15°C (0 Kelvin). Except for low-temperature work, these values are usually taken as −460°F and −273°C. Remember that no degree symbol is used with Kelvin.

Avogadro's law states that all gases at the same temperature and pressure under the action of a given value of g have the same number of molecules per unit of volume, from which it follows that the specific weight of a gas[7] is proportional to its molar mass. Thus, if M denotes ***molar mass*** (formerly called ***molecular weight***), $\gamma_2/\gamma_1 = M_2/M_1$ and, from Eq. (2.5), $\gamma_2/\gamma_1 = R_1/R_2$ for the same temperature, pressure, and value of g. Hence for a perfect gas

$$M_1R_1 = M_2R_2 = \text{constant} = R_0$$

R_0 is known as the ***universal gas constant,*** and has a value of 49,709 ft·lb/(slug-mol·°R) or 8312 N·m/(kg-mol·K). Rewriting the preceding equation in the form

$$R = \frac{R_0}{M}$$

enables us to obtain any gas constant R required for Eq. (2.4) or (2.5).

For real (nonperfect) gases, the specific heats may vary over large temperature ranges, and the right-hand side of Eq. (2.4) is replaced by zRT, so that $R_0 = MzR$, where z is a compressibility factor that varies with pressure and temperature. Values of z and R are given in thermodynamics texts and in handbooks. However, for normally encountered monatomic and diatomic gases, z varies from unity by less than 3%, so the perfect-gas idealizations yield good approximations, and Eqs. (2.4) and (2.5) will give good results.

When various gases exist as a mixture, as in air, ***Dalton's law of partial pressures*** states that each gas exerts its own pressure as if the other(s) were not present. Hence it is the partial pressure of each that we must use in Eqs. (2.4) and (2.5) (see Sample Prob. 2.5). Water vapor as it naturally occurs in the atmosphere has a low partial pressure, so we may treat it as a perfect gas with $R = 49{,}709/18 = 2760$ ft·lb/(slug·°R) [462 N·m/(kg·K)]. But for steam at higher pressures this value is not applicable.

As we increase the pressure and simultaneously lower the temperature, a gas becomes a vapor, and as gases depart more and more from the gas phase and approach the liquid phase, the property relations become much more complicated than Eq (2.4), and we must then obtain specific weight and other properties from vapor tables or charts. Such tables and charts exist for steam, ammonia, sulfur dioxide, freon, and other vapors in common engineering use.

Another fundamental equation for a perfect gas is

$$pv^n = p_1v_1^n = \text{constant} \tag{2.6a}$$

or

$$\frac{p}{p_1} = \left(\frac{\rho}{\rho_1}\right)^n = \text{constant} \tag{2.6b}$$

where p is absolute pressure, v $(=1/\rho)$ is specific volume, ρ is density, and n may have any nonnegative value from zero to infinity, depending on the process to

[7] The specific weight of air (molar mass ≈ 29.0) at 68°F (20°C) and 14.7 psia (1013 mb abs) with $g = 32.2$ ft/sec^2 (9.81 m/s^2) is 0.0752 lb/ft^3 (11.82 N/m^3).

which the gas is subjected. Since this equation describes the change of the gas properties from one state to another for a particular process, we call it a ***process equation.*** If the process of change is at a constant temperature (isothermal), $n = 1$. If there is no heat transfer to or from the gas, the process is ***adiabatic.*** A frictionless (and reversible) adiabatic process is an ***isentropic*** process, for which we denote n by k, where $k = c_p/c_v$, the ratio of specific heat at constant pressure to that at constant volume.[8] This ***specific heat ratio*** k is also called the ***adiabatic exponent.*** For expansion with friction n is less than k, and for compression with friction n is greater than k. Values for k are given in Appendix A, Table A.5, and in thermodynamics texts and handbooks. For air and diatomic gases at usual temperatures, we can take k as 1.4.

By combining Eqs. (2.4) and (2.6), we can obtain other useful relations such as

$$\frac{T_2}{T_1} = \left(\frac{v_1}{v_2}\right)^{n-1} = \left(\frac{\rho_2}{\rho_1}\right)^{n-1} = \left(\frac{p_2}{p_1}\right)^{(n-1)/n} \tag{2.7}$$

SAMPLE PROBLEM 2.5 If an artificial atmosphere consists of 20% oxygen and 80% nitrogen by volume, at 14.7 psia and 60°F, what are (*a*) the specific weight and partial pressure of the oxygen and (*b*) the specific weight of the mixture?

Solution

Table A.5: R (oxygen) $= 1554\ \text{ft}^2/(\text{sec}^2\cdot{}^\circ\text{R})$,
R (nitrogen) $= 1773\ \text{ft}^2/(\text{sec}^2\cdot{}^\circ\text{R})$

Eq. (2.5): 100% O_2: $\gamma = \dfrac{32.2(14.7\times144)}{1554(460+60)} = 0.0843\ \text{lb/ft}^3$

Eq. (2.5): 100% N_2: $\gamma = \dfrac{32.2(14.7\times144)}{1773(520)} = 0.0739\ \text{lb/ft}^3$

(*a*) Each ft^3 of mixture contains 0.2 ft^3 of O_2 and 0.8 ft^3 of N_2.

So for 20% O_2, $\gamma = 0.20(0.0843) = 0.01687\ \text{lb/ft}^3$ ***ANS***

From Eq. (2.5), for 20% O_2, $p = \dfrac{\gamma RT}{g} = \dfrac{0.01687(1554)520}{32.2}$

$= 423\ \text{lb/ft}^2\ \text{abs} = 2.94\ \text{psia}$ ***ANS***

Note that this = 20%(14.7 psia).

(*b*) For 80% N_2, $\gamma = 0.80(0.0739) = 0.0591\ \text{lb/ft}^3$.

Mixture: $\gamma = 0.01687 + 0.0591 = 0.0760\ \text{lb/ft}^3$ ***ANS***

[8] Specific heat and other thermodynamic properties of gases are discussed in Sec. 13.1.

EXERCISES

2.7.1 A gas at 60°C under a pressure of 10000 mb abs has a specific weight of 99 N/m^3. What is the value of R for the gas? What gas might this be? Refer to Appendix A, Table A.5.

2.7.2 A hydrogen-filled balloon of the type used in cosmic-ray studies is to be expanded to its full size, which is a 100-ft-diameter sphere, without stress in the wall at an altitude of 150,000 ft. If the pressure and temperature at this altitude are 0.14 psia and −67°F respectively, find the volume of hydrogen at 14.7 psia and 60°F that should be added on the ground. Neglect the balloon's weight.

2.7.3 Calculate the density, specific weight, and specific volume of air at 120°F and 50 psia.

2.7.4 Calculate the density, specific weight, and specific volume of air at 50°C and 3400 mb abs.

2.7.5 If natural gas has a specific gravity of 0.6 relative to air at 14.7 psia and 68°F, what are its specific weight and specific volume at that same pressure and temperature. What is the value of R for the gas? Solve without using Table A.2.

2.7.6 Given that a sample of dry air at 40°F and 14.7 psia contains 21% oxygen and 78% nitrogen by volume. What is the partial pressure (psia) and specific weight of each gas?

2.7.7 Prove that Eq. (2.7) follows from Eqs. (2.4) and (2.6).

Problem 2.17

2.8 COMPRESSIBILITY OF PERFECT GASES

Differentiating Eq. (2.6) gives $npv^{n-1}dv + v^n dp = 0$. Inserting the value of dp from this into $E_v = -(v/dv)\,dp$ from Sec. 2.5 yields

$$E_v = np \tag{2.8}$$

So for an isothermal process of a gas $E_v = p$, and for an isentropic process $E_v = kp$.

Thus, at a pressure of 15 psia, the isothermal modulus of elasticity for a gas is 15 psi, and for air in an isentropic process it is 1.4(15 psi) = 21 psi. Assuming from Table 2.1 a typical value of the modulus of elasticity of cold water to be 320,000 psi, we see that air at 15 psia is 320,000/15 = 21,000 times as compressible as cold water isothermally, or 320,000/21 = 15,000 times as compressible isentropically. This emphasizes the great difference between the compressibility of normal atmospheric air and that of water.

SAMPLE PROBLEM 2.6 (*a*) Calculate the density, specific weight, and specific volume of oxygen at 100°F and 15 psia (pounds per square inch absolute; see Sec. 2.7). (*b*) What would be the temperature and pressure of this gas if we compressed it isentropically to 40% of its original volume? (*c*) If the process described in (*b*) had been isothermal, what would the temperature and pressure have been?

Solution

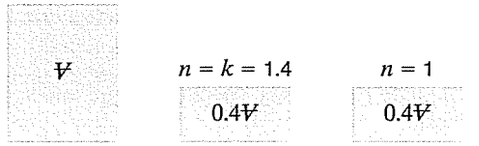


Table A.5 for oxygen (O_2): Molar mass $M = 32.0$, $k = 1.40$

(*a*) Sec. 2.7: $R \approx \dfrac{R_0}{M} = \dfrac{49{,}709}{32.0} = 1553$ ft·lb/(slug·°R) (as in Table A.5)

From Eq. (2.4): $$\rho = \frac{p}{RT} = \frac{15 \times 144 \text{ lb/ft}^2}{[1553 \text{ ft·lb/(slug·°R)}][(460 + 100)\text{°R}]}$$
$$= 0.00248 \text{ slug/ft}^3 \quad \textit{ANS}$$

With $g = 32.2$ ft/sec^2, $\gamma = \rho g = 0.00248(32.2) = 0.0800$ lb/ft^3 ***ANS***

Eq. (2.2): $$v = \frac{1}{\rho} = \frac{1}{0.002\,48} = 403 \text{ ft}^3\text{/slug} \quad \textit{ANS}$$

(*b*) Isentropic compression: $v_2 = 40\% v_1 = 0.4(403) = 161.1$ ft^3/slug

$\rho_2 = 1/v_2 = 0.00621$ slug/ft^3

Eq. (2.6) with $n = k$: $pv^k = (15 \times 144)(403)^{1.4} = (p_2 \times 144)(161.1)^{1.4}$

$p_2 = 54.1$ psia ***ANS***

From Eq. (2.4): $p_2 = 54.1 \times 144$ psia $= \rho RT = 0.00621(1553)(460 + T_2)$

$T_2 = 348$°F ***ANS***

(*c*) Isothermal compression: $T_2 = T_1 = 100$°F ***ANS***

$pv =$ constant: $(15 \times 144)(403) = (p_2 \times 144)(0.4 \times 403)$

$p_2 = 37.5$ psia ***ANS***

SAMPLE PROBLEM 2.7 Calculate the density, specific weight, and specific volume of chlorine gas at 25°C and pressure of 600 kN/m^2 abs (kilonewtons per square meter absolute; see Sec. 2.7). Given the molar mass of chlorine (Cl_2) = 71.

Solution

Sec. 2.7: $$R = \frac{R_0}{M} = \frac{8312}{71} = 117.1 \text{ N·m/(kg·K)}$$

From Eq. (2.4): $$\rho = \frac{p}{RT} = \frac{600\,000 \text{ N/m}^2}{[117.1 \text{ N·m/(kg·K)}][(273 + 25)\text{K}]}$$
$$= 17.20 \text{ kg/m}^3 \quad \textit{ANS}$$

With $g = 9.81$ m/s^2, $\gamma = \rho g = 17.20(9.81) = 168.7$ N/m^3 ***ANS***

Eq. (2.2): $$v = \frac{1}{\rho} = \frac{1}{17.20} = 0.0581 \text{ m}^3\text{/kg} \quad \textit{ANS}$$

EXERCISES

2.8.1 Methane at 22 psia is compressed isothermally, and nitrogen at 16 psia is compressed isentropically. What is the modulus of elasticity of each gas? Which is the more compressible?

2.8.2 Methane at 140 kPa abs is compressed isothermally, and nitrogen at 100 kPa abs is compressed isentropically. What is the modulus of elasticity of each gas? Which is the more compressible?

2.8.3 (*a*) If 10 m^3 of nitrogen at 30°C and 125 kPa are expanded isothermally to 25 m^3, what is the resulting pressure? (*b*) What would the pressure and temperature have been if the process had been isentropic? The adiabatic exponent k for nitrogen is 1.40.

2.8.4 Helium at 25 psia and 65°F is isentropically compressed to one-fifth of its original volume. What is its final pressure?

2.9 STANDARD ATMOSPHERE

Standard atmospheres were first adopted in the 1920s in the United States and in Europe to satisfy a need for standardization of aircraft instruments and aircraft performance. As knowledge of the atmosphere increased, and man's activities in it rose to ever greater altitudes, such standards have been frequently extended and improved.

The International Civil Aviation Organization adopted its latest ***ICAO Standard Atmosphere*** in 1964, which extends up to 32 km (105,000 ft). The International Standards Organization adopted an ***ISO Standard Atmosphere*** to

50 km (164,000 ft) in 1973, which incorporates the ICAO standard. The United States has adopted the ***U.S. Standard Atmosphere,*** last revised in 1976. This incorporates the ICAO and ISO standards, and extends to at least 86 km (282,000 ft or 53.4 mi); for some quantities it extends as far as 1000 km (621 mi).

Figure 2.2 graphically presents variations of temperature and pressure in the U.S. Standard Atmosphere. In the lowest 11.02 km (36,200 ft), called the ***troposphere,*** the temperature decreases rapidly and linearly at a ***lapse rate*** of −6.489°C/km (−3.560°F/1000 ft). In the next layer, called the ***stratosphere,*** about 9 km (30,000 ft) thick, the temperature remains constant at −56.5°C (−69.7°F). Next, in the ***mesosphere,*** it increases, first slowly and then more rapidly, to a maximum of −2.5°C (27.5°F) at an altitude around 50 km (165,000 ft or 31 mi). Above this, in the upper part of the mesosphere known as the ***ionosphere,*** the temperature again decreases.

The standard absolute pressure[9] behaves very differently from temperature (Fig. 2.2), decreasing quite rapidly and smoothly to almost zero at an altitude of

[9] Absolute pressure is discussed in Secs. 2.7 and 3.4.

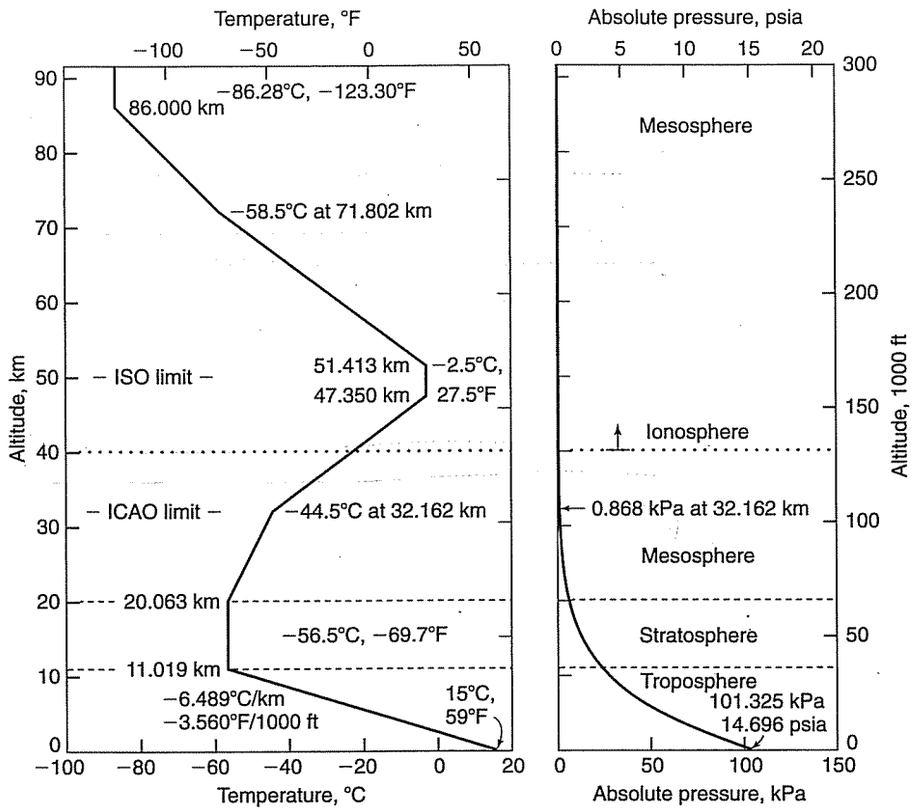


Figure 2.2
The U.S. Standard Atmosphere, temperature, and pressure distributions.

30 km (98,000 ft). The pressure profile was computed from the standard temperatures using methods of fluid statics (Sec. 3.2). The representation of the standard temperature profile by a number of linear functions of elevation (Fig. 2.2) greatly facilitates such computations (see Sample Prob. 3.1*d*).

Temperature, pressure, and other variables from the ICAO Standard Atmosphere, including density and viscosity, are tabulated together with gravitational acceleration out to 30 km and 100,000 ft in Appendix A, Table A.3. Engineers generally use such data in design calculations where the performance of high-altitude aircraft is of interest. The standard atmosphere serves as a good approximation of conditions in the atmosphere; of course the actual conditions vary somewhat with the weather, the seasons, and the latitude.

2.10 IDEAL FLUID

An ***ideal*** fluid is usually defined as a fluid in which there is no *friction;* it is ***inviscid*** (its viscosity is zero). Thus the internal forces at any section within it are always normal to the section, even during motion. So these forces are purely pressure forces. Although such a fluid does not exist in reality, many fluids approximate frictionless flow at sufficient distances from solid boundaries, and so we can often conveniently analyze their behaviors by assuming an ideal fluid. As noted in Sec. 2.7, take care to not confuse an ideal fluid with a perfect (ideal) gas.

In a ***real*** fluid, either liquid or gas, tangential or shearing forces always develop whenever there is motion relative to a body, thus creating fluid friction, because these forces oppose the motion of one particle past another. These friction forces give rise to a fluid property called viscosity.

2.11 VISCOSITY

The ***viscosity*** of a fluid is a measure of its resistance to shear or angular deformation. Motor oil, for example, has high viscosity and resistance to shear, is cohesive, and feels "sticky," whereas gasoline has low viscosity. The friction forces in flowing fluid result from the cohesion and momentum interchange between molecules. Figure 2.3 indicates how the viscosities of typical fluids depend on temperature. As the temperature increases, the viscosities of all liquids *decrease,* while the viscosities of all gases *increase*. This is because the force of cohesion, which diminishes with temperature, predominates with liquids, while with gases the predominating factor is the interchange of molecules between the layers of different velocities. Thus a rapidly-moving gas molecule shifting into a slower-moving layer tends to speed up the latter. And a slow-moving molecule entering a faster-moving layer tends to slow down the faster-moving layer. This molecular interchange sets up a shear, or produces a friction force between adjacent layers. At higher temperatures molecular activity increases, so causing the viscosity of gases to increase with temperature.

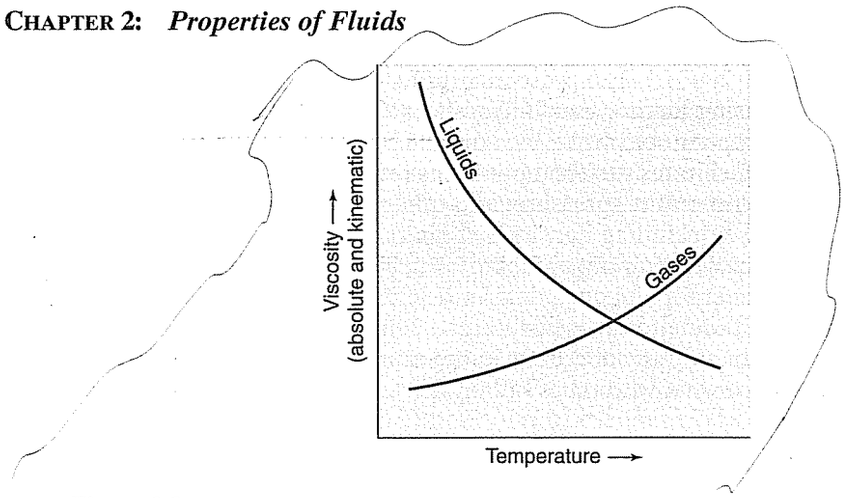


Figure 2.3
Trends in viscosity variation with temperature.

Figures A.1 and A.2 in Appendix A graphically present numerical values of absolute and kinematic viscosities for a variety of liquids and gases, and show how they vary with temperature.

Consider the classic case of two parallel plates (Fig. 2.4), sufficiently large that we can neglect edge conditions, a small distance Y apart, with fluid filling the space between. The lower plate is stationary, while the upper one moves parallel to it with a velocity U due to a force F corresponding to some area A of the moving plate.

At boundaries, particles of fluid adhere to the walls, and so their velocities are zero relative to the wall. This so-called ***no-slip condition*** occurs with all viscous fluids. Thus in Fig. 2.4 the fluid velocities must be U where in contact with the plate at the upper boundary and zero at the lower boundary. We call the form of the velocity variation with distance between these two extremes, as depicted in Fig. 2.4, a ***velocity profile.*** If the separation distance Y is not too great, if the velocity U is not too high, and if there is no net flow of fluid through the space, the velocity profile will be linear, as in Fig. 2.4*a*. If, in addition, there is a small amount of bulk fluid transport between the plates, as could result from pressure-fed lubrication for example, the velocity profile becomes the sum of

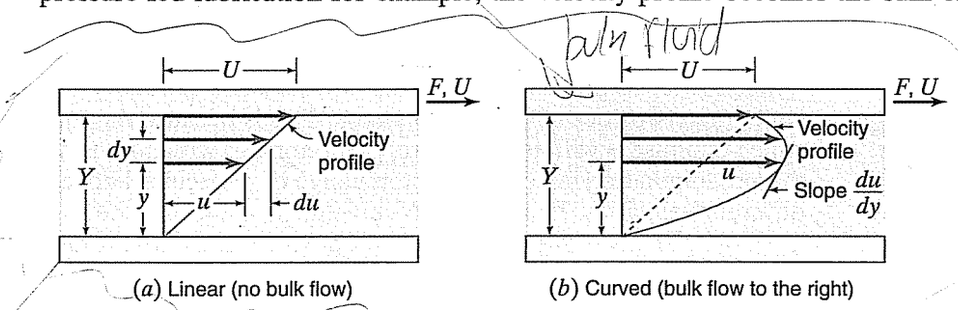


Figure 2.4
Velocity profiles.

the previous linear profile plus a parabolic profile (Fig. 2.4*b*); the parabolic additions to (or subtractions from) the linear profile are zero at the walls (plates) and maximum at the centerline. The behavior of the fluid is much as if it consisted of a series of thin layers, each of which slips a little relative to the next.

For a large class of fluids under the conditions of Fig. 2.4*a*, experiments have shown that

$$F \propto \frac{AU}{Y}$$

We see from similar triangles that we can replace U/Y by the velocity gradient du/dy. If we now introduce a constant of proportionality μ (mu), we can express the shearing stress τ (tau) between any two thin sheets of fluid by

$$\tau = \frac{F}{A} = \mu\frac{U}{Y} = \mu\frac{du}{dy} \tag{2.9}$$

We call Eq. (2.9) ***Newton's equation of viscosity,*** since Sir Isaac Newton (1642–1727) first suggested it. Although better known for his formulation of the fundamental laws of motion and gravity and for the development of differential calculus, Newton, an English mathematician and natural philosopher, also made many pioneering studies in fluid mechanics. In transposed form, Eq. (2.9) defines the proportionality constant

$$\mu = \frac{\tau}{du/dy} \tag{2.10}$$

known as the ***coefficient of viscosity,*** the ***absolute viscosity,*** the ***dynamic viscosity*** (since it involves force), or simply the ***viscosity*** of the fluid. We shall use "absolute viscosity" to help differentiate it from another viscosity that we will discuss shortly.

We noted in Sec. 2.1 that the distinction between a solid and a fluid lies in the manner in which each can resist shearing stresses. We will clarify a further distinction among various kinds of fluids and solids by referring to Fig. 2.5. In

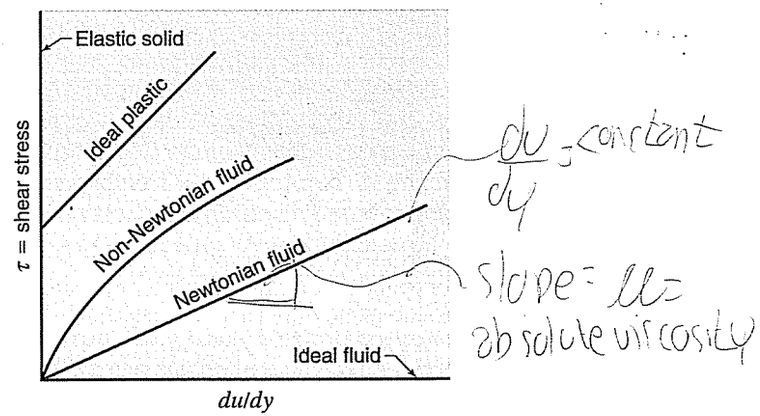


Figure 2.5

the case of a solid, shear stress depends on the *magnitude* of the deformation; but Eq. (2.9) shows that in many fluids the shear stress is proportional to the *time rate* of (angular) deformation.

A fluid for which the constant of proportionality (i.e., the absolute viscosity) does not change with rate of deformation is called a ***Newtonian fluid,*** and this plots as a straight line in Fig. 2.5. The slope of this line is the absolute viscosity, μ. The ideal fluid, with no viscosity (Sec. 2.10), falls on the horizontal axis, while the true elastic solid plots along the vertical axis. A plastic that sustains a certain amount of stress before suffering a plastic flow corresponds to a straight line intersecting the vertical axis at the yield stress. There are certain non-Newtonian fluids[10] in which μ varies with the rate of deformation. These are relatively uncommon in engineering usage, so we will restrict the remainder of this text to the common fluids that under normal conditions obey Newton's equation of viscosity.

In a ***journal bearing,*** lubricating fluid fills the small annular space between a shaft and its surrounding support. This fluid layer is very similar to the layer between the two parallel plates, except it is curved. There is another more subtle difference, however. For coaxial cylinders (Fig. 2.6) with constant rotative speed ω (omega), the resisting and driving torques are equal. But because the radii at the inner and outer walls are different, it follows that the shear stresses

[10] Typical non-Newtonian fluids include paints, printer's ink, gels and emulsions, sludges and slurries, and certain plastics. An excellent treatment of the subject is given by W. L. Wilkinson in *NonNewtonian Fluids,* Pergamon Press, New York, 1960.

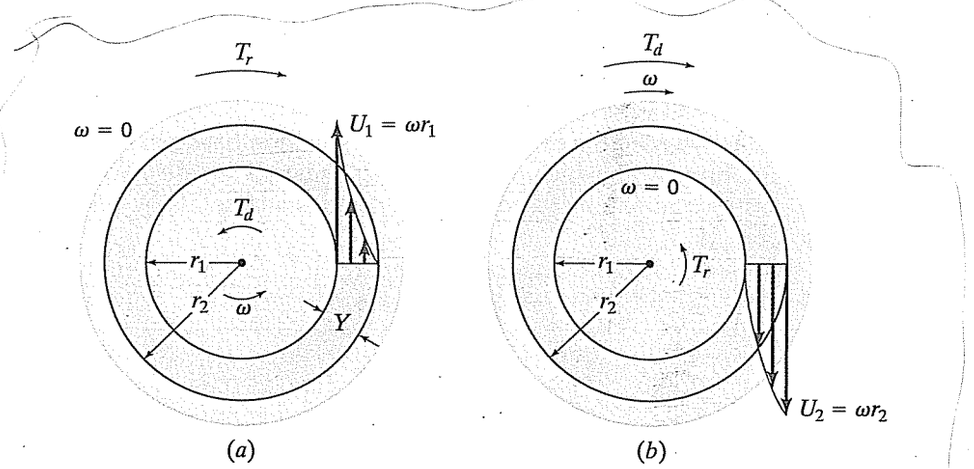


Figure 2.6
Velocity profile, rotating coaxial cylinders with gap completely filled with liquid. (*a*) Inner cylinder rotating. (*b*) Outer cylinder rotating. Z is the dimension at right angles to the plane of the sketch. Resisting torque = driving torque and $\tau \propto (du/dy)$.

$$\tau_1(2\pi r_1 Z)r_1 = \tau_2(2\pi r_2 Z)r_2, \qquad \left(\frac{du}{dy}\right)_1 = \left(\frac{du}{dy}\right)_2 \frac{r_2^2}{r_1^2}$$

and velocity gradients there must also be different (see Fig. 2.6 and equations that accompany it). The shear stress and velocity gradient must vary continuously across the gap, and so the velocity profile must curve. However, as the gap distance $Y \to 0$, $du/dy \to U/Y$ = constant. So, when the gap is very small, we can assume the velocity profile to be a straight line, and we can solve problems in a similar manner as for flat plates.

The dimensions of absolute viscosity are force per unit area divided by velocity gradient. In the British Gravitational (BG) system the dimensions of absolute viscosity are as follows:

$$\text{Dimensions of } \mu = \frac{\text{dimensions of } \tau}{\text{dimensions of } du/dy} = \frac{\text{lb/ft}^2}{\text{fps/ft}} = \text{lb}\cdot\text{sec/ft}^2$$

In SI units

$$\text{Dimensions of } \mu = \frac{\text{N/m}^2}{\text{s}^{-1}} = \text{N}\cdot\text{s/m}^2$$

A widely used unit for viscosity in the metric system is the ***poise*** (P), named after Jean Louis Poiseuille (1799–1869). A French anatomist, Poiseuille was one of the first investigators of viscosity. The poise = 0.10 N·s/m^2. The ***centipoise*** (cP) (= 0.01 P = 1 mN·s/m^2) is frequently a more convenient unit. It has a further advantage in that the viscosity of water at 68.4°F is 1 cP. Thus the value of the viscosity in centipoises is an indication of the viscosity of the fluid relative to that of water at 68.4°F.

In many problems involving viscosity the absolute viscosity is divided by density. This ratio defines the ***kinematic viscosity*** ν (nu), so called because force is not involved, the only dimensions being length and time, as in kinematics (Sec. 1.1). Thus

$$\nu = \frac{\mu}{\rho} \tag{2.11}$$

We usually measure kinematic viscosity ν in ft^2/sec in the BG system, and in m^2/s in the SI. Previously, in the metric system the common units were cm^2/s, also called the ***stoke*** (St), after Sir George Stokes (1819–1903), an English physicist and pioneering investigator of viscosity. Many found the ***centistoke*** (cSt) (0.01 St = 10^{-6} m^2/s) a more convenient unit to work with.

An important practical distinction between the two viscosities is the following. The absolute viscosity μ of most fluids is virtually independent of pressure for the range that is ordinarily encountered in engineering work; for extremely high pressures, the values are a little higher than those shown in Fig. A.1. The kinematic viscosity ν of gases, however, varies strongly with pressure because of changes in density. Therefore, if we need to determine the kinematic viscosity ν at a nonstandard pressure, we can look up the (pressure-independent) value of μ and calculate ν from Eq. (2.11). This will require knowing the gas density, ρ, which, if necessary, we can calculate using Eq. (2.4).

The *measurement* of viscosity is described in Sec. 11.1.

SAMPLE PROBLEM 2.8 A 1-in-wide space between two horizontal plane surfaces is filled with SAE 30 Western lubricating oil at 80°F. What force is required to drag a very thin plate of 4-ft^2 area through the oil at a velocity of 20 ft/min if the plate is 0.33 in from one surface?

Solution

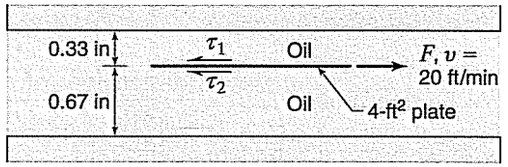


Fig. A.1: $\mu = 0.0063$ lb·sec/ft^2

Eq. (2.9): $\tau_1 = 0.0063 \times (20/60)/(0.33/12) = 0.0764$ lb/ft^2

Eq. (2.9): $\tau_2 = 0.0063 \times (20/60)/(0.67/12) = 0.0394$ lb/ft^2

From Eq. (2.9): $F_1 = \tau_1 A = 0.0764 \times 4 = 0.305$ lb

From Eq. (2.9): $F_2 = \tau_2 A = 0.0394 \times 4 = 0.158$ lb

Force $= F_1 + F_2 = 0.463$ lb ***ANS***

SAMPLE PROBLEM 2.9 In Fig. S2.9 oil of absolute viscosity μ fills the small gap of thickness Y. (*a*) Neglecting fluid stress exerted on the circular underside, obtain an expression for the torque T required to rotate the truncated cone at constant speed ω. (*b*) What is the rate of heat generation, in joules per second, if the oil's absolute viscosity is 0.20 N·s/m^2, $\alpha = 45°$, $a = 45$ mm, $b = 60$ mm, $Y = 0.2$ mm, and the speed of rotation is 90 rpm?

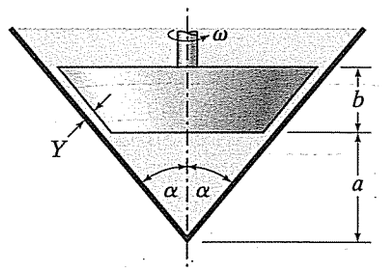


Figure S2.9

Solution

(*a*) $U = \omega r$; for small gap Y, $\dfrac{du}{dy} = \dfrac{U}{Y} = \dfrac{\omega r}{Y}$

Eq. (2.9): $\tau = \mu \dfrac{du}{dy} = \dfrac{\mu \omega r}{Y}$; $dA = 2\pi r\, ds = \dfrac{2\pi r\, dy}{\cos\alpha}$

From Eq. (2.9): $dF = \tau\, dA = \dfrac{\mu \omega r}{Y}\left(\dfrac{2\pi r\, dy}{\cos\alpha}\right)$

$$dT = r\, dF = \frac{2\pi\mu\omega}{Y\cos\alpha} r^3 dy; \quad r = y\tan\alpha$$

$$dT = \frac{2\pi\mu\omega \tan^3\alpha}{Y\cos\alpha} y^3 dy$$

$$T = \frac{2\pi\mu\omega \tan^3\alpha}{Y\cos\alpha} \int_a^{a+b} y^3 dy; \quad \frac{y^4}{4}\bigg|_a^{a+b} = \left[\frac{(a+b)^4}{4} - \frac{a^4}{4}\right]$$

$$T = \frac{2\pi\mu\omega\tan^3\alpha}{4Y\cos\alpha}[(a+b)^4 - a^4] \qquad \textit{ANS}$$

(*b*) $[(a+b)^4 - a^4] = (0.105\text{ m})^4 - (0.045\text{ m})^4 = 0.0001175\text{ m}^4$

$$\omega = \left(90\frac{\text{rev}}{\text{min}}\right)\left(2\pi\frac{\text{radians}}{\text{rev}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right) = 3\pi\text{ rad/s} = 3\pi\text{ s}^{-1}$$

Heat generation rate = power $= T\omega = \dfrac{2\pi\mu\omega^2\tan^3\alpha}{4Y\cos\alpha}[(a+b)^4 - a^4]$

$$= \frac{2\pi(0.20\text{ N·s/m}^2)(3\pi\text{ s}^{-1})^2(1)^3[0.0001175\text{ m}^4]}{4(2\times10^{-4}\text{ m})\cos45°}$$

$= 23.2$ N·m/s $= 23.2$ J/s ***ANS***

EXERCISES

2.11.1 At 60°F what is the kinematic viscosity of the gasoline in Fig. A.2, the specific gravity of which is 0.680? Give the answer in both BG and SI units.

2.11.2 To what temperature must the fuel oil with the higher specific gravity in Fig. A.2 be heated in order that its kinematic viscosity may be reduced to three times that of water at 40°F?

2.11.3 Compare the ratio of the absolute viscosities of air and water at 70°F with the ratio of their kinematic viscosities at the same temperature and at 14.7 psia.

2.11.4 A flat plate 200 mm × 750 mm slides on oil ($\mu = 0.85$ N·s/m^2) over a large plane surface (Fig. X2.11.4). What force F is required to drag the plate at a velocity v of 1.2 m/s, if the thickness t of the separating oil film is 0.6 mm?

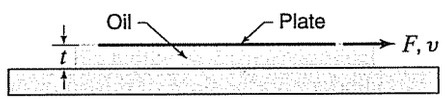


Figure X2.11.4

2.11.5 Refer to Fig. X2.11.4. A flat plate 2 ft × 3 ft slides on oil ($\mu = 0.024$ lb·sec/ft^2) over a large plane surface. What force F is required to drag the plate at a velocity v of 4 ft/sec, if the thickness t of the separating oil film is 0.025 in?

2.11.6 A liquid has an absolute viscosity of 3.2×10^{-4} lb·sec/ft^2. It weighs 56 lb/ft^3. What are its absolute and kinematic viscosities in SI units?

2.11.7 (*a*) What is the ratio of the absolute viscosity of water at a temperature of 70°F to that of water at 200°F? (*b*) What is the ratio of the absolute viscosity of the crude oil in Fig. A.1 ($s = 0.925$) to that of the gasoline ($s = 0.680$), both being at a temperature of 60°F? (*c*) In cooling from 300 to 80°F, what is the ratio of the change of the absolute viscosity of the SAE 30 Western oil to that of the SAE 30 Eastern oil? Refer to Appendix A.

2.11.8 A space 16 mm wide between two large plane surfaces is filled with SAE 30 Western lubricating oil at 35°C (Fig. X2.11.8). What force F is required to drag a very thin plate of 0.4 m^2 area between the surfaces at a speed $v = 0.25$ m/s (*a*) if the plate is equally spaced between the two surfaces, and (*b*) if $t = 5$ mm? Refer to Appendix A.

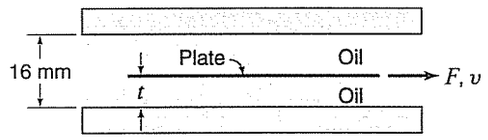


Figure X2.11.8

2.11.9 A journal bearing consists of an 80-mm shaft in an 80.4-mm sleeve 120 mm long, the clearance space (assumed to be uniform) being filled with SAE 30 Western lubricating oil at 40°C (Fig. X2.11.9). Calculate the rate at which heat is generated at the bearing when the shaft turns at 150 rpm. Express the answer in kN·m/s, J/s, Btu/hr, ft·lb/sec, and hp. Refer to Appendix A.

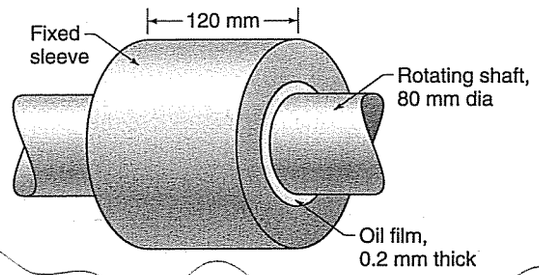


Figure X2.11.9

2.11.10 In using a rotating-cylinder viscometer, a bottom correction must be applied to account for the drag on the flat bottom of the inner cylinder. Calculate the theoretical amount of this torque correction, neglecting centrifugal effects, for a cylinder of diameter d, rotated at a constant angular velocity ω, in a liquid of absolute viscosity μ, with a clearance Δh between the bottom of the inner cylinder and the floor of the outer one.

2.11.11 Assuming a velocity distribution as shown in Fig. X2.11.11, which is a parabola having its vertex 12 in from the boundary, calculate the velocity gradients for $y = 0, 3, 6, 9$, and 12 in. Also calculate the shear stresses in lb/ft^2 at these points if the fluid's absolute viscosity is 600 cP.

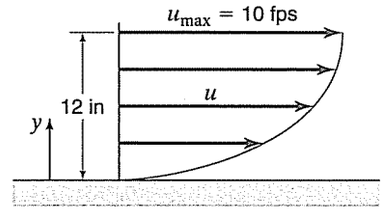


Figure X2.11.11

2.11.12 Air at 50 psia and 60°F is flowing through a pipe. Table A.2 indicates that its kinematic viscosity ν is 0.158×10^{-3} ft^2/sec. (*a*) Why is this ν value incorrect? (*b*) What is the correct value?

2.12 SURFACE TENSION

Liquids have cohesion and adhesion, both of which are forms of molecular attraction. ***Cohesion*** enables a liquid to resist tensile stress, while ***adhesion*** enables it to adhere to another body.[11] At the interface between a liquid and a gas, i.e., at the liquid surface, and at the interface between two ***immiscible*** (not mixable) liquids, the out-of-balance attraction force between molecules forms an imaginary surface film which exerts a tension force in the surface. This liquid property is known as ***surface tension.*** Because this tension acts in a surface, we compare such forces by measuring the tension force per unit length of surface. When a second fluid is not specified at the interface, it is understood that the liquid surface is in contact with air. The surface tensions of various liquids cover a wide range, and they decrease slightly with increasing temperature. Values of the surface tension for water between the freezing and boiling points vary from 0.00518 to 0.00404 lb/ft (0.0756 to 0.0589 N/m); Table A.1 of Appendix A contains more typical values. Table A.4 includes values for other liquids. ***Capillarity*** is the property of exerting forces on fluids by fine tubes or porous media; it is due to both cohesion and adhesion. When the cohesion is of less effect than the adhesion, the liquid will wet a solid surface it touches and rise at the point of contact; if cohesion predominates, the liquid surface will depress at the point of contact. For example, capillarity makes water rise in a glass tube, while mercury depresses below the true level, as shown in the insert in Fig. 2.7, which is drawn to scale and reproduced actual size. We call the curved liquid surface that develops in a tube a ***meniscus.***

A cross section through capillary rise in a tube looks like Fig. 2.8. From free-body considerations, equating the lifting force created by surface tension to

[11] In 1877 Osborne Reynolds demonstrated that a $\frac{1}{4}$-in-diameter column of mercury could withstand a tensile stress (negative pressure, below atmospheric) of 3 atm (44 psi or 304 kPa) for a time, but that it would separate upon external jarring of the tube. Liquid tensile stress (said to be as high as 400 atm) accounts for the rise of water in the very small channels of xylem tissue in tall trees. For practical engineering purposes, however, we assume liquids are incapable of resisting any direct tensile stress.

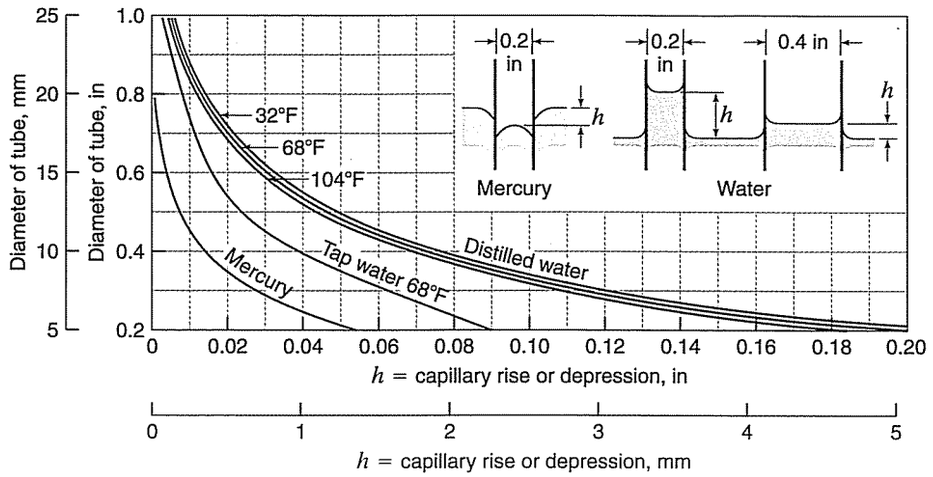


Figure 2.7
Capillarity in clean circular glass tubes, for liquid in contact with air.

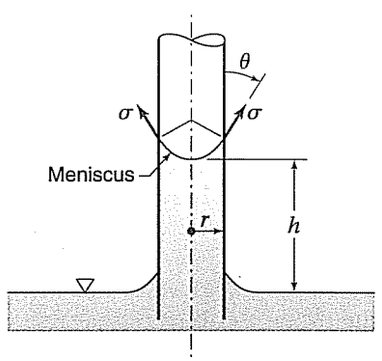


Figure 2.8
Capillary rise.

the gravity force,

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

so

$$h = \frac{2\sigma \cos\theta}{\gamma r} \qquad (2.12)$$

where σ = surface tension (sigma) in units of force per unit length
θ = wetting angle (theta)
γ = specific weight of liquid
r = radius of tube
h = capillary rise[12]

[12] Measurements to a meniscus are usually taken to the point on the centerline.

We can use this expression to compute the *approximate* capillary rise or depression in a tube. If the tube is clean, $\theta = 0°$ for water and about 140° for mercury. Note that the meniscus (Figs. 2.7 and 2.8) lifts a small volume of liquid, near the tube walls, in addition to the volume $\pi r^2 h$ used in Eq. (2.12). For larger tube diameters, with smaller capillary rise heights, this small additional volume can become a large fraction of $\pi r^2 h$. So Eq. (2.12) overestimates the amount of capillary rise or depression, particularly for larger diameter tubes. The curves of Fig. 2.7 are for water or mercury in contact with air; if mercury is in contact with water, the surface tension effect is slightly less than when in contact with air. For tube diameters larger than $\frac{1}{2}$ in (12 mm), capillary effects are negligible.

Surface tension effects are generally negligible in most engineering situations. However, they can be important in problems involving capillary rise, such as in the soil water zone; without capillarity most forms of vegetable life would perish. When we use small tubes to measure fluid properties, such as pressures, we must take the readings while aware of the surface tension effects; a true reading would occur if surface tension effects were zero. These effects are also important in hydraulic model studies when the model is small, in the breakup of liquid jets, and in the formation of drops and bubbles. The formation of drops is extremely complex to analyze, but is, for example, of critical concern in the design of inkjet printers, a multi-billion-dollar business.

SAMPLE PROBLEM 2.10 Water at 10°C stands in a clean glass tube of 2-mm diameter at a height of 35 mm. What is the true static height?

Solution

Table A.1 at 10°C: $\gamma = 9804\ \text{N/m}^3$, $\sigma = 0.0742\ \text{N/m}$.

Sec. 2.12 for clean glass tube: $\theta = 0°$.

Eq. (2.12): $$h = \frac{2\sigma}{\gamma r} = \frac{2(0.0742\ \text{N/m})}{(9804\ \text{N/m}^3)0.001\ \text{m}}$$
$$= 0.015\,14\ \text{m} = 15.14\ \text{mm}$$

Sec. 2.12: True static height = 35.00 − 15.14 = 19.86 mm ***ANS***

EXERCISES

2.12.1 Tap water at 68°F stands in a glass tube of 0.32-in diameter at a height of 4.50 in. What is the true static height?

2.12.2 Distilled water at 20°C stands in a glass tube of 6.0-mm diameter at a height of 18.0 mm. What is the true static height?

2.12.3 Use Eq. (2.12) to compute the capillary depression of mercury at 68°F ($\theta = 140°$) to be expected in a 0.05-in-diameter tube.

2.12.4 Compute the capillary rise in mm of pure water at 10°C expected in an 0.8-mm-diameter tube.

2.12.5 Use Eq. (2.12) to compute the capillary rise of water to be expected in a 0.28-in-diameter tube. Assume pure water at 68°F. Compare the result with Fig. 2.7.

2.13 VAPOR PRESSURE OF LIQUIDS

All liquids tend to evaporate or vaporize, which they do by projecting molecules into the space above their surfaces. If this is a confined space, the partial pressure exerted by the molecules increases until the rate at which molecules reenter the liquid is equal to the rate at which they leave. For this equilibrium condition, we call the vapor pressure the ***saturation pressure.***

Molecular activity increases with increasing temperature and decreasing pressure, and so the saturation pressure does the same. At any given temperature, if the pressure on the liquid surface falls below the saturation pressure, a rapid rate of evaporation results, known as ***boiling.*** Thus we can refer to the saturation pressure as the ***boiling pressure*** for a given temperature, and it is of practical importance for liquids.[13]

We call the rapid vaporization and recondensation of liquid as it briefly passes through a region of low absolute pressure ***cavitation.*** This phenomenon is often very damaging, and so we must avoid it; we shall discuss it in more detail in Sec. 5.10.

Table 2.3 calls attention to the wide variation in saturation vapor pressure of various liquids; Appendix A, Table A.4 contains more values. The very low vapor pressure of mercury makes it particularly suitable for use in barometers. Values for the vapor pressure of water at different temperatures are in Appendix A, Table A.1.

TABLE 2.3 **Saturation vapor pressure of selected liquids at 68°F (20°C)**

	psia	N/m² abs	mb abs
Mercury	0.000025	0.17	0.0017
Water	0.34	2340	23.4
Carbon tetrachloride	1.90	13100	131
Gasoline	8.0	55200	552

[13] Values of the saturation pressure for water for temperatures from 32 to 705.4°F can be found in J. H. Keenan, *Thermodynamic Properties of Water including Vapor, Liquid and Solid States,* John Wiley & Sons, Inc., New York, 1969, and in other steam tables. There are similar vapor tables published for ammonia, carbon dioxide, sulfur dioxide, and other vapors of engineering interest.

SAMPLE PROBLEM 2.11 At approximately what temperature will water boil if the elevation is 10,000 ft?

Solution
From Appendix A, Table A.3, the pressure of the standard atmosphere at 10,000-ft elevation is 10.11 psia. From Appendix A, Table A.1, the saturation vapor pressure p_v of water is 10.11 psia at about 193°F (by interpolation). Hence the water at 10,000 ft will boil at about 193°F. ***ANS***

Compared with the boiling temperature of 212°F at sea level, this explains why it takes longer to cook at high elevations.

EXERCISES

2.13.1 At what pressure in millibars absolute will 70°C water boil?

2.13.2 At approximately what temperature will water boil in Mexico City (elevation 7400 ft)? Refer to Appendix A.

PROBLEMS

2.1 If the specific weight of a gas is 12.40 N/m³, what is its specific volume in m³/kg?

2.2 A gas sample weighs 0.108 lb/ft³ at a certain temperature and pressure. What are the values of its density, specific volume, and specific gravity relative to air weighing 0.075 lb/ft³?

2.3 If a certain liquid weighs 8600 N/m³, what are the values of its density, specific volume, and specific gravity relative to water at 15°C? Use Appendix A.

2.4 Find the change in volume of 15.00 lb of water at ordinary atmospheric pressure for the following conditions: (*a*) reducing the temperature by 50°F from 200°F to 150°F; (*b*) reducing the temperature by 50°F from 150°F to 100°F; (*c*) reducing the temperature by 50°F from 100°F to 50°F. Calculate each and note the trend in the changes in volume.

2.5 Initially when 1000.00 mL of water at 10°C are poured into a glass cylinder, the height of the water column is 1000.0 mm. The water and its container are heated to 70°C. Assuming no evaporation, what then will be the depth of the water column if the coefficient of thermal expansion for the glass is 3.8×10^{-6} mm/mm per °C?

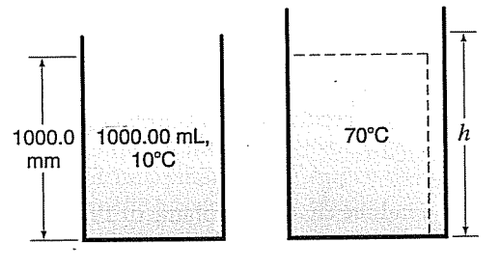


Figure P2.5

2.6 At a depth of 4 miles in the ocean the pressure is 9520 psi. Assume that the specific weight at the surface is 64.00 lb/ft³ and that the average volume modulus is 320,000 psi 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? (*d*) What is the

percentage change in the specific volume? (*e*) What is the percentage change in the specific weight?

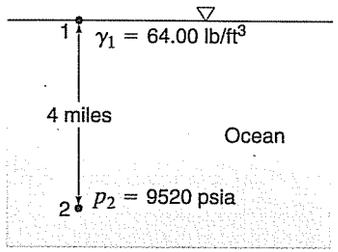


Figure P2.6

2.7 Water at 68°F is in a long, rigid cylinder of inside diameter 0.600 in. A plunger applies pressure to the water. If, with zero force, the initial length of the water column is 25.00 in, what will its length be if a force of 420 lb is applied to the plunger. Assume no leakage and no friction.

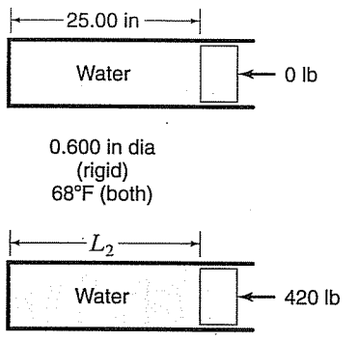


Figure P2.7

2.8 Find the change in volume of 10 m³ of water for the following situations: (*a*) a temperature increase from 60°C to 70°C with constant atmospheric pressure, (*b*) a pressure increase from zero to 10 MN/m² with temperature remaining constant at 60°C, (*c*) a temperature decrease from 60°C to 50°C combined with a pressure increase of 10 MN/m².

2.9 A heavy closed steel chamber is filled with water at 40°F and atmospheric pressure. If the temperature of the water and the chamber is raised to 80°F, what will be the new pressure of the water? The coefficient of thermal expansion of the steel is 6.6 × 10^{-6} in/in per °F; assume the chamber is unaffected by the water pressure. Use Table A.1 and Fig. 2.1.

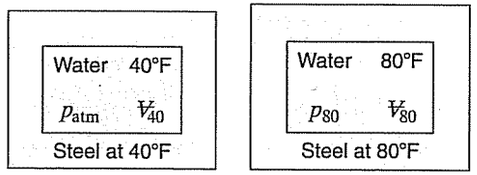


Figure P2.9

2.10 Repeat Exer. 2.6.4 for the case where the tank is made of a material that has a coefficient of thermal expansion of 4.6 × 10^{-6} mm/mm per °C.

2.11 (*a*) Calculate the density, specific weight, and specific volume of oxygen at 20°C and 50 kN/m² abs. (*b*) If the oxygen is enclosed in a rigid container of constant volume, what will be the pressure if the temperature is reduced to −100°C?

2.12 (*a*) If water vapor in the atmosphere has a partial pressure of 0.50 psia and the temperature is 90°F, what is its specific weight? (*b*) If the barometer reads 14.50 psia, what is the partial pressure of the (dry) air, and what is its specific weight? (*c*) What is the specific weight of the atmosphere (air plus the water vapor present)?

2.13 (*a*) If water vapor in the atmosphere has a partial pressure of 3500 Pa and the temperature is 30°C, what is its specific weight? (*b*) If the barometer reads 102 kPa abs, what is the partial pressure of the (dry) air, and what is its specific weight? (*c*) What is the specific weight of the atmosphere (air plus the water vapor present)?

2.14 If the specific weight of water vapor in the atmosphere is 0.00065 lb/ft³ and that of the (dry) air is 0.074 lb/ft³ when the temperature is 70°F, (*a*) what are the partial pressures of the water vapor and the dry air in psia, (*b*) what is the specific weight of the atmosphere (air and water vapor), and (*c*) what is the barometric pressure in psia?

2.15 If an artificial atmosphere consists of 20% oxygen and 80% nitrogen by volume, at 101.32 kN/m² abs and 20°C, what are (*a*) the specific weight and partial pressure of the oxygen, (*b*) the specific weight and partial pressure of the nitrogen, and (*c*) the specific weight of the mixture?

2.16 When the ambient air is at 70°F, 14.7 psia, and contains 21% oxygen by volume, 4.5 lb of air are pumped into a scuba tank, capacity 0.75 ft³. (*a*) What volume of ambient air was compressed? (*b*) When the filled tank has cooled to ambient conditions, what is the (gage) pressure of the air in the tank? (*c*) What is the partial pressure (psia) and specific weight of the ambient oxygen? (*d*) What weight of oxygen was put in the tank? (*e*) What is the partial pressure (psia) and specific weight of the oxygen in the tank?

2.17 (*a*) If 10 ft³ of carbon dioxide at 50°F and 15 psia is compressed isothermally to 2 ft³, what is the resulting pressure? (*b*) What would the pressure and temperature have been if the process had been isentropic? The adiabatic exponent *k* for carbon dioxide is 1.28.

2.18 (*a*) If 350 L of carbon dioxide at 20°C and 120 kN/m² abs is compressed isothermally to 50 L, what is the resulting pressure? (*b*) What would the pressure and temperature have been if the process had been isentropic? The isentropic exponent *k* for carbon dioxide is 1.28.

2.19 Helium at 180 kN/m² abs and 20°C is isentropically compressed to one-fifth of its original volume. What is its final pressure?

2.20 The absolute viscosity of a certain gas is 0.0234 cP while its kinematic viscosity is 181 cSt, both measured at 1013 mb abs and 100°C. Calculate its approximate molar mass, and suggest what gas it may be.

2.21 A hydraulic lift of the type commonly used for greasing automobiles consists of a 10.000-in-diameter ram that slides in a 10.006-in-diameter cylinder (Fig. P2.21), the annular space being filled with oil having a kinematic viscosity of 0.0038 ft²/sec and specific gravity of 0.83. If the rate of travel of the ram v is 0.5 fps, find the frictional resistance, F when 6 ft of the ram is engaged in the cylinder.

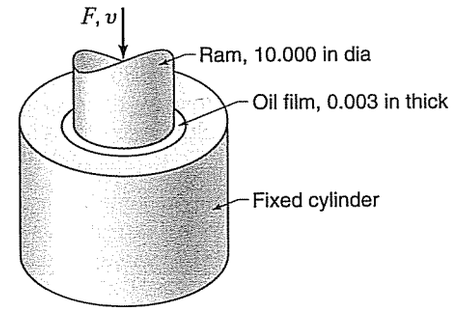


Figure P2.21

2.22 A hydraulic lift of the type commonly used for greasing automobiles consists of a 280.00-mm-diameter ram that slides in a 280.18-mm-diameter cylinder (similar to Fig. P2.21), the annular space being filled with oil having a kinematic viscosity of 0.00042 m²/s and specific gravity of 0.86. If the rate of travel of the ram is 0.22 m/s, find the frictional resistance when 2 m of the ram is engaged in the cylinder.

2.23 A journal bearing consists of an 8.00-in shaft in an 8.01-in sleeve 10 in long, the clearance space (assumed to be uniform) being filled with SAE 30 Eastern lubricating oil at 100°F. Calculate the rate at which heat is generated at the bearing when the shaft turns at 100 rpm. Refer to Appendix A. Express the answer in Btu/hr.

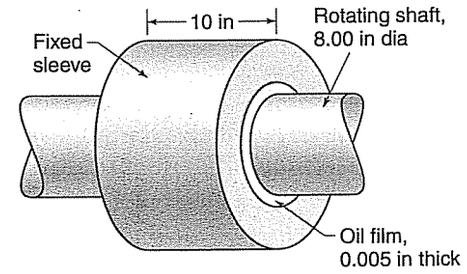


Figure P2.23

2.24 Repeat Prob. 2.23 for the case where the sleeve has a diameter of 8.50 in. Compute as accurately as possible the velocity gradient in the fluid at the shaft and sleeve.

2.25 A disk spins within an oil-filled enclosure, having 2.4-mm clearance from flat surfaces each side of the disk. The disk surface extends from radius 12 to 86 mm. What torque is required to drive the disk at 660 rpm if the oil's absolute viscosity is 0.12 $N \cdot s/m^2$?

2.26 It is desired to apply the general case of Sample Prob. 2.9 to the extreme cases of a journal bearing ($\alpha = 0$) and an end bearing ($\alpha = 90°$). But when $\alpha = 0$, $r = \tan \alpha = 0$, so $T = 0$; when $\alpha = 90°$, contact area $= \infty$ due to b, so $T = \infty$. Therefore devise an alternative general derivation that will also provide solutions to these two extreme cases.

2.27 Some free air at standard sea-level pressure (101.33 kPa abs) and 20°C has been compressed. Its pressure is now 200 kPa abs and its temperature is 20°C. Table A.2 indicates that its kinematic viscosity ν is 15×10^{-6} m^2/s. (*a*) Why is this ν incorrect? (*b*) What is the correct value?

2.28 Some free air at standard sea-level pressure (101.33 kPa abs) and 20°C has been compressed isentropically. Its pressure is now 194.5 kPa abs and its temperature is 80°C. Table A.2 indicates that its kinematic viscosity ν is 20.9×10^{-6} m^2/s. (*a*) Why is this ν incorrect? (*b*) What is the correct value? (*c*) What would the correct value be if the compression were isothermal instead?

2.29 Pure water at 50°F stands in a glass tube of 0.04-in diameter at a height of 6.78 in. Compute the true static height.

2.30 (*a*) Derive an expression for capillary rise (or depression) between two vertical parallel plates. (*b*) How much would you expect 10°C water to rise (in mm) if the clean glass plates are separated by 1.2 mm?

2.31 By how much does the pressure inside a 2-mm-diameter air bubble in 15°C water exceed the pressure in the surrounding water?

2.32 Determine the excess pressure inside an 0.5-in-diameter soap bubble floating in air, given the surface tension of the soap solution is 0.0035 lb/ft.

2.33 Water at 170°F in a beaker is placed within an airtight container. Air is gradually pumped out of the container. What reduction below standard atmospheric pressure of 14.7 psia must be achieved before the water boils?

2.34 At approximately what temperature will water boil on top of Mount Kilimanjaro (elevation 5895 m)? Refer to Appendix A.

CHAPTER 3
Fluid Statics

In fluids at rest there are no shear stresses; hence only normal forces due to pressure are present. Normal forces produced by static fluids are often very important. For example, they tend to overturn concrete dams, burst pressure vessels, and break lock gates on canals. Obviously, to design such facilities, we need to be able to compute the magnitudes and locations of normal pressure forces. Understanding them, we can also develop instruments to measure pressures, and systems that transfer pressures, such as for automobile brakes and hoists.

Note that normal pressure forces alone can occur in a moving fluid if the fluid is moving in bulk without deformation, i.e., as if it were solid or rigid. For such an example, see Sec. 3.10. However, this is relatively rare.

The ***average pressure intensity*** p is the force exerted on a unit area. If F represents the total normal pressure force on some finite area A, while dF represents the force on an infinitesimal area dA, the pressure is

$$p = \frac{dF}{dA} \tag{3.1}$$

If the pressure is uniform over the total area, then $p = F/A$. In the British Gravitational (BG) system we generally express pressure in pounds per square inch (psi) or pounds per square foot (lb/ft^2 = psf), while in SI units we commonly use the pascal (Pa = N/m^2) or kPa (kN/m^2). Previously, bars and millibars were used in metric systems to express pressure; 1 mb = 100 Pa.

3.1 PRESSURE AT A POINT THE SAME IN ALL DIRECTIONS

In a solid, because of the possibility of tangential stresses between adjacent particles, the stresses at a given point may be different in different directions. But no tangential stresses can exist in a fluid at rest, and the only forces between adjacent surfaces are pressure forces normal to the surfaces. Therefore the pressure at any point in a fluid at rest is the same in every direction.

We can prove this by referring to Fig. 3.1, which represents a very small wedge-shaped element of fluid at rest whose thickness perpendicular to the

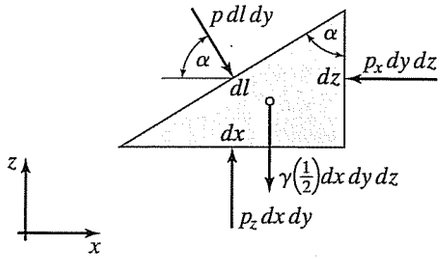


Figure 3.1

plane of the paper is constant and equal to dy. Let p be the average pressure in any direction in the plane of the paper, let α be as shown, and let p_x and p_z be the average pressures in the horizontal and vertical directions.[1] The forces acting on the element of fluid, with the exception of those in the y direction on the two faces parallel to the plane of the paper, are shown in the diagram. For our purpose, forces in the y direction need not be considered because they cancel. Since the fluid is at rest, no tangential forces are involved. As this is a condition of equilibrium, the sum of the force components on the element in any direction must be equal to zero. Writing such an equation for the components in the x direction, $p\,dl\,dy\cos\alpha - p_x dy\,dz = 0$. Since $dz = dl\cos\alpha$, it follows that $p = p_x$. Similarly, summing forces in the z direction gives $p_z dx\,dy - p\,dl\,dy\sin\alpha - \frac{1}{2}\gamma dx\,dy\,dz = 0$. The third term is of higher order than the other two terms and so may be neglected. It follows from this that $p = p_z$. We can also prove that $p = p_y$ by considering a three-dimensional case. The results are independent of α; thus the pressure at any point in a fluid at rest is the same in all directions.

3.2 VARIATION OF PRESSURE IN A STATIC FLUID

Consider the differential element (or control volume) of static fluid shown in Fig. 3.2. Since the element is very small, we can assume that the density of the fluid within the element is constant. Assume that the pressure at the center of the element is p and that the dimensions of the element are δx, δy and δz.[1] The forces acting on the fluid element in the vertical direction are (*a*) the ***body force,*** the action of gravity on the mass within the element, and (*b*) the ***surface forces,*** transmitted from the surrounding fluid and acting at right angles against the top, bottom, and sides of the element. Because the fluid is at rest, the element is in equilibrium and the summation of forces acting on the element in any direction must be zero. If forces are summed in the horizontal direction, that is, x or y, the only forces acting are the pressure forces on the vertical faces of the element. To satisfy $\sum F_x = 0$ and $\sum F_y = 0$, the pressures on the opposite vertical faces must be equal. Thus $\partial p/\partial x = \partial p/\partial y = 0$ for the case of the fluid at rest.

[1] Note that the axes are arranged differently from those usually used in solid mechanics. They are chosen to retain a right-handed coordinate system, and to make z vertical, because z is traditionally used for elevation in fluid mechanics.

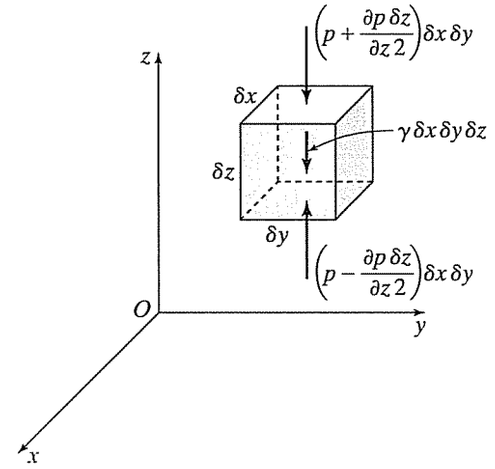


Figure 3.2

Summing forces in the vertical direction and setting the sum equal to zero,

$$\sum F_z = \left(p - \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x\,\delta y - \left(p + \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x\,\delta y - \gamma\,\delta x\,\delta y\,\delta z = 0$$

This results in $\partial p/\partial z = -\gamma$, which, since p is independent of x and y, we can write as

$$\frac{dp}{dz} = -\gamma \tag{3.2}$$

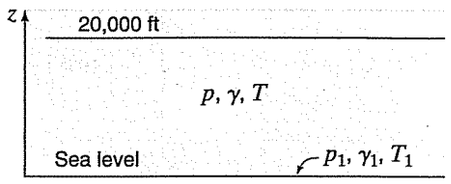
This is the general expression that relates variation of pressure in a static fluid to vertical position. The minus sign indicates that as z gets larger (increasing elevation), the pressure gets smaller.

To evaluate the pressure anywhere in a fluid at rest, we must integrate Eq. (3.2) between appropriately chosen limits. For incompressible fluids (γ = constant), we can integrate Eq. (3.2) directly. For compressible fluids, however, we must express γ algebraically as a function of z or p if we wish to determine pressure accurately as a function of elevation. The variation of pressure in the earth's atmosphere is an important problem, and several approaches are illustrated in the following example.

SAMPLE PROBLEM 3.1 Compute the atmospheric pressure at elevation 20,000 ft, considering the atmosphere as a static fluid. Assume standard atmosphere at sea level. Use four methods; (*a*) air of constant density; (*b*) constant temperature between sea level and 20,000 ft; (*c*) isentropic conditions; (*d*) air temperature decreasing linearly with elevation at the standard lapse rate of 0.00356°F/ft.

Solution

From Appendix A, Table A.3, the conditions of the standard atmosphere at sea level are $T_1 = 59.0°\text{F}$, $p_1 = 14.70$ psia, $\gamma_1 = 0.07648$ lb/ft^3, where subscript 1 indicates conditions at our reference elevation, sea level.



(a) Constant density

From Sec. 3.2: $\dfrac{dp}{dz} = -\gamma; \quad dp = -\gamma\, dz; \quad \displaystyle\int_{p_1}^{p} dp = -\gamma \int_{z_1}^{z} dz$

so $$p - p_1 = -\gamma(z - z_1)$$

and $p = 14.70(144) - 0.07648(20{,}000) = 587$ lb/ft^2 abs $= 4.08$ psia ***ANS***

(b) Isothermal

From Sec. 2.7: $pv =$ constant; so $\dfrac{p}{\gamma} = \dfrac{p_1}{\gamma_1}$ if g is constant

Eq. (3.2): $$\frac{dp}{dz} = -\gamma, \quad \text{where} \quad \gamma = \frac{p\gamma_1}{p_1}$$

so $$\frac{dp}{p} = -\frac{\gamma_1}{p_1}\,dz$$

Integrating, $$\int_{p_1}^{p} \frac{dp}{p} = \ln\frac{p}{p_1} = -\frac{\gamma_1}{p_1}\int_{z_1}^{z} dz = -\frac{\gamma_1}{p_1}(z - z_1)$$

and $$\frac{p}{p_1} = \exp\left[-\left(\frac{\gamma_1}{p_1}\right)(z - z_1)\right]$$

Thus $$p = 14.70 \exp\left[-\frac{0.07648}{14.70(144)}(20{,}000)\right] = 7.14 \text{ psia} \qquad \textit{ANS}$$

(c) Isentropic

From Sec. 2.7: $pv^{1.4} = \dfrac{p}{\rho^{1.4}} =$ constant; so $\dfrac{p}{\gamma^{1.4}} =$ constant $= \dfrac{p}{\gamma_1^{1.4}}$

Eq. (3.2): $$\frac{dp}{dz} = -\gamma, \quad \text{where } \gamma = \gamma_1\left(\frac{p}{p_1}\right)^{1/1.4} = \gamma_1\left(\frac{p}{p_1}\right)^{0.714}$$

so $$dp = -\gamma_1\left(\frac{p}{p_1}\right)^{0.714} dz$$

Integrating:

$$\int_{p_1}^{p} p^{-0.714}\, dp = -\gamma_1 p_1^{-0.714} \int_{z_1}^{z} dz$$

$$p^{0.286} - p_1^{0.286} = -0.286\gamma_1 p_1^{-0.714}(z - z_1)$$

$$p^{0.286} = (14.70 \times 144)^{0.286} - 0.286(0.07648)(14.70 \times 144)^{-0.714}(20{,}000)$$

$$p = 942 \text{ lb/ft}^2 \text{ abs} = 6.54 \text{ psia} \qquad \textit{ANS}$$

(d) Temperature decreasing linearly with elevation

For the standard lapse rate (Fig. 2.2): $T = a + bz$,

where $a = 59.00 + 459.67 = 518.67°\text{R}$ and $b = -0.003560°\text{R/ft}$

Eqs. (3.2) and (2.4): $$\frac{dp}{dz} = -\rho g; \quad \rho = \frac{p}{RT}$$

Combining to eliminate ρ, which varies, rearranging, and substituting for T,

$$\frac{dp}{p} = -\frac{g\, dz}{R(a + bz)}$$

Integrating: $$\int_1^2 \frac{dp}{p} = -\frac{g}{R}\int_1^2 \frac{dz}{a + bz}$$

$$\ln\left(\frac{p_2}{p_1}\right) = -\frac{g}{Rb}\ln\left(\frac{a + bz_2}{a + bz_1}\right) = \ln\left(\frac{a + bz_2}{a + bz_1}\right)^{-g/Rb}$$

i.e., $$\frac{p_2}{p_1} = \left(\frac{a + bz_2}{a + bz_1}\right)^{-g/Rb}$$

Here $$\frac{-g}{Rb} = \frac{-32.174}{1716(-0.003560)} = 5.27$$

and, from Table A.3: $p_1 = 14.696$ psia when $z_1 = 0$.

Thus $$\frac{p_2}{14.696} = \left(\frac{518.67 - 0.003560 \times 20{,}000}{518.67 + 0}\right)^{5.27} = 0.459$$

$$p_2 = 14.696(0.459) = 6.75 \text{ psia} \qquad \textit{ANS}$$

The latter approach corresponds to the standard atmosphere, described in Sec. 2.9 and in Table A.3 of Appendix A.

In Sample Prob. 3.1*a* we saw that, for the case of an *incompressible* fluid,

Incompressible: $$p - p_1 = -\gamma(z - z_1) \qquad (3.3)$$

where p is the pressure at an elevation z. This expression is generally applicable to liquids, since they are only very slightly compressible. Only where there are large changes in elevation, as in the ocean, do we need to consider the

compressibility of the liquid, to arrive at an accurate determination of pressure variation. For small changes in elevation, Eq. (3.3) will give accurate results when applied to gases.

For the case of a liquid at rest, it is convenient to measure distances vertically downward from the free liquid surface. If h is the distance below the free liquid surface and if the pressure of air and vapor on the surface is arbitrarily taken as zero, we can also write Eq. (3.3) as

$$\text{Incompressible:} \qquad p = \gamma h \tag{3.4}$$

In fact, there must always be some pressure on the surface of any liquid, so the total pressure at any depth h is given by Eq. (3.4) plus the pressure on the surface. In many situations this surface pressure may be disregarded, as is pointed out in Sec. 3.4.

From Eq. (3.4), we can see that all points in a connected body of constant density fluid at rest are under the same pressure if they are at the same depth below the liquid surface. This is known as Pascal's law, in honor of Blaise Pascal (1623–1662), a French mathematician who clarified and contributed to early principles of hydrostatics, and after whom we now name the unit of pressure in the SI system. Pascal's law indicates that a surface of equal pressure for a liquid at rest is a horizontal plane. Strictly speaking, it is a surface everywhere normal to the direction of gravity and is approximately a spherical surface concentric with the earth. For practical purposes, a limited portion of this surface may be considered a plane area.

EXERCISES

3.2.1 Neglecting the pressure on the surface and the compressibility of water, what is the pressure in pounds per square inch on the ocean floor at a depth of 15,500 ft? The specific weight of ocean water under ordinary conditions is 64.0 lb/ft^3.

3.2.2 Neglecting the pressure on the surface and the compressibility of water, what is the pressure in kPa at a depth of an instrument 4600 m below the surface of the ocean? The specific weight of ocean water under ordinary conditions is 10.05 kN/m^3.

3.2.3 A pressure gage at elevation 18.0 ft on the side of an industrial tank containing a liquid reads 11.4 psi. Another gage at elevation 12.0 ft reads 13.7 psi. Compute the specific weight, density, and specific gravity of the liquid.

3.2.4 Where an underground oil pipeline crosses under a stream in a gully, it is 68 ft deeper than on either side. When the oil ($s = 0.88$) is not flowing, what is the oil pressure in the line under the stream, if it is 32 psi at each side of the gully?

3.3 PRESSURE EXPRESSED IN HEIGHT OF FLUID

Imagine an open tank of liquid with no pressure acting on its surface (Fig. 3.3), though in reality the minimum pressure upon any liquid surface is the pressure of its own vapor. Disregarding this for the moment, by Eq. (3.4), the pressure at any depth h is $p = \gamma h$. If we assume γ to be constant, there is a definite relation between p and h. That is, pressure (i.e., force per unit area) is equivalent to a

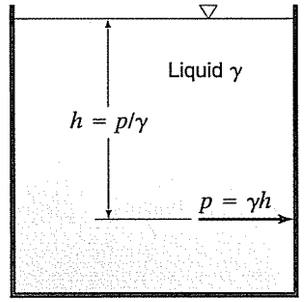


Figure 3.3

height h of some fluid of constant specific weight γ. Often we find it more convenient to express pressure in terms of a height of a column of fluid rather than in pressure per unit area.

Even if the surface of the liquid is under some pressure, we only need to convert this pressure into an equivalent height of the fluid in question and add this to the value of h shown in Fig. 3.3, to obtain the total pressure.

For the preceding discussion we considered a liquid, but, providing it is appropriate, it is equally possible to apply it to a gas or vapor by specifying some *constant* specific weight γ for the gas or vapor in question. Thus we may relate pressure p to the height of a column of *any* fluid by the expression

$$h = \frac{p}{\gamma} \tag{3.5}$$

This relationship is true for any consistent system of units. If p is in pounds per square foot, γ must be in pounds per cubic foot, and then h will be in feet. In SI units, we may express p in kilopascals (kilonewtons per square meter), in which case if γ is in kilonewtons per cubic meter, h will be in meters. When we express pressure in this way, in terms of a height of fluid, we commonly refer to it as ***pressure head*** (see Sec. 5.8). Because we commonly express pressure in pounds per square inch (or kPa in SI units), and since we usually assume the value of γ for water to be 62.4 lb/ft^3 (9.81 kN/m^3), a convenient relationship is

$$h\ (\text{ft of } H_2O) = \frac{144 \times \text{psi}}{62.4} = 2.308 \times \text{psi}$$

or

$$h\ (\text{m of } H_2O) = \frac{\text{kPa}}{9.81} = 0.1020 \times \text{kPa}$$

Often we find it more convenient to express pressures occurring in one fluid in terms of the height of another fluid, e.g., barometric pressure in millimeters of mercury.

An important property follows from Eq. (3.3), which we can express as:

$$\text{Incompressible:} \qquad \frac{p}{\gamma} + z = \frac{p_1}{\gamma} + z_1 = \text{constant} \tag{3.6}$$

This shows that for an incompressible fluid at rest, at any point in the fluid the sum of the elevation z and the pressure head p/γ is equal to the sum of these two quantities at any other point. The significance of this statement is that, in a fluid

at rest, with an increase in elevation there is a decrease in pressure head, and vice versa. This concept is depicted in Fig. 3.4.

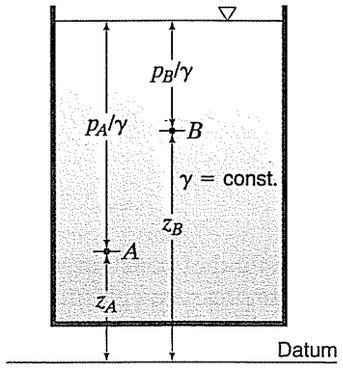
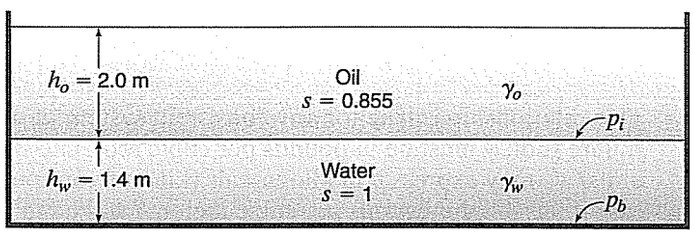


Figure 3.4

SAMPLE PROBLEM 3.2 An open tank contains water 1.40 m deep covered by a 2-m-thick layer of oil ($s = 0.855$). What is the pressure head at the bottom of the tank, in terms of a water column?

Solution 1



From inside cover of book: $\gamma_w = 9.81$ kN/m^3.

Sec. 2.3: $\gamma_o = 0.855(9.81) = 8.39$ kN/m^3

Eq. (3.4) for interface: $p_i = \gamma_o h_o = (8.39)2 = 16.78$ kN/m^2 = 16.78 kPa

Eq. (3.5) for water equivalent of oil:

$$h_{oe} = \frac{p_i}{\gamma_w} = \frac{16.78 \text{ kN/m}^2}{9.81 \text{ kN/m}^3} = 1.710 \text{ m of water}$$

So $h_{we} = h_w + h_{oe} = 1.40 + 1.710 = 3.11$ m of water ***ANS***

Solution 2

From Eq. (3.4) for bottom of tank:

$$p_b = \gamma_o h_o + \gamma_w h_w = (8.39)2 + 9.81(1.4) = 30.51 \text{ kN/m}^2 = 30.51 \text{ kPa}$$

Eq. (3.5) for total water equivalent:

$$h_{we} = \frac{p_b}{\gamma_w} = \frac{30.51 \text{ kN/m}^2}{9.81 \text{ kN/m}^3} = 3.11 \text{ m of water} \quad \textbf{ANS}$$

EXERCISES

3.3.1 An open tank contains 5.0 m of water covered with 2 m of oil ($\gamma = 8.0$ kN/m^3). Find the gage pressure (a) at the interface between the liquids and (b) at the bottom of the tank.

3.3.2 An open tank contains 7 ft of water covered with 2.2 ft of oil ($s = 0.88$). Find the gage pressure (a) at the interface between the liquids and (b) at the bottom of the tank.

3.3.3 If air had a constant specific weight of 12 N/m^3 and were incompressible, what would be the height of air surrounding the earth to produce a pressure at the surface of 101.3 kPa abs?

3.4 ABSOLUTE AND GAGE PRESSURES

If we measure pressure relative to absolute zero, we call it ***absolute*** pressure; when we measure it relative to atmospheric pressure as a base, we call it ***gage*** pressure. This is because practically all pressure gages register zero when open to the atmosphere, and so they measure the difference between the pressure of the fluid they are connected to and that of the surrounding air.

If the pressure is below that of the atmosphere, we call it a ***vacuum,*** and its gage value is the amount by which it is *below* that of the atmosphere. What we call a "high vacuum" is really a low absolute pressure; a perfect vacuum would correspond to absolute zero pressure.

All values of absolute pressure are positive, since a negative value would indicate tension, which we normally consider impossible in any fluid.[2] Gage pressures are positive if they are above that of the atmosphere and negative if they are vacuum (Fig. 3.5).

We can see from the preceding discussion that the following relation holds,

$$p_{\text{abs}} = p_{\text{atm}} + p_{\text{gage}} \tag{3.7}$$

where p_{gage} may be positive or negative (vacuum).

[2] For an exception to this statement, see footnote 11 in Chap. 2.

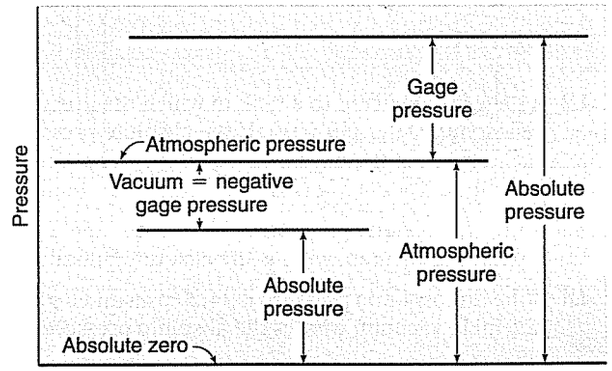


Figure 3.5

We also call the atmospheric pressure the ***barometric*** pressure, and it varies with elevation above sea level (Sec. 2.9). Also, at a given place it varies slightly from time to time because of changes in meteorological conditions.

In thermodynamics it is essential to use absolute pressure, because most thermal properties are functions of the actual (absolute) pressure of the fluid, regardless of the atmospheric pressure. For example, the property relations for a perfect gas (Eq. 2.4) is an equation in which we must use absolute pressure. In fact, we must use absolute pressures in *most* problems involving gases and vapors.

Pressure does not usually much affect the properties of liquids, so we commonly use gage pressures in problems dealing with liquids. Also, we usually find that the atmospheric pressure appears on both sides of an equation, and hence cancels. Thus the value of atmospheric pressure is usually of no significance when dealing with liquids, and, for this reason as well, we almost universally use gage pressures with liquids. About the only situation where we need to consider the absolute pressure of a liquid is where its pressure approaches or equals the saturated vapor pressure (Sec. 2.13). Throughout this text we shall take all numerical pressures to be gage pressures unless they are specifically given as absolute pressures. But whenever confusion is possible, we should specify gage pressures with units like psig or kPa gage.

EXERCISES

3.4.1 A gage is connected to a tank in which the pressure of the fluid is 42 psi above atmospheric (Fig.X3.4.1*a*). If the absolute pressure of the fluid remains unchanged but the gage is in a chamber where the air pressure is reduced to a vacuum of 25 inHg (Fig. X3.4.1*b*), what reading in psi will then be observed?

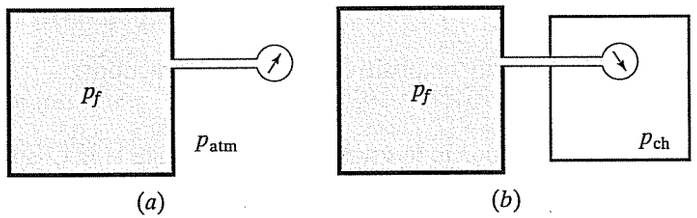


Figure X3.4.1

3.4.2 A gage is connected to a tank in which the pressure of the fluid is 305 kPa above atmospheric (Fig. X3.4.1*a*). If the absolute pressure of the fluid remains unchanged but the gage is in a chamber where the air pressure is reduced to a vacuum of 648 mmHg (Fig. X3.4.1*b*), what reading in kPa will then be observed?

3.4.3 If the atmospheric pressure is 780 mb abs and a gage attached to a tank reads 330 mmHg vacuum, what is the absolute pressure within the tank?

3.4.4 If the atmospheric pressure is 14.20 psia and a gage attached to a tank reads 12.5 inHg vacuum, what is the absolute pressure within the tank?

3.4.5 If the atmospheric pressure is 955 mb abs and a gage attached to a tank reads 190 mmHg vacuum, what is the absolute pressure within the tank?

3.4.6 If the atmospheric pressure is 29.92 inHg, what will be the height of water in a water barometer if the temperature of the water is (*a*) 70°F; (*b*) 120°F? Be as precise as possible.

3.5 MEASUREMENT OF PRESSURE

There are many ways to measure pressure in a fluid. Some of these are discussed in this section.

Barometer

We measure the absolute pressure of the atmosphere with a barometer. If we immerse the open end of a tube such as that in Fig. 3.6*a* in a liquid that is open to the atmosphere (atmospheric pressure), and if we exhaust air from the tube, liquid will rise in the tube. If the tube is long enough and if we have removed all the air, the only pressure on the surface of the liquid in the tube will be that of its own vapor pressure, and the liquid will have reached its maximum possible height.

From the concepts developed in Sec. 3.2, we see that the pressure at O within the tube and at a on the surface of the liquid outside the tube must be the same; that is, $p_O = p_a = p_{atm}$. But, from Eq. (3.4) and Sec. 3.2,

$$p_O = \gamma y + p_{vapor}$$

Because of the static equilibrium, we may equate the pressures at O to obtain

$$p_{atm} = \gamma y + p_{vapor} \tag{3.8}$$

If the vapor pressure on the surface of the liquid in the tube were negligible, then we would have

$$p_{atm} = \gamma y$$

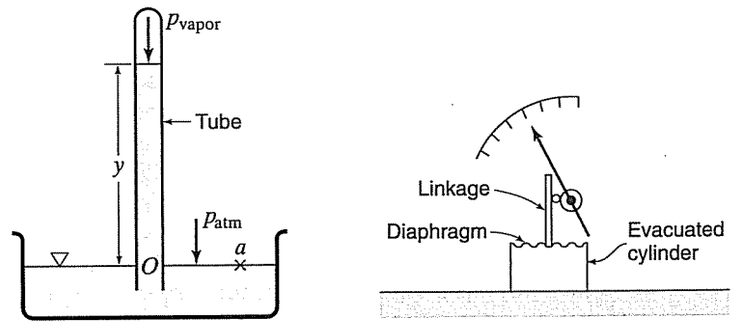


Figure 3.6
Types of barometers.

The liquid used in barometers of this type is usually mercury, because its density is sufficiently great to enable a reasonably short tube to be used, and also because its vapor pressure is negligibly small at ordinary temperatures. If we used some other liquid, the tube would need to be so high as to be inconvenient and its vapor pressure at ordinary temperatures would be appreciable; so a nearly perfect vacuum at the top of the column would not be attainable. Consequently the height attained by the liquid would be less than the true barometric height and we would have to make a correction to the reading. When using a mercury barometer, to get as accurate a measurement of atmospheric pressure as possible, we should make corrections for capillarity and vapor pressure to the reading (Secs. 2.12–2.13).

An ***aneroid*** barometer measures the difference in pressure between the atmosphere and an evacuated cylinder by means of a sensitive elastic diaphragm and linkage system as depicted in Fig. 3.6*b*.

Since we use atmospheric pressure at sea level so widely and often, it is good to keep in mind equivalent forms of expression. By using Eq. (3.5) we find that we can express standard sea-level atmospheric pressure in the following different ways:

$$\begin{aligned} 14.696 \text{ psia } (2116.2 \text{ psfa}) \quad &\text{or} \quad 101.325 \text{ kPa abs } (1013.25 \text{ mb abs}) \\ 29.92 \text{ inHg} \quad &\text{or} \quad 760 \text{ mmHg} \\ 33.91 \text{ ft of water} \quad &\text{or} \quad 10.34 \text{ m of water.} \end{aligned}$$

For convenience, these values are listed on the pages facing the inside covers of the book. For most engineering work, we generally round them to three or four significant figures.

SAMPLE PROBLEM 3.3 What would be the reading on a barometer containing carbon tetrachloride at 68°F at a time when the atmospheric pressure was equivalent to 30.26 inHg?

Solution

$$p_{\text{atm}} = 30.26 \text{ inHg} \times \frac{14.696 \text{ psia}}{29.92 \text{ inHg}} = 14.86 \text{ psia}$$

Table A.4 for carbon tetrachloride at 68°F:

$$\rho = 3.08 \text{ slugs/ft}^3, \qquad p_{\text{vapor}} = 1.90 \text{ psia}$$

From Eq. (3.8):

$$y = \frac{p_{\text{atm}} - p_{\text{vapor}}}{\rho g} = \frac{(14.86 - 1.90)144}{3.08(32.2)} = 18.82 \text{ ft of carbon tetrachloride} \qquad \textit{ANS}$$

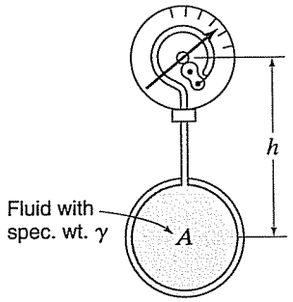


Figure 3.7
Bourdon gage.

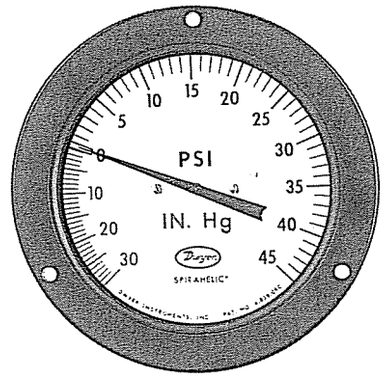


Figure 3.8
Compound pressure and vacuum gage. Pressures in pounds per square inch, vacuums in inches of mercury. (Courtesy of Dwyer Instruments, Inc.)

Bourdon Gage

We commonly measure pressures or vacuums with the ***Bourdon gage*** of Fig. 3.7. In this gage, a curved tube of elliptical cross section changes its curvature with changes in pressure inside the tube; higher pressures tend to "straighten" it. The moving end of the tube rotates a hand on a dial through a linkage system. When a pressure and vacuum gage is combined into one we call this a ***compound gage*** (Fig. 3.8). The pressure indicated by such gages is that at their centers. If the connecting piping is filled completely with fluid of the same density as that in A of Fig. 3.7, and if the pressure gage is graduated to read in pounds per square inch, as is customary, then

$$p_A \text{ (psi)} = \text{gage reading (psi)} + \frac{\gamma h}{144}$$

where γ is expressed in pounds per cubic foot and h in feet.

A vacuum gage, or the negative-pressure portion of a compound gage, is traditionally graduated to read in millimeters or inches of mercury. For vacuums,

$$\text{inHg vacuum at } A = \text{gage reading (inHg vacuum)} - \frac{\gamma h}{144}\left(\frac{29.92}{14.70}\right)$$

Here, once again, we assume that this fluid completely fills the connecting tube of Fig. 3.7. The elevation-correction terms, i.e., those containing h, may be positive or negative, depending on whether the gage is above or below the point where we want to determine the pressure. The expressions given are for the situation depicted in Fig. 3.7. When measuring liquid pressures, the gage is usually set to measure the pressure at the centerline of the pipe. When measuring gas pressures, the elevation correction terms are generally negligible.

These expressions, when written in SI units, require no conversion factors; however, we must take care in dealing with decimal points when adding terms.

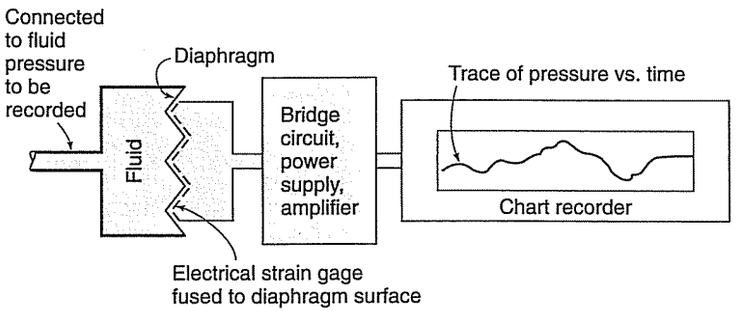


Figure 3.9
Schematic of an electrical strain-gage pressure transducer with a strip-chart recorder.

Pressure Transducer

A ***transducer*** is a device that transfers energy (in any form) from one system to another. A Bourdon gage, for example, is a mechanical transducer in that it has an elastic element that converts energy from the pressure system to a displacement in the mechanical measuring system. An ***electrical pressure transducer*** converts the displacement of a mechanical system (usually a metal diaphragm) to an electric signal, either actively if it generates its own electrical output or passively if it requires an electrical input that it modifies as a function of the mechanical displacement. In one type of pressure transducer (Fig. 3.9) an electrical strain gage is attached to a diaphragm. As the pressure changes, the deflection of the diaphragm changes. This, in turn, changes the electrical output, which, through proper calibration, can provide pressure. If we connect such a device to a strip-chart recorder we can use it to give a continuous record of pressure. Instead of a strip-chart recorder, we may record the data at fixed time intervals on a tape or disk using a computer data acquisition system and/or we may display it on a panel in digital form.

Piezometer Column

A piezometer column is a simple device for measuring moderate pressures of liquids. It consists of a sufficiently long tube (Fig. 3.10) in which the liquid can freely rise without overflowing. The height of the liquid in the tube will give the value of the pressure head, p/γ (Secs. 3.3, 5.9), directly. To reduce capillary error (Sec. 2.12) the tube diameter should be at least 0.5 in (12 mm).

If we wish to measure the pressure of a *flowing* fluid, we should take special precautions in making the connection. The hole must be absolutely normal to the interior surface of the wall, and the piezometer tube or the connection for any other pressure-measuring device must not project beyond the surface. There can be no burrs and surface roughness near the hole, and it is well to round the edge of the hole slightly. Also, the hole should be small, preferably not larger than $\frac{1}{8}$ in (3 mm) diameter.

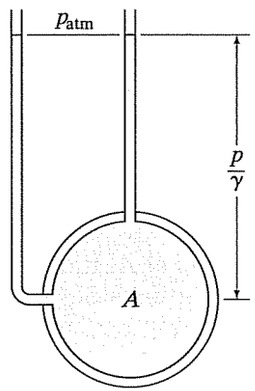


Figure 3.10
Piezometer (for measuring p/γ in liquids only).

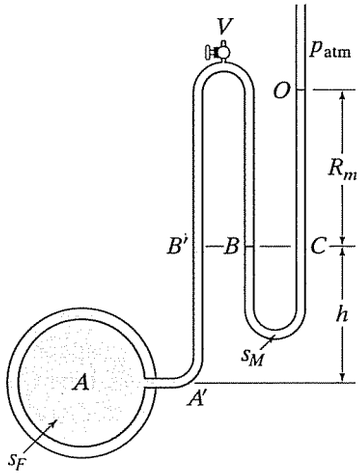


Figure 3.11
Open-end manometer (for measuring p/γ in liquids or gases).

Simple Manometer

Since the open piezometer tube is too tall and cumbersome for use with liquids under high pressure, and it cannot be used with gases, the simple manometer or mercury U tube of Fig. 3.11 is a convenient device for measuring many pressures. To determine the ***gage pressure*** or the ***gage pressure head*** at A, in terms of the liquid at A, we may write a ***gage equation*** based on the fundamental relations of hydrostatic pressures (Eq. 3.3). We can use any units of pressure or pressure head in the gage equation, providing the resulting dimensions of each term are the same. Let us define s_M as the specific gravity of the *manometer* (M) fluid (or gage fluid) and s_F as the specific gravity of the *fluid* (F) whose pressure is being measured. Also, let us identify a manometer reading by R_m; in Fig. 3.11 this is the height OC. If y' is the height of a column of measured fluid (F) that would exert the same pressure at C as does the column of manometer fluid OC, height R_m, then, from Eq. (3.4),

$$\text{gage pressure } p_C = \gamma_M R_m = \gamma_F y'$$

and by rearranging, making use of Sec. 2.3,

$$y' = (\gamma_M/\gamma_F)R_m = (\rho_M/\rho_F)R_m = (s_M/s_F)R_m$$

Thus the gage pressure at C, in terms of the fluid whose pressure we are measuring, as required, is $\gamma(s_M/s_F)R_m$. This is also the pressure at B because the fluid in BC is in balance. The pressure at A is greater than this by γh, assuming the fluid in the connecting tube $A'B$ is of the same specific weight as that of the fluid at A. For this simple case we can write down the pressure at A directly. But for

more complicated gages it is helpful to commence the equation at the open end of the manometer with the pressure there, then proceed through the entire tube to A, adding pressure terms when descending and subtracting them when ascending, all in terms of equivalent pressures of measured fluid (F), finally equating the result to the pressure at A. We can omit portions of the same fluid with the same end elevations, like BC and $B'B$, because they are in balance and so do not affect the pressure at A. Thus, for Fig. 3.11,

$$0 + \gamma\left(\frac{s_M}{s_F}\right)R_m + \gamma h = p_A \tag{3.9a}$$

where γ is the specific weight of the liquid at A. If desired, we can perform the same analysis by expressing the terms in units of head rather than pressure. Then, proceeding from O to A in Fig. 3.11,

$$0 + \left(\frac{s_M}{s_F}\right)R_m + h = \frac{p_A}{\gamma} \tag{3.9b}$$

If we multiply this through by γ ($= \gamma_F$), we see it is the same as Eq. (3.9a).

If we want the absolute pressure or the ***absolute pressure head*** at A then the zero of the first term in Eq. (3.9) must be replaced by the atmospheric pressure (or pressure head) expressed in terms of the fluid whose pressure we are measuring. When measuring the pressure in liquids, an air-relief valve V (Fig. 3.11) will provide a means for the escape of gas should any become trapped in tube $A'B$. If the fluid in A is a gas, the pressure and pressure head contribution from the distance h is generally very small and we can neglect it because of the relatively small specific gravity (or density) of the gas.

When measuring a vacuum, for which we might use the arrangement in Fig. 3.12, the resulting gage equation for pressure head, subject to the same conditions as in the preceding case, is

$$0 - \left(\frac{s_M}{s_F}\right)R_m + h = \frac{p_A}{\gamma} \tag{3.10}$$

Again, it would simplify the equation if we were measuring pressure in a gas, because the h term is then negligible. In measuring vacuums in liquids the arrangement in Fig. 3.13 is advantageous, since gas and vapors cannot become trapped in the tube. For this case,

$$0 - \left(\frac{s_M}{s_F}\right)R_m - h = \frac{p_A}{\gamma}$$

or

$$\frac{p_A}{\gamma} = -\left[h + \left(\frac{s_M}{s_F}\right)R_m\right] \tag{3.11}$$

Although we generally use mercury as the measuring fluid in the simple manometer, we sometimes use other liquids (carbon tetrachloride, for example). As the specific gravity of the measuring fluid approaches that of the fluid

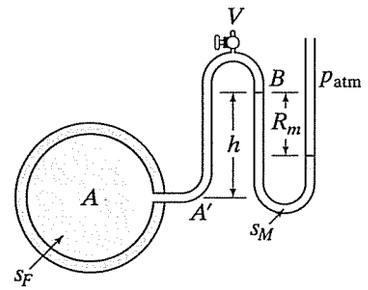


Figure 3.12
Negative-pressure manometer.

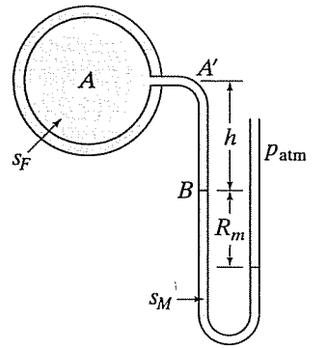


Figure 3.13
Negative-pressure manometer.

whose pressure we are measuring, the reading becomes larger for a given pressure, thus increasing the accuracy of the instrument, providing the two specific gravities are accurately known.

Differential Manometers

In many cases we need to know only the *difference* between two pressures, and for this purpose we can use differential manometers, such as shown in Fig. 3.14. In Fig. 3.14a the measuring fluid has a *greater* density than that of the fluid whose pressure difference we seek. If the fluids in A and B (Fig. 3.14a) have the same density, then, proceeding in a similar manner as before, through the manometer tubing from A to B, we obtain

$$\frac{p_A}{\gamma} - h_A - \left(\frac{s_M}{s_F}\right)R_m + h_B = \frac{p_B}{\gamma}$$

So, by rearranging,

$$\frac{p_A}{\gamma} - \frac{p_B}{\gamma} = h_A - h_B + \left(\frac{s_M}{s_F}\right)R_m$$

But, from Fig. 3.14a,

$$h_A + R_m = h_B + (z_B - z_A)$$

where z represents elevation, so

$$h_A - h_B = (z_B - z_A) - R_m$$

so that

$$\frac{p_A}{\gamma} - \frac{p_B}{\gamma} = (z_B - z_A) - R_m + \left(\frac{s_M}{s_F}\right)R_m = (z_B - z_A) + \left(\frac{s_M}{s_F} - 1\right)R_m \tag{3.12a}$$

Later, we will find it convenient to also write this as

$$\left(\frac{p_A}{\gamma} + z_A\right) - \left(\frac{p_B}{\gamma} + z_B\right) = \Delta\left(\frac{p}{\gamma} + z\right) = \left(\frac{s_M}{s_F} - 1\right)R_m \tag{3.12b}$$

where the left of these equations provides a definition of $\Delta(p/\gamma + z)$.

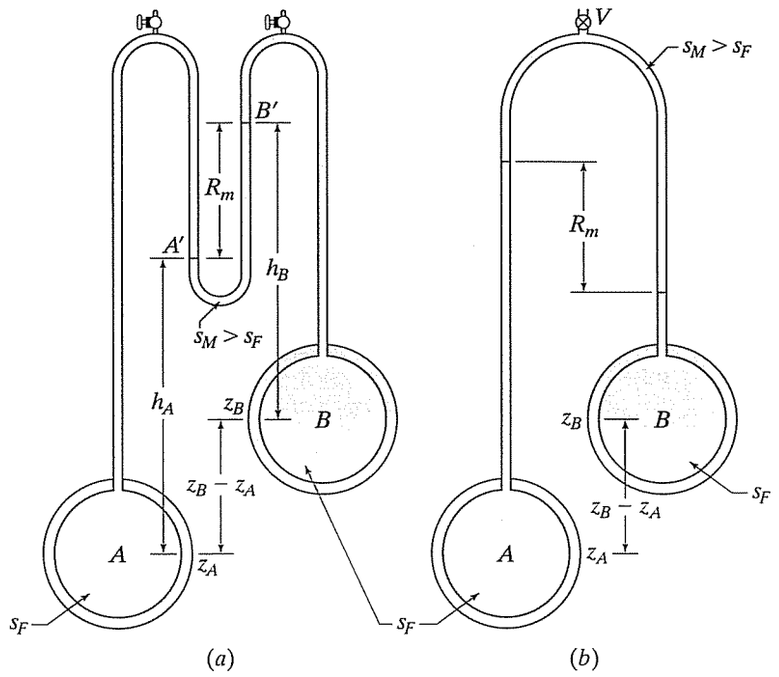


Figure 3.14
Differential manometers. (*a*) For measuring Δp in liquids or gases. (*b*) For measuring Δp in liquids only.

Equation (3.12) is applicable only if the fluids in A and B have the same density. If these densities are different, we can find the pressure head difference by expressing all head components between A and B in terms of one or other of the fluids, as in Sample Prob. 3.4. We must emphasize that by far the most common mistakes made in working differential-manometer problems are to omit the factor $(s_M/s_F - 1)$ for the gage difference R_m, or to omit the -1 from this factor. The term $(s_M/s_F - 1)R_m$ accounts for the *difference* in pressure heads due to the two columns of liquids (M) and (F) of height R_m in the U tube.

The differential manometer, when used with a heavy liquid such as mercury, is suitable for measuring large pressure differences. For a small pressure difference, however, a *light* fluid, such as oil, or even air, is preferable, in which case the manometer is arranged as in Fig. 3.14*b*. Of course, the manometer fluid must be one that will not mix with the fluid in A or B. By the same method of analysis as above, we can show for Fig. 3.14*b* that, for identical liquids in A and B,

$$\frac{p_A}{\gamma} - \frac{p_B}{\gamma} = (z_B - z_A) + \left(1 - \frac{s_M}{s_F}\right)R_m \qquad (3.13a)$$

or

$$\Delta\left(\frac{p}{\gamma} + z\right) = \left(1 - \frac{s_M}{s_F}\right)R_m \qquad (3.13b)$$

Here s_M/s_F, the ratio of the specific gravities (or densities or specific weights), has a value less than one. As the density of the manometer fluid approaches that of the fluid being measured, $(1 - s_M/s_F)$ approaches zero, and we will obtain larger values of R_m for small pressure differences, thus increasing the sensitivity of the gage. Once again, we must modify the equation if the densities of fluids A and B are different.

To determine pressure difference between liquids, we often use air or some other gas as the measuring fluid, with the manometer arrangement of Fig. 3.14*b*. Air can be pumped through valve V until the pressure is sufficient to bring the two liquid columns to suitable levels. Any change in pressure raises or lowers both liquid columns by the same amount, so that the difference between them is constant. In this case the value of s_M/s_F can be considered to be zero, since the density of gas is so much less than that of a liquid.

Another way to obtain increased sensitivity is simply to incline the gage tube so that a vertical gage difference R_m is transposed into a reading that is magnified by $1/\sin\alpha$, where α is the angle of inclination with the horizontal.

SAMPLE PROBLEM 3.4 In Fig. S3.4 liquid A weighs 53.5 lb/ft^3 (8.4 kN/m^3) and liquid B weighs 78.8 lb/ft^3 (12.4 kN/m^3). Manometer liquid M is mercury. If the pressure at B is 30 psi (207 kPa), find the pressure at A. Express all pressure heads in terms of the liquid in bulb B.

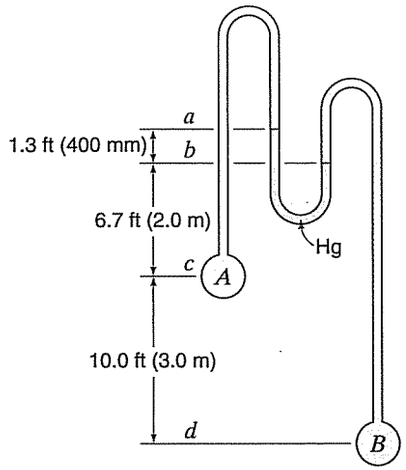


Figure S3.4

Solution
Proceeding from A to B:

$$\frac{p_A}{\gamma_B} - (z_a - z_c)\frac{\gamma_A}{\gamma_B} + (z_a - z_b)\frac{\gamma_M}{\gamma_B} + (z_b - z_d)\frac{\gamma_B}{\gamma_B} = \frac{p_B}{\gamma_B}$$

BG units:

$$\frac{p_A}{\gamma_B} - 8.0\frac{53.5}{78.8} + 1.3\frac{13.56(62.4)}{78.8} + 16.7 = \frac{p_B}{\gamma_B}$$

$$\frac{p_A}{\gamma_B} - 5.43 + 13.96 + 16.7 = \frac{30(144)}{78.8} = 54.8 \text{ ft}$$

$$\frac{p_A}{\gamma_B} = 29.6 \text{ ft} \qquad p_A = 29.6\frac{78.8}{144} = 16.19 \text{ psi} \qquad \textbf{ANS}$$

SI units:

$$\frac{p_A}{\gamma_B} - 2.4\frac{8.4}{12.4} + 0.4\frac{13.56(9.81)}{12.4} + 5.0 = \frac{p_B}{\gamma_B}$$

$$\frac{p_A}{\gamma_B} - 1.626 + 4.29 + 5.00 = \frac{207 \text{ kN/m}^2}{12.4 \text{ kN/m}^3} = 16.69 \text{ m}$$

$$\frac{p_A}{\gamma_B} = 9.03 \text{ m}, \qquad p_A = 9.03(12.4) = 112.0 \text{ kN/m}^2 = 112.0 \text{ kPa} \qquad \textbf{ANS}$$

EXERCISES

3.5.1 If the atmospheric pressure is equivalent to 33.40 ft of water, what must be the reading (to 0.01 ft) on a barometer containing an alcohol ($s = 0.78$) if the vapor pressure of the alcohol at the temperature of observation is 2.09 psia?

3.5.2 A scientist plans to build a water barometer. When the atmospheric pressure is 990 mb abs and the water temperature is 70°C, what would you expect the barometer reading (water rise) to be?

3.5.3 In Sample Prob. 3.4 suppose the atmospheric pressure is 1028 mb abs. What must be the absolute pressure at A? Express in mb abs and in mHg.

3.5.4 In Fig. X3.5.4, originally the manometer reading $R_m = 4$ in when $h = 5$ ft. Atmospheric pressure is 14.70 psia. If the absolute pressure at A is doubled, what will be the manometer reading?

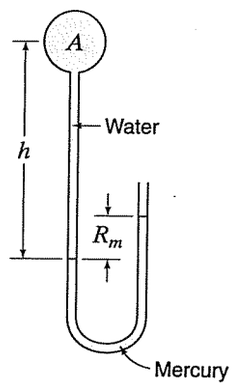


Figure X3.5.4

3.5.5 Gas confined in a rigid container exerts a pressure of 25 psi when its temperature is 40°F. What pressure would the gas exert if the temperature were raised to 165°F? Barometric pressure remains constant at 29.0 inHg.

3.5.6 Gas confined in a rigid container exerts a pressure of 200 kPa when its temperature is 5°C. What pressure would the gas exert if the temperature were raised to 80°C? Barometric pressure remains constant at 29.0 inHg.

3.5.7 (a) A mercury manometer (Fig. 3.11) is connected to a pipeline carrying water at 150°F and located in a room where the temperature is also 150°F. If the elevation of point B is 6 ft above A and the mercury manometer reading is 48 in, what is the pressure in the pipe in psi? Be as precise as possible, and note the effect of temperature. Note that at 150°F the specific gravity of mercury is 13.45. (b) Repeat, assuming all temperatures are 68°F.

3.5.8 In Fig. X3.5.8, atmospheric pressure is 14.80 psia; the gage reading at A is 3.7 psi; the vapor pressure of the alcohol is 1.4 psia. Compute x and y.

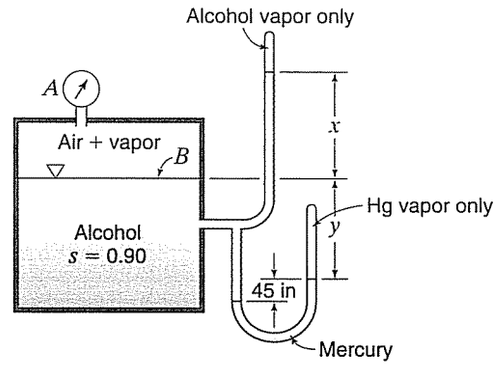


Figure X3.5.8

3.5.9 Refer to the manometer of Fig. 3.14b. A and B are at the same elevation. Water is contained in A and rises in the tube to a level 52 in above A. Kerosene is contained in B. The inverted U tube is filled with air at 11 psi and 70°F. Atmospheric pressure is 14.70 psia. (a) Determine the difference in pressure between A and B if the manometer reading is 12 in. Express the answer in psi. (b) What is the absolute pressure in B in inches of mercury, and feet of kerosene?

3.5.10 (a) Two vessels are connected to a differential manometer using mercury ($s = 13.56$), the connecting tubing being filled with water. The higher-pressure vessel is 5 ft lower in elevation than the other. Room temperature prevails. If the mercury reading is 4.0 in, what is the pressure difference in feet of water, and in psi? (b) If carbon tetrachloride ($s = 1.59$) were used instead of mercury, what would the manometer reading be for the same pressure difference?

3.5.11 (a) Two vessels are connected to a differential manometer using mercury ($s = 13.56$), the connecting tubing being filled with water. The higher-pressure vessel is 1.5 m lower in elevation than the other. Room temperature prevails. If the mercury reading is 100 mm, what is the pressure difference in m of water, and in kPa? (b) If carbon tetrachloride ($s = 1.59$) were used instead of mercury, what would the manometer reading be for the same pressure difference?

3.6 FORCE ON A PLANE AREA

As we noted previously in Sec. 3.1, no tangential force can exist within a fluid at rest. All forces are then normal to the surfaces in question. If the pressure is uniformly distributed over an area, the force is equal to the pressure times the area, and the point of application of the force is at the centroid of the area. For submerged horizontal areas, the pressure is uniform. In the case of compressible fluids (gases), the pressure variation with vertical distance is very small because of the low specific weight; therefore, *when we compute the static fluid force exerted by a gas, we usually treat p as a constant.* Thus, for such cases,

$$F = \int p\,dA = p\int dA = pA \tag{3.14}$$

In the case of liquids the distribution of pressure is generally not uniform, so further analysis is necessary. Let us consider a vertical plane whose upper edge lies in the free surface of a liquid (Fig. 3.15). Let this plane be perpendicular to the plane of the figure, so that MN is merely its trace, or edge. The gage pressure will vary from zero at M to NK at N. The total force on one side of the plane is the sum of the products of the elementary areas and the pressure upon them. From the pressure distribution, we can see that the resultant of this system of parallel forces must act at a point *below* the centroid of the area, since the centroid of an area is the point where the resultant of a system of *uniform* parallel forces would act.

If we lower the plane to position $M'N'$, the *proportionate* change of pressure from M' to N' is less than it was from M to N. Hence the resultant pressure force will act nearer to the centroid of the plane surface. The deeper we

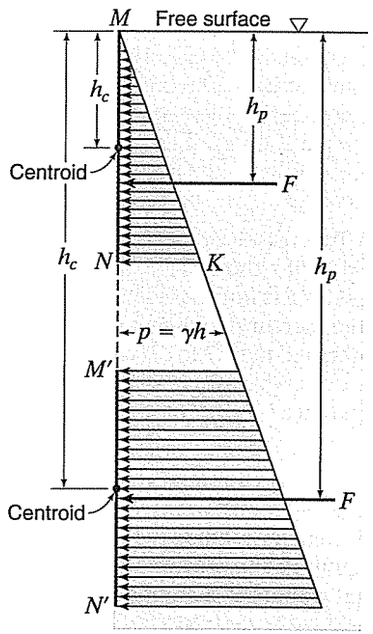


Figure 3.15
Pressure distributions on two vertical plane areas (viewed from edges).

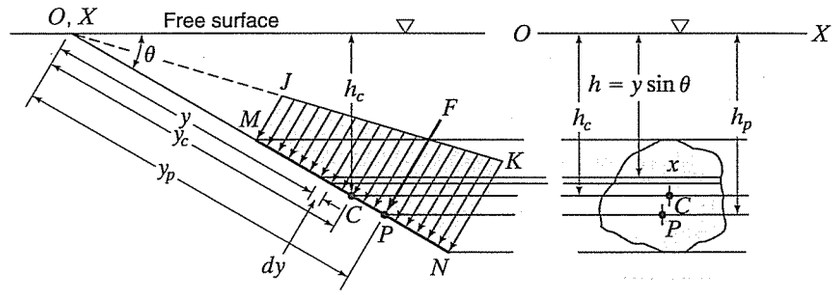


Figure 3.16
Pressure distribution on a sloping plane area (viewed from edge). C is centroid, P is center of pressure. Sloping y distances correspond to vertical h distances.

submerge the plane, the smaller the proportional pressure variation becomes, and the closer the resultant moves to the centroid.

In Fig. 3.16 let MN be the edge of a plane area making an angle θ with the horizontal. To the right we see the projection of this area onto a vertical plane. The pressure distribution over the sloping area forms a ***pressure prism*** ($MNKJ$ times width in Fig. 3.16), whose volume is equal to the total force F acting on the area. If the width x is constant then we can easily compute the volume of the pressure prism, using a mean pressure $= 0.5(MJ + NK)$, and so obtain F.

If x varies, we must integrate to find F. Let h be the variable depth to any point and let y be the corresponding distance from OX, the intersection of the plane containing the area and the free surface.

Choose an element of area so that the pressure over it is uniform. Such an element is a horizontal strip, of width x, so $dA = x\,dy$. As $p = \gamma h$ and $h = y\sin\theta$, the force dF on the horizontal strip is

$$dF = p\,dA = \gamma h\,dA = \gamma y \sin\theta\,dA$$

Integrating,
$$F = \int dF = \gamma\sin\theta\int y\,dA = \gamma\sin\theta\, y_c A \tag{3.15}$$

where y_c is, by definition, the distance from OX along the sloping plane to the centroid C of the area A. If h_c is the vertical depth to the centroid, then $h_c = y_c\sin\theta$, and in general we have

$$F = \gamma h_c A \tag{3.16}$$

Thus we find the total force on any plane area submerged in a liquid by multiplying the specific weight of the liquid by the product of the area and the depth of its centroid. The value of F is independent of the angle of inclination of the plane so long as the depth of its centroid is unchanged.[3]

[3] For a plane submerged as in Fig. 3.16, it is obvious that Eq. (3.16) applies to one side only. As the pressure forces on the two sides are identical but opposite in direction, the net force on the plane is zero. In most practical cases where the thickness of the plane is not negligible, the pressures on the two sides are not the same.

Since γh_c is the pressure at the centroid, we can also say that the total force on any plane area submerged in a liquid is the product of the area and the pressure at its centroid.

3.7 CENTER OF PRESSURE

The point of application of the resultant pressure force on a submerged area is called the ***center of pressure.*** We need to know its location whenever we wish to work with the moment of this force.

The most general way of looking at the problem of forces on a submerged plane area is through the use of the recently discussed ***pressure prism concept*** (Sec. 3.6 and Fig. 3.16). The line of action of the resultant pressure force must pass through the centroid of the pressure prism (volume). As noted earlier, this concept is very convenient to apply for simple areas such as rectangles. For example, if the submerged area in Fig. 3.15 is of constant width then we know that the centroid of the pressure prism on area MN is $\frac{2}{3}MN$ below M.

If the shape of the area is not so regular, i.e., if the width x in Fig. 3.16 varies, then we must take moments and integrate. Taking OX in Fig. 3.16 as an axis of moments, the moment of an elementary force $dF = \gamma y \sin\theta\, dA$ is

$$y\,dF = \gamma y^2 \sin\theta\, dA$$

and if y_p denotes the distance to the center of pressure, using the basic concept that the moment of the resultant force equals the sum of the moments of the component forces,

$$y_p F = \gamma \sin\theta \int y^2\, dA = \gamma \sin\theta\, I_o$$

where we recognize that I_o is the moment of inertia of the plane area about axis OX.

If we divide this last expression by the value of F given by Eq. (3.15), we obtain

$$y_p = \frac{\gamma \sin\theta\, I_o}{\gamma \sin\theta\, y_c A} = \frac{I_o}{y_c A} \tag{3.17}$$

The product $y_c A$ is the static moment of area A about OX. Therefore Eq. (3.17) tells us that we can obtain the distance from the center of pressure to the axis where the plane (extended) intersects the liquid surface by dividing the moment of inertia of the area A about the surface axis by its static moment about the same axis.

We may also express this in another form, by noting from the parallel axis theorem that

$$I_o = A y_c^2 + I_c$$

where I_c is the moment of inertia of an area about its centroidal axis. By substituting for I_o into Eq. (3.17),

$$y_p = \frac{A y_c^2 + I_c}{y_c A}$$

so

$$y_p = y_c + \frac{I_c}{y_c A} \tag{3.18}$$

From this equation, we again see that the location of the center of pressure P is independent of the angle θ; that is, we can rotate the plane area about axis OX without affecting the location of P. Also, we see that P is always *below* the centroid C and that, as the depth of immersion is increased, y_c increases and therefore P approaches C.

For convenient reference, Table A.7 of Appendix A contains values of y_c and I_c for a variety of area shapes.

We can find the lateral position of the center of pressure P by considering that the area is made up of a series of elemental horizontal strips. The center of pressure for each strip is at the midpoint of the strip. Since the moment of the resultant force F must be equal to the moment of the distributed force system about any axis, say, the y axis,

$$X_p F = \int x_p p\, dA \tag{3.19}$$

where X_p is the lateral distance from the selected y axis to the center of pressure P of the resultant force F, and x_p is the lateral distance to the center of any elemental horizontal strip of area dA on which the pressure is p.

SAMPLE PROBLEM 3.5 Figure S3.5 represents a gate, 2 ft wide perpendicular to the sketch. It is pivoted at hinge H. The gate weighs 500 lb. Its center of gravity is 1.2 ft to the right of and 0.9 ft above H. For what values of water depth x above H will the gate remain closed? Neglect friction at the pivot and neglect the thickness of the gate.

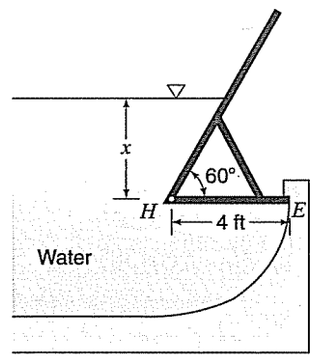


Figure S3.5

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Solution

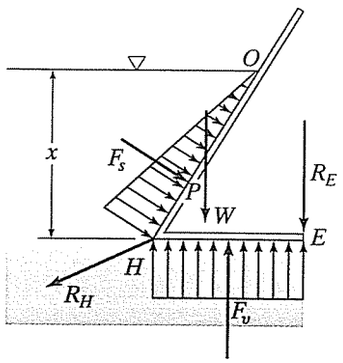
In addition to the reactive forces R_H at the hinge and R_E at end E, there are three forces acting on the gate: its weight W, the vertical hydrostatic force F_v upward on the rectangular bottom of the gate, and the slanting hydrostatic force F_s acting at right angles to the sloping rectangular portion of the gate. The magnitudes of the latter three forces are:

Given: $$W = 500 \text{ lb}$$

Eq. (3.16): $$F_v = \gamma h_c A = \gamma(x)(4 \times 2) = 8\gamma x$$

Eq. (3.16): $$F_s = \gamma h_c A = \gamma\left(\frac{x}{2}\right)\left(\frac{x}{\cos 30°} \times 2\right) = 1.155\gamma x^2$$

A diagram showing these three forces is as follows:



The moment arms of W and F_v with respect to H are 1.2 ft and 2.0 ft, respectively. The moment arm of F_s gets larger as the water depth increases because the location of the center of pressure changes. We can find the location of the center of pressure of F_s from Eq. (3.18):

$$y_p = y_c + \frac{I_c}{y_c A}, \quad \text{where, from Table A.7, } I_c = \frac{bh^3}{12}$$

with $h = x/\cos 30°$ and $y_c = 0.5h$. So

$$y_p = \frac{0.5x}{\cos 30°} + \frac{(1/12)2(x/\cos 30°)^3}{(0.5x/\cos 30°)[2(x/\cos 30°)]}$$

i.e., for F_s: $$OP = y_p = 0.577x + \frac{2x}{12 \cos 30°} = 0.770x$$

Hence the moment arm of F_s with respect to H is $PH = x/\cos 30° - 0.770x = 0.385x$. [*Note:* In this case we need not use Eq. (3.18) to find the lever arm of F_s because we know the line of action of F_s for the triangular distributed load on the rectangular area is at the one-third point between H and O, i.e., $HP = (1/3)(x/\cos 30°) = 0.385x$.]

When the gate is about to open (incipient rotation), $R_E = 0$ and the sum of the moments of all forces about H is zero, viz

$$\sum M = P_s(0.385x) - F_v(2.0) + W(1.2) = 0$$

i.e., $$1.155\gamma x^2(0.385x) - 8\gamma x(2) + 500(1.2) = 0$$

Substituting $\gamma = 62.4 \text{ lb/ft}^3$ gives

$$27.73x^3 - 998.4x + 600 = 0$$

This is a cubic, polynomial equation (Appendix B). With a polynomial solver, available on some hand calculators (see Appendix B and Appendix C.1), we may find the three roots directly. With an equation solver, available on some scientific calculators (Appendixes C.1 and D.1) and in some spreadsheets and mathematics software (Appendixes C.2-3 and D.2-3), we may obtain the root closest to a guessed value we provide.

Without any of these aids, we can solve this equation by trials ("trial and error"), seeking an x value that makes the left side of the equation equal to zero. After two trials, we can use linear interpolation (or extrapolation) to estimate the next, better trial value. We then repeat this until x is sufficiently accurate, e.g., accurate to three significant figures after rounding:

Trial x	*Left side*
0.1	500.2
0.5	104.3
0.6	6.95
0.61	−2.73
0.607	0.173

We can find the other two roots by more, similar, trials. We could use a spreadsheet to facilitate such trials. But, more conveniently, dividing the cubic by $(x - 0.607)$ yields a quadratic (Eq. B.6) from which we can easily find that the other two roots (Eq. B.7) are $x = 5.67$ and -6.28.

Thus $x = 0.607$ ft or 5.67 ft or a negative (meaningless) root. Therefore, from inspection of the moment equation, the gate will remain closed when $0.607 \text{ ft} < x < 5.67 \text{ ft}$. ***ANS***

Note: Sections D.1–D.3 of Appendix D include complete example input and output for solutions to this problem using an HP 48G programmable calculator, and using Excel (spreadsheet) and Mathcad (mathematics software).

SAMPLE PROBLEM 3.6 The cubic tank shown in Fig. S3.6 is half full of water. Find (*a*) the pressure on the bottom of the tank, (*b*) the force exerted by the fluids on a tank wall, and (*c*) the location of the center of pressure on a wall.

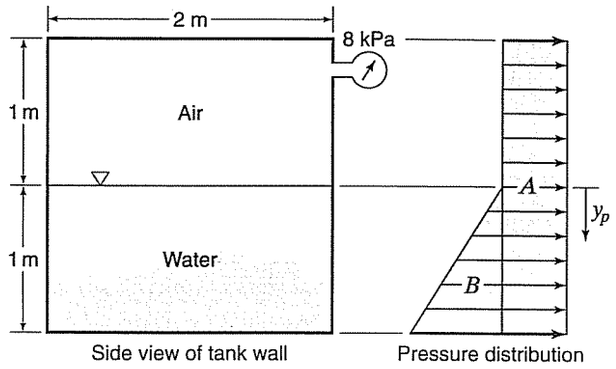


Figure S3.6

Solution

(*a*) $$p_{bott} = p_{air} + \gamma_{water} h_{water} = 8\ \text{kN/m}^2 + (9.81\ \text{kN/m}^3)(1\ \text{m})$$
$$= 17.81\ \text{kN/m}^2 = 17.81\ \text{kPa} \qquad \textit{ANS}$$

(*b*) The force acting on the tank end is divided into two components, labeled A and B on the pressure distribution sketch. Component A has a *uniform* pressure distribution, due to the pressure of the confined air, which acts throughout the water:

$$F_A = p_{air} A_{air} = (8\ \text{kN/m}^2)(4\ \text{m}^2) = 32.0\ \text{kN}$$

For component B, i.e., the varying water pressure distribution on the lower half of the tank wall, the centroid C of the area of application is at

$$h_c = y_c = 0.5(1\ \text{m}) = 0.5\ \text{m below the water top surface,}$$

so, from Eq. (3.16),

$$F_B = \gamma_{water} h_c A_{water} = 9.81(0.5)2 = 9.81\ \text{kN}$$

So the total force on the tank wall is

$$F = F_A + F_B = 32.0 + 9.81 = 41.8\ \text{kN} \qquad \textit{ANS}$$

(*c*) The locations of the centers of pressure of the component forces, as distances y_p below the water top surface, are

$$(y_p)_A = 0\ \text{m}$$

below the water top surface, to the centroid of the 2-m-square area for the uniform air pressure.

$$(y_p)_B = \tfrac{2}{3} h_{water} = \tfrac{2}{3}(1\ \text{m}) = 0.667\ \text{m}$$

below the water top surface for the varying pressure on the rectangular wetted wall area. We could also find this using Eq. (3.18) with $y_c = 0.5$ m, $I_c = bh^3/12 = 2(1)^3/12 = 0.1667\ \text{m}^4$, and $A = bh = 2\ \text{m}^2$.

Taking moments: $F(y_p) = F_A(y_p)_A + F_B(y_p)_B$

from which $y_p = 0.1565$ m below the water top surface ***ANS***

SAMPLE PROBLEM 3.7 Water and oil in an open storage tank are in contact with the end wall as shown in Fig. S3.7. (*a*) Find the pressure at the bottom (lowest point) of the tank caused by the liquids. Also find (*b*) the total force exerted on the end wall by the liquids, and (*c*) the depth of its center of pressure.

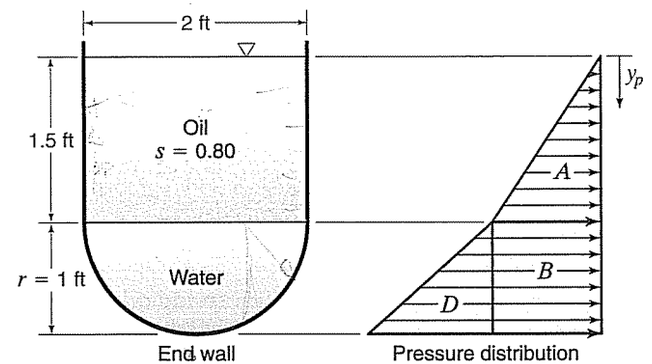


Figure S3.7

Solution

(*a*) $$p_{bott} = \gamma_{oil} h_{oil} + \gamma_{water} h_{water}$$
$$= (0.8 \times 62.4\ \text{lb/ft}^3)(1.5\ \text{ft}) + (62.4\ \text{lb/ft}^3)(1.5\ \text{ft})$$
$$= 137.3\ \text{lb/ft}^3 = 0.953\ \text{psi} \qquad \textit{ANS}$$

(*b*) The force acting on the end consists of three components, labeled A, B, and D, on the pressure distribution sketch. Note that component B has a *uniform* pressure distribution, due to the oil (A) above, which acts throughout the liquid below.

As a preliminary, we note for the semicircular end area ($r = 1$ ft) that

(i) $A = \pi r^2/2 = \pi 1^2/2 = 1.571\ \text{ft}^2$;

(ii) from Appendix A, Table A.7, the centroid is $4r/3\pi = 0.424$ ft from the center of the circle, i.e., below the water top surface.

For component A, i.e., the varying oil pressure distribution on the 1.5-ft height of the end wall, the centroid C of the area of application is at

$$h_c = y_c = 0.5(1.5\ \text{ft}) = 0.75\ \text{ft below the free oil surface,}$$

so, from Eq. (3.16),

$$F_A = \gamma_{oil} h_c A_{oil} = (0.8 \times 62.4)0.75(1.5 \times 2) = 112.3 \text{ lb}$$

For component B, the force F_B on the water-wetted area of the end wall due to the uniform pressure produced by the 1.5-ft depth of oil above is

$$F_B = pA = \gamma hA = (0.8 \times 62.4)1.5(\pi 1^2/2) = 117.6 \text{ lb}$$

For component D, i.e., the varying pressure distribution due to the water (only) on the water-wetted area of the end wall, the centroid C is at

$$h_c = y_c = 0.424 \text{ ft below the water top surface,}$$

so:

$$F_D = \gamma h_c A = 62.4(0.424)\pi 1^2/2 = 41.6 \text{ lb}$$

The total force F on the end of the tank is therefore

$$F = F_A + F_B + F_D = 272 \text{ lb} \qquad \textit{ANS}$$

(*c*) As a preliminary to locating the center of pressure, we note that for the *semi*circular end area with $D = 2$ ft,

(i) from Table A.7: I about the center of the circle, is $I = \pi D^4/128 = \pi 2^4/128 = 0.393 \text{ ft}^4$, and

(ii) by the parallel axis theorem: I_c about the centroid, distance 0.424 ft below the center of the circle, is $I_c = I + Ad^2 = 0.393 + (\pi 1^2/2)(0.424)^2 = 0.1098 \text{ ft}^4$.

The locations of the centers of pressure, below the free oil surface, of the component forces are:

$(y_p)_A = \frac{2}{3}(1.5 \text{ ft}) = 1.000$ ft for the varying oil pressure on the oil-wetted area, and $(y_p)_B = y_c = 1.5 + 0.424 = 1.924$ ft to the *centroid* of the water-wetted semicircular area, for the uniform pressure on this area due to 1.5 ft of oil above the water; and

Eq. (3.18): $$(y_p)_D = y_c + \frac{I_c}{y_c A} = 0.424 + \frac{0.1098}{0.424(\pi 1^2/2)} = 0.589 \text{ ft}$$

below the water top surface, for the varying water pressure on the water-wetted semicircular area,

$$= 1.5 + 0.589 = 2.09 \text{ ft below the free oil surface.}$$

Finally,

$$Fy_p = F_A(y_p)_A + F_B(y_p)_B + F_D(y_p)_D$$

$$y_p = 1.567 \text{ ft} \qquad \textit{ANS}$$

EXERCISES

3.7.1 A circular area of diameter d is vertical and submerged in a liquid. Its upper edge is coincident with the liquid surface. Derive an expression for the depth to its center of pressure.

3.7.2 If a triangle of height d and base b is vertical and submerged in a liquid with its base at the liquid surface, derive an expression for the depth to its center of pressure.

3.7.3 If a triangle of height d and base b is vertical and submerged in liquid with its vertex at the liquid surface, derive an expression for the depth to its center of pressure.

3.7.4 Repeat Exer. 3.7.3 for the same triangle but with its vertex a distance a below the liquid surface.

3.7.5 A vertical right-triangle of height d and base b submerged in liquid has its vertex at the liquid surface. Find the distance from the vertical side to the center of pressure by (*a*) inspection; (*b*) calculus.

3.7.6 A plane surface is circular with a diameter of 2 m. If it is vertical and the top edge is 0.5 m below the water surface, find the magnitude of the force on one side and the depth to the center of pressure.

3.7.7 Find the magnitude and depth of the point of application of the force on the circular gate shown in Fig. X3.7.7 if $h = 5$ ft and $D = 4$ ft dia.

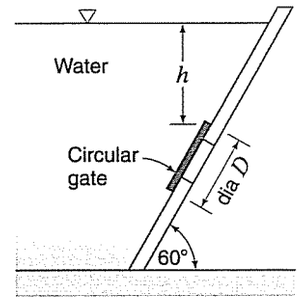


Figure X3.7.7

3.7.8 A rectangular plate 5 ft by 4 ft is at an angle of 30° with the horizontal, and the 5-ft side is horizontal. Find the magnitude of the force on one side of the plate and the depth of its center of pressure when the top edge is (*a*) at the water surface; (*b*) 1 ft below the water surface.

3.7.9 In Fig. X3.7.9 the rectangular flashboard MN shown in cross section ($a = 5.4$ m) is pivoted at B. (*a*) What must be the maximum height of B above N if the flashboard is on the verge of tipping when the water surface rises to M? (*b*) If the flashboard is pivoted at the location determined in (*a*) and the water surface is 1 m below M, what are the reactions at B and N per m length of board perpendicular to the figure?

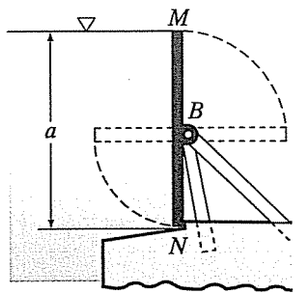


Figure X3.7.9

3.7.10 The gate MN in Fig. X3.7.10 rotates about an axis through N. If $a = 3.3$ ft, $b = 1.3$ ft, $d = 2$ ft, and the width perpendicular to the plane of the figure is 3 ft, what torque applied to the shaft through N is required to hold the gate closed?

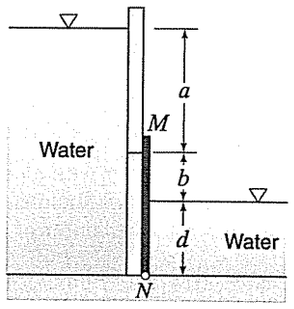


Figure X3.7.10

3.7.11 What minimum value of b in Fig. X3.7.11 is necessary to keep the rectangular masonry wall from sliding if it weighs 160 lb/ft^3, $a = 14$ ft, $c = 16$ ft, and the coefficient of friction is 0.45? With this minimum b value, will it also be safe against overturning? Assume that water does not get underneath the block.

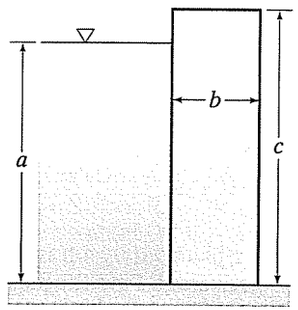


Figure X3.7.11

3.7.12 A rectangular plate submerged in water is 5 m by 4 m, the 5-m side being horizontal and the 4-m side being vertical. Determine the magnitude of the force on one side of the plate and the depth to its center of pressure if the top edge is (a) at the water surface; (b) 1 m below the water surface; (c) 100 m below the water surface.

3.7.13 The right-triangular plate shown in Fig. X3.7.13 is submerged in a vertical plane with its base horizontal. Determine the depth and horizontal position of the center of pressure when $a = 1$ ft, $b = 3$ ft, and $d = 4.5$ ft.

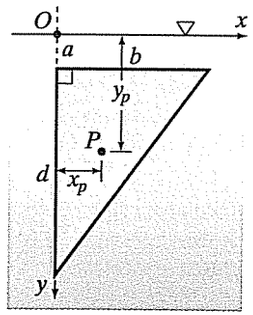


Figure X3.7.13

3.7.14 Repeat Exer. 3.7.13, but with $a = 0.2$ m, $b = 1.0$ m, and $d = 2.0$ m.

3.7.15 A rectangular area is 5 m by 6 m, with the 5 m side horizontal. It is placed with its centroid 4 m below a water surface and rotated about a horizontal axis in the plane area and through its centroid. Find the magnitude of the force on one side and the distance between the center of pressure and the centroid of the plane when the angle with the horizontal, $\theta = 90$, 60, 30, and 0°.

3.7.16 Figure X3.7.16 shows a cylindrical tank with 0.25-in-thick walls, containing water. What is the force on the bottom? What is the force on the annular surface MM? What is the weight of the water? Find the longitudinal (vertical) tensile stress in the sidewalls BB if (a) the tank is suspended from the top; (b) it is supported on the bottom. Neglect the weight of the tank.

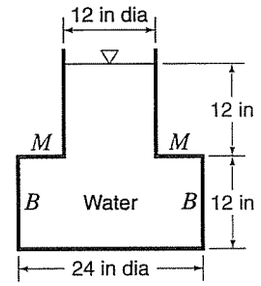


Figure X3.7.16

3.8 FORCE ON A CURVED SURFACE

On any curved or warped surface such as MN in Fig. 3.17a, the force on the various elementary areas that make up the curved surface are different in direction and magnitude, so an algebraic summation is impossible. Hence we can apply Eq. (3.16) only to a plane area. But for nonplanar areas, we can find component forces in certain directions, and often without integration.

Horizontal Force on Curved Surface

We may project any irregular curved area MN (Fig. 3.17a) onto a vertical plane whose trace is $M'N'$ (Fig. 3.17b). The projecting elements, which are all horizontal, enclose a volume whose ends are the vertical plane $M'N'$ and the irregular area MN. This volume of liquid is in static equilibrium. A force F' acts on the projected vertical area $M'N'$. The horizontal force component F_x' acts on the irregular end area MN and is equal and opposite to the F_x of Fig. 3.17a. Gravity force W' is vertical, and the lateral forces on all the horizontal projection elements are normal to these elements and hence normal to F'. Thus the only horizontal forces on $MNN'M'$ are F' and F_x', and therefore

$$F' - F_x' = 0$$

and

$$F_x = F_x' = F' \tag{3.20}$$

Hence the horizontal force in any given direction on any area is equal to the force on the *projection* of that area onto a vertical plane normal to the given

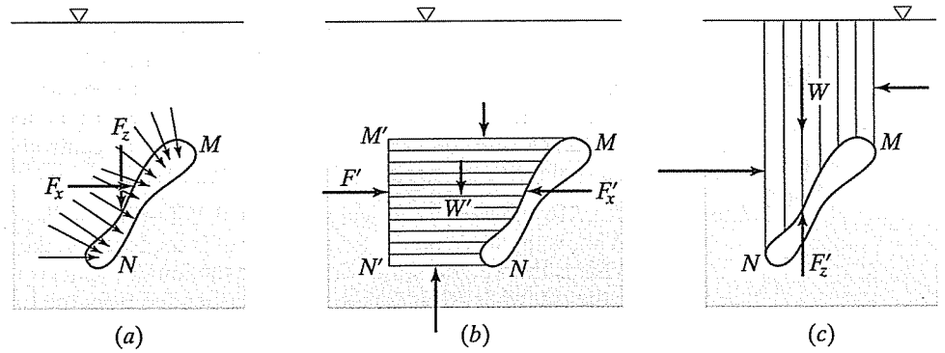


Figure 3.17
Hydrostatic forces on curved surfaces.

direction. The line of action of F_x must be the same as that of F'. Equation (3.20) applies to gases as well as liquids. In the case of a gas the horizontal force on a curved surface is equal to the pressure multiplied by the projection of that area onto a vertical plane normal to the force.

Vertical Force on a Curved Surface

We can find the vertical force F_z on a curved or warped area, such as MN in Fig. 3.17*a*, by considering the volume of liquid enclosed by the area and vertical elements extending to the free surface (Fig. 3.17*c*). This volume of liquid is in static equilibrium. The only vertical forces on this volume of liquid are the force $F_G = p_G A$ due to any gas (at pressure p_G) above the liquid, the gravity force W downward, and F_z', the upward vertical force on the irregular area MN. The force F_z', (Fig. 3.17*c*) is equal and opposite to the force F_z (Fig. 3.17*a*). Any other forces on the vertical elements are normal to the elements, and so are horizontal. Therefore

$$F_z' - W - F_G = 0$$

and

$$F_z = F_z' = W + F_G \tag{3.21}$$

Therefore the vertical force acting on any area is equal to the weight of the volume of liquid above it, plus any superimposed gas pressure force. The line of action of F_z is the resultant of F_G and W. F_G must pass through the centroid of the plan (surface projection) area, and W must pass through the center of gravity of the liquid volume. The portion of this volume above $M'M$ (Fig. 3.17*b*) has a regular shape with volume equal to height times projected plan area, and has its centroid beneath the centroid of the plan area; the other portion, below $M'M$ and above the curved surface MN, may have a difficult shape and so may require integration to find its volume and centroid. If only a gas is involved, the procedure is similar, but is much simpler because W is negligible.

For the case where a force acts on the lower side of the surface but not on the upper side, the vertical force component is the same in magnitude as that given by Eq. (3.21) but opposite in sense.

Resultant Force on a Curved Surface

In general, there is no single resultant force on an irregular area, because the horizontal and vertical forces, as found in the above discussion, may not be in the same plane. But in certain cases these two forces will lie in the same plane and then we can combine them into a single force.

SAMPLE PROBLEM 3.8 Find the horizontal and vertical components of the force exerted by the fluids on the horizontal cylinder in Fig. S3.8 if (*a*) the fluid to the left of the cylinder is a gas confined in a closed tank at a pressure of 35.0 kPa; (*b*) the fluid to the left of the cylinder is water with a free surface at an elevation coincident with the uppermost part of the cylinder. Assume in both cases that atmospheric pressure occurs to the right of the cylinder.

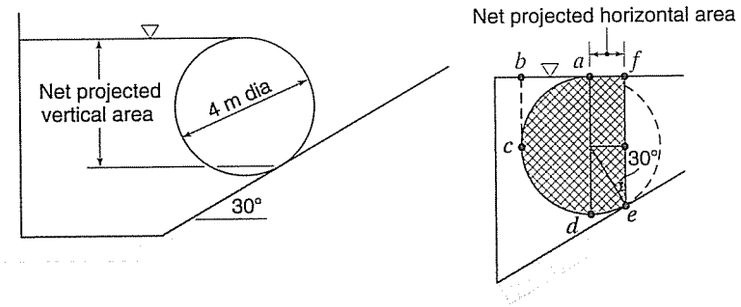


Figure S3.8

Solution

The net projection on a vertical plane of the portion of the cylindrical surface under consideration (see left-hand diagram) is, from the right-hand diagram, $ef = 2 + 2\cos 30° = 3.73$ m.

(*a*) For the gas,

$$F_x = pA_2 = 35.0 \text{ kN/m}^2(3.73 \text{ m}) = 130.5 \text{ kN/m to the right} \qquad \textit{ANS}$$

The vertical force of the gas on the surface *ac* is equal and opposite to that on the surface *cd*. Hence the net projection on a horizontal plane for the gas is $af = 2 \sin 30° = 1$ m. Thus

$$F_z = pA_x = 35.0 \text{ kN/m}^2(1 \text{ m}) = 35.0 \text{ kN/m upward} \qquad \textit{ANS}$$

(*b*) For the fluid,

Eq. (3.16):
$$F_x = \gamma h_c A = 9.81 \text{ kN/m}^2(\tfrac{1}{2} \times 3.73 \text{ m})(3.73 \text{ m})$$
$$= 68.3 \text{ kN/m to the right} \qquad \textit{ANS}$$

Net F_z = upward force on surface *cde* − downward force on surface *ca*
= weight of volume *abcdefa* − weight of volume *abca*
= weight of cross-hatched volume of liquid
$= 9.81 \text{ kN/m}^3[\tfrac{210}{360}\pi 2^2 + \tfrac{1}{2}(1 \times 2\cos 30°) + (1 \times 2)] \text{ m}^2$
= 100.0 kN/m upward *ANS*

EXERCISES

3.8.1 A vertical-thrust bearing for a large hydraulic gate consists of a 9-in-radius bronze hemisphere mating into a steel hemispherical shell in the gate bottom. What minimum oil pressure will maintain a complete oil film if the vertical thrust on the bearing is 600,000 lb?

3.8.2 The cross section of a tank is as shown in Fig. X3.8.2. BC is a cylindrical surface with $r = 6$ ft, and $h = 10$ ft. If the tank contains gas at a pressure of 8 psi, determine the magnitude and location of the horizontal- and vertical-force components acting on unit width of tank wall ABC.

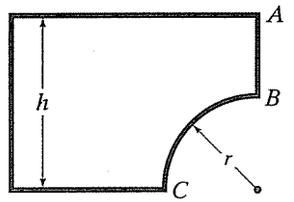


Figure X3.8.2

3.8.3 Find the answers called for in Exer. 3.8.2 if $r = 2$ m, $h = 3.5$ m, and the tank contains gas at a pressure of 50 kPa.

3.8.4 A spherical steel tank of 15-m diameter contains gas under a pressure of 350 kPa. The tank consists of two half-spheres joined together with a weld. What will be the tensile force across the weld in kN/m? If the steel is 20.0 mm thick, what is the tensile stress in the steel? Express in kPa and in psi. Neglect the effects of cross-bracing and stiffeners.

3.8.5 Determine the force F required to hold the cone in the position shown in Fig. X3.8.5. Assume the cone is weightless.

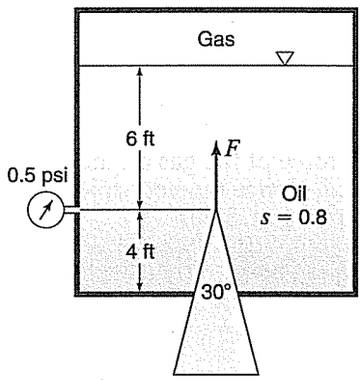


Figure X3.8.5

3.8.6 The hemispherical body shown in Fig. X3.8.6 ($r = 2$ ft) projects into a tank. Find the horizontal and vertical forces acting on the hemispherical projection for the following cases: (*a*) the tank is full of water with the free surface 5 ft above A; (*b*) the tank contains CCl_4 ($s = 1.59$) to the level of A overlain with water having its free surface 5 ft above A; (*c*) the tank is closed and contains only gas at a pressure of 6 psi; (*d*) the tank is closed and contains water to the level of A overlain with gas at a pressure of 2 psi. Assume the gas weighs 0.075 lb/ft^3.

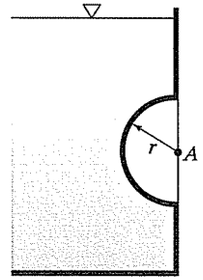


Figure X3.8.6

3.8.7 Repeat Exer. 3.8.2 where the tank is open at the top and contains water to a depth $h = 10$ ft.

3.8.8 Repeat Exer. 3.8.2 where $r = 2$ m, and the tank is open at the top and contains water to a depth $h = 3.5$ m.

3.8.9 A tank with vertical ends contains water and is 6 m long normal to the plane of Fig. X3.8.9. The sketch shows a portion of its cross section where MN is one-quarter of an ellipse with semiaxes b and d. If $a = 1.0$ m, $b = 2.5$ m, and $d = 4$ m, find, for the surface represented by MN, the magnitude and position of the line of action of (*a*) the horizontal component of force; (*b*) the vertical component of force; (*c*) the resultant force and its direction relative to the horizontal.

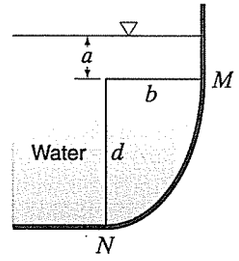


Figure X3.8.9

3.8.10 Find the answers called for in Exer. 3.8.9 if $a = 2$ ft, $b = 6$ ft, $d = 9$ ft, the tank is 12 ft long, and MN represents a parabola with vertex at N.

3.9 BUOYANCY AND STABILITY OF SUBMERGED AND FLOATING BODIES

Submerged Body

When a body such as $DHCK$ in Fig. 3.18 is immersed in a fluid, the forces acting on it are gravity and the pressures of the surrounding fluid. On its upper surface the vertical component of the force is F_z and is equal to the weight of the volume of fluid $ABCHD$. In a similar manner, the vertical component of force on the undersurface is F_z' and is equal to the weight of the volume of fluid $ABCKD$. The difference between these two volumes is the volume $\mathcal{V}$ of the body $DHCK$.

Buoyancy. Let us denote the buoyant force of a fluid by F_B, and observe that it is vertically upward and equal to $F_z' - F_z$, which is equal to the weight of the

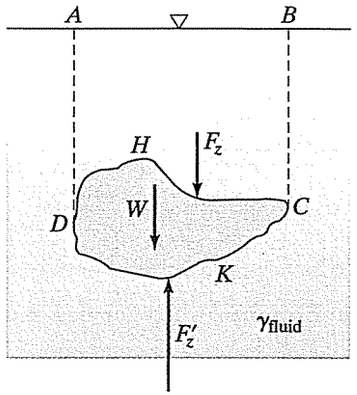


Figure 3.18

volume of fluid *DHCK*. That is, *the* ***buoyant force*** *on any body is equal to the weight of fluid displaced,* or in equation form,

$$F_B = \gamma_{\text{fluid}} \forall$$

This is probably the best-known discovery of Archimedes (287–212 B.C.), a Greek philosopher acclaimed as the father of hydrostatics, and one of the earliest known pioneers of fluid mechanics.

If the body in Fig. 3.18 is in equilibrium, W is equal and opposite to F_B, which means that the densities of the body and the fluid are equal. If W is greater than F_B, the body will sink. If W is less than F_B, the body will rise until its density and that of the fluid are equal, as in the case of a balloon in the air or, in the case of a liquid with a free surface, the body will rise to the surface until the weight of the displaced liquid equals the weight of the body. If the body is less compressible than the fluid, there is a definite level at which it will reach equilibrium. If it is more compressible than the fluid, it will rise indefinitely, provided the fluid has no definite upper limit.

Stability. When we give a body in equilibrium a slight angular displacement (tilt or ***list***), a horizontal distance a then separates W and F_B, which in combination create moments that tend to rotate the body, as we can see in Fig. 3.19. If the

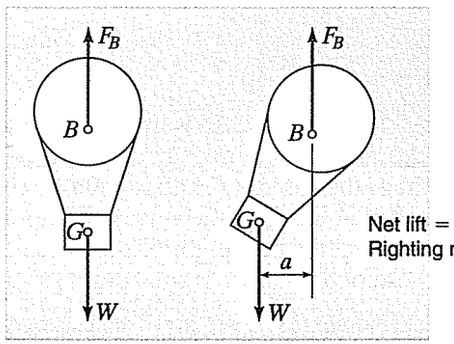


Figure 3.19
Submerged body (balloon).

moments tend to restore the body to its original position, the *lesser* of the two moments is called the ***righting moment*** (Fig. 3.19), and we say the body is in ***stable equilibrium.*** The stability of submerged or floating bodies depends on the relative positions of the buoyant force and the weight of the body. The buoyant force acts through the ***center of buoyancy*** B, which corresponds to the center of gravity of the displaced fluid. *The criterion for stability of a fully submerged body* (balloon or submarine, etc.) *is that the center of buoyancy is above the center of gravity of the body.* From Fig. 3.19 we can see that if B were initially below G, the center of gravity, then the moment created by a tilt would tend to increase the displacement.

Floating Body

For a body in a liquid with a free surface, if its weight W is less than that of the same volume of liquid, it will rise and float on the surface as in Fig. 3.20, so that $W = F_B$. The forces then acting on body $AHBK$ are gravity and the pressures of the fluids in contact with it. The vertical component of force on the undersurface is F'_z and this is equal to the weight of the volume of liquid AKB. This volume is the volume of liquid displaced by the body.

Buoyancy. The buoyant force F_B is vertically upward and equal to F'_z. So, just as for a fully submerged body, the buoyant force acting on a floating body is equal to the weight of liquid displaced. Thus a *floating body displaces a volume of liquid equivalent to its weight.* For equilibrium, the two forces W and F_B must be equal and opposite, and must lie in the same vertical line.

The atmospheric pressure is transmitted through the liquid to act equally on all surfaces of the body. As a result, it has zero net effect. Any buoyancy due to the weight of air displaced by the portion of the body above the liquid surface is usually negligible in comparison with the weight of liquid displaced.

A practical application of the buoyancy principle is the ***hydrometer,*** an instrument we use to measure the specific gravity of liquids. It has a thin, uniform stem of constant cross-sectional area, say A. Weights make it float upright as in Fig. 3.21*a*, with a reference mark that is at the water surface when floating in pure water ($s = 1.0$). When floating in a denser liquid of specific gravity s (Fig. 3.21*b*), the volume of liquid displaced is smaller, so less is submerged and the reference mark is some height Δh above the water surface. If the submerged volume in pure water is $\forall$, then in the denser liquid it is $\forall - A\Delta h$, and the

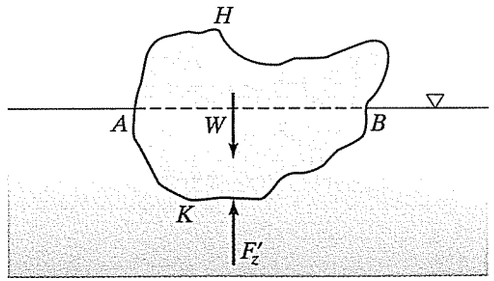
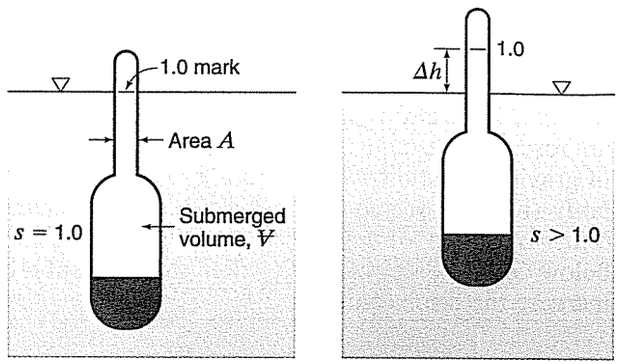


Figure 3.20



(*a*) In pure water (*b*) In denser liquid

Figure 3.21
Hydrometer floating in two different liquids.

hydrometer's weight

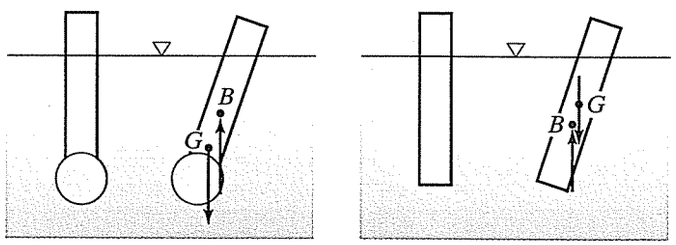
$$W = \gamma_w \forall = (s\gamma_w)(\forall - A\Delta h)$$

from which

$$\Delta h = \frac{\forall}{A}\left(\frac{s-1}{s}\right) \tag{3.22}$$

Using Eq. (3.22) we can calculate the spacing for a specific gravity scale on the stem.

Stability. If a righting moment develops when a floating body lists, the body will be stable regardless of whether the center of buoyancy is above or below the center of gravity. Examples of stable and unstable floating bodies are shown in Fig. 3.22. In these examples the stable body is the one where the center of buoyancy B is above the center of gravity G (Fig. 3.22*a*), and the unstable body has B below G (Fig. 3.22*b*). However, for floating bodies note that the location of B below G does not guarantee instability as it does for submerged bodies, discussed previously. This is because the position of the center of buoyancy B can move relative to a floating body as it tilts, due to its shape, whereas for a fully submerged body the position of B is fixed relative to the body. Figure 3.23 illustrates this point; from these cross sections through the hull of a ship we can see that it is stable even though B is below G. Because of the cross-sectional shape, as the ship



(*a*) Stable (*b*) Unstable

Figure 3.22

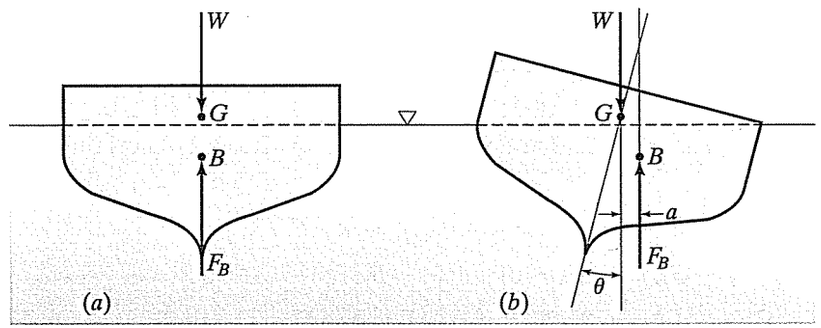


Figure 3.23

tilts to the right (Fig. 3.23*b*) the center of gravity of the displaced water (i.e., B) moves to the right further than the line of action of the body weight W, and so the buoyancy provides a righting moment $F_B \times a$. Clearly, therefore, the stabilities of many floating bodies (those with B below G) depend upon their shapes.

If liquid in the hull of a ship is not constrained, the center of mass of the floating body will move toward the center of buoyancy when the ship rolls, thus decreasing the righting couple and the stability. For this reason, floating vessels usually store liquid ballast or fuel oil in tanks or bulkheaded compartments.

SAMPLE PROBLEM 3.9 The pontoon shown in Fig. S3.9 is 15 ft long, 9 ft wide, and 4 ft high, and is built of uniform material, $\gamma = 45$ lb/ft^3. (*a*) How much of it is submerged when floating in water? (*b*) If it is tilted about its long axis by an applied couple (no net force), to an angle of 12°, what will be the moment of the righting couple?

Solution

(*a*) Floating level, let d = the depth of submergence. Then

$$W = F_B; \quad 15(9)4(45) = 15(9)d(62.4); \quad d = 2.885 \text{ ft} \quad \textit{ANS}$$

(*b*) At 12° tilt, let AD be the water line (see Fig. S3.9).

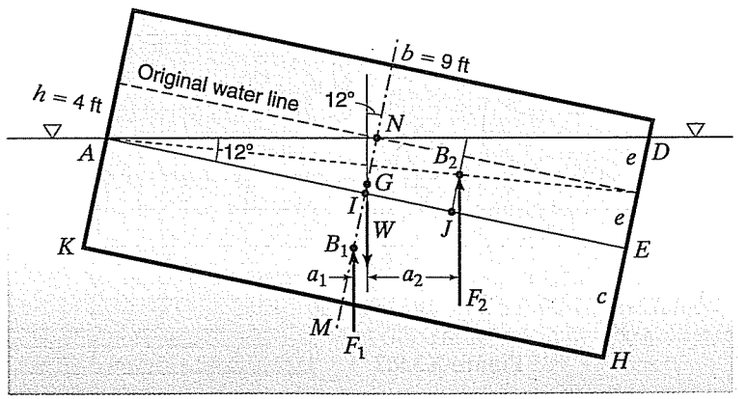


Figure S3.9

Divide the buoyancy force into two components B_1 and B_2, due to the rectangular block $AEHK$ and the triangular prism ADE of displaced water, respectively.

$$DE = 2e = b\tan 12° = 9\tan 12° = 1.913 \text{ ft}; \quad NI = e = 0.957 \text{ ft}$$

As there is no net force, $MN = d = 2.885$ ft. Therefore

$$c = IM = MN - NI = 2.885 - 0.957 = 1.928 \text{ ft}$$

B_1 is at the centroid of the block $AEHK$, so

$$GB_1 = \tfrac{1}{2}(h - c) = \tfrac{1}{2}(4 - 1.928) = 1.036 \text{ ft}; \quad a_1 = GB_1 \sin 12° = 0.215 \text{ ft}$$
$$F_1 = \gamma Lbc = 45(15)9(1.928) = 11{,}710 \text{ lb}$$

B_2 is at the centroid of the triangle ADE, so

$$JE = b/3, \quad IJ = b/6 = 1.5 \text{ ft}, \quad B_2J = \tfrac{2}{3}e = 0.638 \text{ ft}$$

G is at the centroid of the major rectangle, so $MG = h/2 = 2$ ft,

$$GI = MG - MI = MG - c = 2 - 1.928 = 0.0719 \text{ ft}$$
$$a_2 = IJ\cos 12° + (B_2J - GI)\sin 12° = 1.585 \text{ ft}$$
$$F_2 = \gamma Lbe = 45(15)9(0.957) = 5810 \text{ lb}$$

Counterclockwise moments about G:

$$\text{Righting moment} = F_2a_2 - F_1a_1 = 5810(1.585) - 11{,}710(0.215)$$
$$= 6690 \text{ lb·ft} \quad \textbf{\textit{ANS}}$$

EXERCISES

3.9.1 A balloon weighs 160 lb and has a volume of 7200 ft^3. It is filled with helium, which weighs 0.0112 lb/ft^3 at the temperature and pressure of the air, which in turn weighs 0.0807 lb/ft^3. What load will the balloon support, or what force in a cable would be required to keep it from rising?

3.9.2 For the conditions shown in Fig. X3.9.2, find the force F required to lift the concrete-block gate if the concrete weighs 23.6 kN/m^3. Neglect friction.

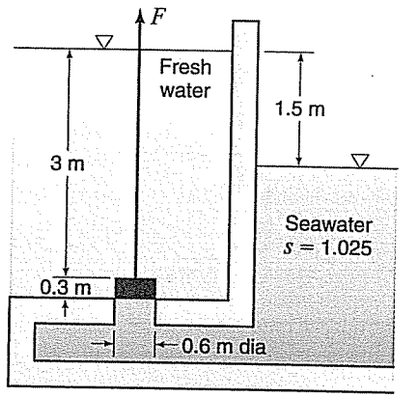


Figure X3.9.2

3.9.3 An iceberg in the ocean floats with one-eighth of its volume above the surface. What is its specific gravity relative to ocean water, which weighs 64 lb/ft^3? What portion of its volume would be above the surface if the ice were floating in pure water?

3.9.4 Determine the volume of an object that weighs 200 N in water and 300 N in oil ($s = 0.88$). What is the specific weight of the object?

3.9.5 An 8-in-diameter solid cylinder 3 in high weighing 3.4 lb is immersed in liquid ($\gamma = 52$ lb/ft^3) contained in a tall, upright metal cylinder of 9 in inside diameter (Fig. X3.9.5). Before immersion, the liquid was 3 in deep ($= x + z$). At what level will the solid cylinder float? Find the distance z between the bottoms of the two cylinders.

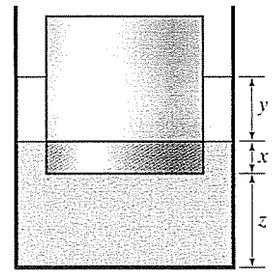


Figure X3.9.5

3.9.6 A metal block 1.5 ft square and 1 ft deep is floated on a body of liquid consisting of a 10-in-layer of water above a layer of mercury. The block metal weighs 120 lb/ft^3. (*a*) What is the position of the bottom of the block? (*b*) If a downward vertical force of 600 lb now acts on the center of this block, what is the new position of the bottom of the block? Assume that the tank containing the fluid is of infinite dimensions.

3.9.7 Two spheres, each of 1.5 m diameter, weigh 8 and 24 kN, respectively. They are connected with a short rope and placed in water. (*a*) What is the tension in the rope and what portion of the lighter sphere's volume protrudes from the water? (*b*) What should be the weight of the heavier sphere in order for the lighter sphere to float halfway out of the water? Assume that the sphere volumes remain constant.

3.9.8 A hydrometer (Fig. 3.22*a*) consists of a 6-mm-diameter cylinder of length 180 mm attached to a 20-mm-diameter weighted sphere. The cylinder has a mass of 0.6 g and the mass of the sphere is 6.4 g. At what level will this device float in liquids having specific gravities 0.8, 1.0, and 1.2? Is the scale spacing on the cylindrical stem uniform? Why or why not?

3.9.9 A cylindrical bucket of 250 mm diameter and 400 mm high weighing 20.0 N contains oil ($s = 0.80$) to a depth of 180 mm. (*a*) When placed to float in water, what will be the immersion depth to the bottom of the bucket? (*b*) What is the maximum volume of oil the bucket can hold and still float?

3.9.10 End D of an 8-ft-long, uniformly thin wooden rod ($s = 0.7$) is held 1 ft below the surface of still water. (*a*) How much of the rod remains above the water surface? (*b*) If the rod diameter is 1 in, what force at D is required to hold it in place?

3.9.11 A solid, half-cylinder-shaped log, of 1.50 ft radius and 10 ft long, floats in water with the flat face up (Fig. X3.9.11). (*a*) If the draft (immersion depth of the lowest point) is 0.90 ft, what is the uniform specific weight of the log? (*b*) The log tilts about its axis (zero net applied force) by less than 23°. Is it in stable equilibrium? Justify your answer with a sketch and logic. (*c*) If the log tilts by 20° (right side down; zero net applied force), what is the magnitude and sense of any moment that results?

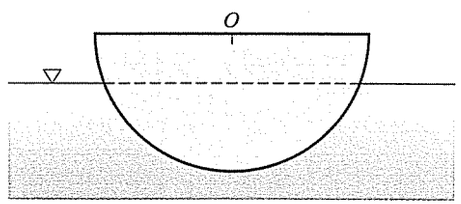


Figure X3.9.11

3.9.12 A solid, half-cylinder-shaped log, of 0.48 m radius and 2.5 m long, floats in water with the flat face up (see Fig. X3.9.11). (*a*) If the draft (immersion depth of the lowest point) is 0.30 m, what is the uniform specific weight of the log? (*b*) The log tilts about its axis (zero net applied force) by less than 22°. Is it in stable equilibrium? Justify your answer with a sketch and logic. (*c*) If the log tilts by 18° (left side down; zero net applied force), what is the magnitude and sense of any moment that results?

3.10 LIQUID MASSES SUBJECTED TO ACCELERATION

Under certain conditions there may be no relative motion between the particles of a liquid mass yet the mass itself may be in motion. If a body of liquid in a tank is transported at a uniform velocity, the conditions are those of ordinary fluid statics. But if it is subjected to acceleration, special treatment is required. Consider the case of a liquid mass in an open tank moving horizontally with a linear acceleration a_x, as in Fig. 3.24*a*. A free-body diagram (Fig. 3.24*b*) of a small particle (mass m) of liquid on the surface indicates that the forces exerted by the surrounding liquid on the particle are such that $F_z = F_B = -W$ and $F_x = ma_x$. F_z counterbalances W, so there is no acceleration in the z direction. F_x is the force required to produce acceleration a_x of the particle. Equal and opposite to these forces are F_x' and F_z' of Fig. 3.24*a*, the forces exerted by the particle on the surrounding fluid. The resultant of these forces is F'. The liquid surface must be at right angles to F', for if it were not, the particle would not maintain its fixed relative position in the liquid. Hence (Fig. 3.24*a*) $\tan\theta = -a_x/g$. The liquid surface and all other planes of equal hydrostatic pressure must be inclined at angle θ with the horizontal as in Fig. 3.24*a*.

Next let us consider the more general case where a fluid mass is accelerating in both the x and z directions. Figure 3.25 is a free-body diagram of an elemental cube of fluid, volume $\delta x\,\delta y\,\delta z$, with pressure p at its center. Applying the

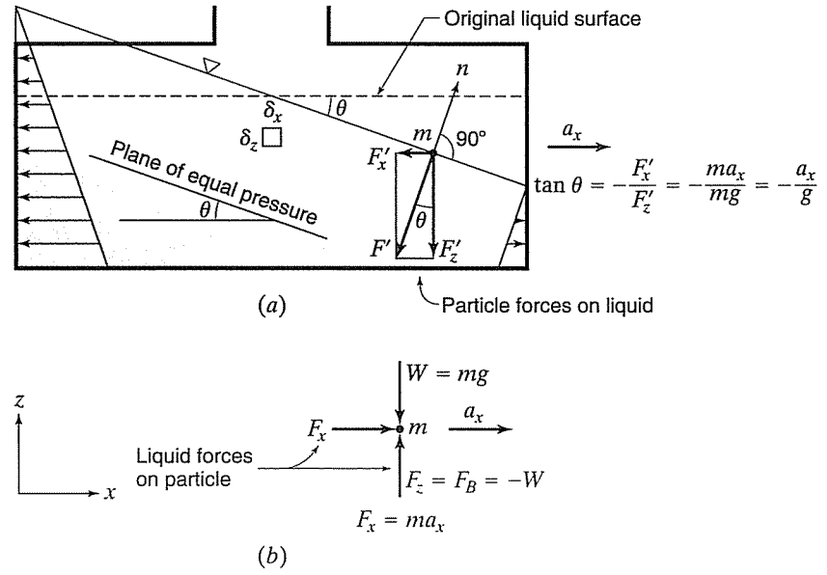


Figure 3.24
Liquid mass subjected to horizontal acceleration.

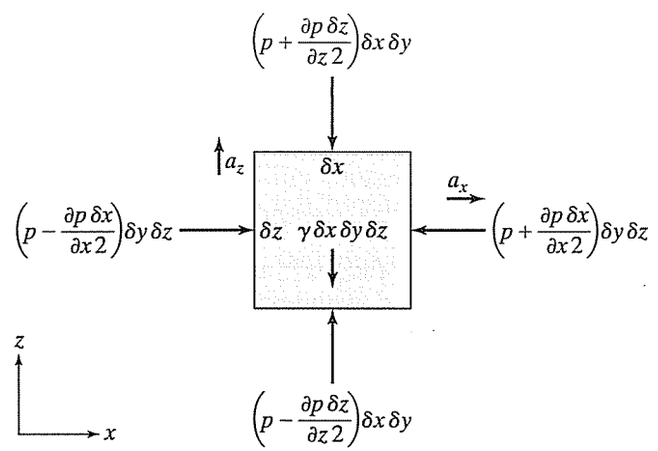


Figure 3.25
Elemental cube of fluid, thickness δy.

equation of motion in the x direction,

$$\sum F_x = ma_x$$

$$\left(p - \frac{\partial p}{\partial x}\frac{\delta x}{2}\right)\delta y\,\delta z - \left(p + \frac{\partial p}{\partial x}\frac{\delta x}{2}\right)\delta y\,\delta z = \rho\,\delta x\,\delta y\,\delta z\,a_x$$

which reduces to

$$\frac{\partial p}{\partial x} = -\rho a_x \tag{3.23}$$

Thus, as seems intuitive, fluid pressure reduces in the direction of acceleration. In the vertical direction

$$\sum F_z = ma_z$$

or $$\left(p - \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x\,\delta y - \left(p + \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x\,\delta y - \gamma\,\delta x\,\delta y\,\delta z = \rho\,\delta x\,\delta y\,\delta z\,a_z$$

where $\gamma = \rho g$ (Eq. 2.1). This yields

$$\frac{\partial p}{\partial z} = -\rho(a_z + g) \tag{3.24}$$

Therefore the pressure decreases with elevation z in a static fluid (Sec. 3.3), and it does so more rapidly if the fluid is being accelerated upward.

We can use Eqs. (3.23) and (3.24) to obtain a general result for a liquid mass that is accelerating in both the x and z directions. The chain rule for the total differential of dp in terms of its partial derivatives is

$$dp = \frac{\partial p}{\partial x}dx + \frac{\partial p}{\partial z}dz$$

So substituting the expressions for $\partial p/\partial x$ and $\partial p/\partial z$ from Eqs. (3.23) and (3.24), we get

$$dp = -\rho(a_x)dx - \rho(a_z + g)dz \tag{3.25}$$

Along a line of constant pressure, $dp = 0$. From Eq. (3.25), if $dp = 0$,

For p = constant: $$\frac{dz}{dx} = -\frac{a_x}{a_z + g} \tag{3.26}$$

This defines the slope $dz/dx = \tan\theta$ of a line of constant pressure within the accelerated liquid mass; the liquid surface is one such line.

In obtaining Eq. (3.26) from (3.25) we divided out the mass, represented by ρ. If we consider a liquid particle of mass m within the liquid, then using Newton's second law ($F = ma$):

$$\tan\theta = \frac{dz}{dx} = -\frac{ma_x}{ma_z + mg} = -\frac{F_x}{F_z + F_B}$$

So we see that the three accelerations in the right side of Eq. (3.26) represent the three forces exerted by the liquid on the liquid particle, as depicted in Fig. 3.26. These forces together produce the net force F, which is normal to lines of constant pressure. Note that g is upward-acting, in the positive z direction, because here it represents buoyancy F_B ($= -W$).

From Eqs. (3.23) and (3.24), we may obtain the resultant of $\partial p/\partial x$ and $\partial p/\partial z$, namely;

$$\frac{\partial p}{\partial n} = -\rho\sqrt{a_x^2 + (a_z + g)^2} \tag{3.27}$$

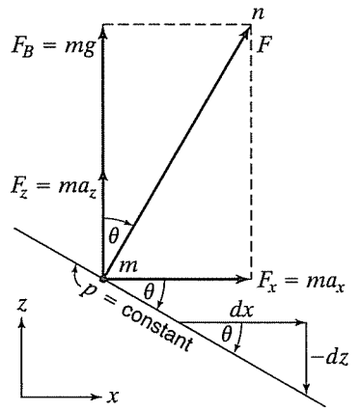
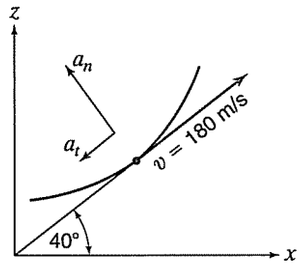


Figure 3.26
Liquid forces on an accelerating particle.

where n is at right angles to the lines of equal pressure and in the direction of the most rapidly decreasing pressure (Fig. 3.26). When $a_x = a_z = 0$, this equation reduces to $\partial p/\partial n = -\rho g = -\gamma$, which is essentially the same as the basic hydrostatic equation (3.2). Equation (3.27) indicates that, if liquid in a container experiences an upward acceleration, this increases pressures within the liquid; downward acceleration decreases them.

SAMPLE PROBLEM 3.10 At a particular instant an airplane is traveling upward at a velocity of 180 m/s in a direction that makes an angle of 40° with the horizontal. At this instant the airplane is losing speed at the rate of 4 m/s^2. Also it is moving on a concave-upward circular path having a radius of 2600 m. Determine for the given conditions the slope of the free liquid surface in the fuel tank of this vehicle.

Solution



$a_n = \frac{v^2}{r} = \frac{180^2}{2600} = 12.5$ m/s^2 toward the center of curvature of the path

$a_t = 4$ m/s^2 downward to the left (given)

$a_x = -4\cos 40° - 12.5\sin 40° = -11.10$ m/s^2

$a_z = -4\sin 40° + 12.5\cos 40° = 7.00$ m/s^2

Eq. (3.26): Slope of the free surface $= \frac{dz}{dx} = -\left(\frac{-11.10}{7.00 + 9.81}\right) = +0.660$ ***ANS***

$\theta = \tan^{-1}(0.660) = 33.4°$ ***ANS***

EXERCISES

3.10.1 What must be the hydrostatic gage pressure at a depth of 8 inches in a bucket of oil ($s = 0.86$) that is in an elevator being accelerated upward at 15 ft/sec^2?

3.10.2 What must be the hydrostatic gage pressure at a depth of 250 mm in a bucket of oil ($s = 0.88$) that is in an elevator being accelerated upward at 4 m/s^2?

3.10.3 A tank containing water to a depth of 5 ft is accelerated upward at 8 ft/sec^2. Calculate the pressure on the bottom of the tank.

3.10.4 A tank containing water to a depth of 2.5 m is accelerated upward at 3.6 m/s^2. Calculate the pressure on the bottom of the tank.

3.10.5 Suppose the tank shown in Fig. 3.24 is rectangular and completely open at the top. It is 15 ft long, 6 ft wide, and 4 ft deep. If it is initially filled to the top, how much liquid will be spilled if it is given a horizontal acceleration $a_x = 0.2g$ in the direction of its length?

3.10.6 Suppose the tank of Fig. 3.24 is rectangular and completely open at the top. It is 15 m long, 5 m wide, and 4 m deep. If it is initially filled to the top, how much liquid will be spilled if it is given a horizontal acceleration $a_x = 0.5g$ in the direction of its length?

3.10.7 If the tank of Exer. 3.10.5 is closed at the top and is completely filled, what must be the pressure difference between the left-hand end at the top and the right-hand end at the top if the liquid has a specific weight of 50 lb/ft^3 and the horizontal acceleration is $a_x = 0.3g$? Sketch planes of equal pressure, indicating their magnitude; assume zero pressure in the upper right-hand corner.

3.10.8 If the tank of Exer. 3.10.6 is closed at the top and is completely filled, what must be the pressure difference between the left-hand end at the top and the right-hand end at the top if the liquid has a specific weight of 8.0 kN/m^3 and the horizontal acceleration is $a_x = 0.3g$? Sketch planes of equal pressure, indicating their magnitude; assume zero pressure in the upper right-hand corner.

PROBLEMS

3.1 A pressure gage at elevation 4.8 m on the side of a storage tank containing oil reads 34.7 kPa. Another gage at elevation 2.2 m reads 57.5 kPa. Compute the specific weight, density, and specific gravity of the oil.

3.2 On a certain day the barometric pressure at sea level is 30.0 inHg and the temperature is 60°F. The pressure gage on an airplane flying overhead indicates that the atmospheric pressure at that point is 9.7 psia and that the air temperature is 42°F. Calculate as accurately as you can the height of the airplane above sea level. Assume a linear decrease of temperature with elevation.

3.3 Repeat Exer. 3.2.1, but consider the effects of compressibility ($E_v = 330{,}000$ psi). Neglect changes in density caused by temperature variations. (*Hint:* As a starting point, express Eq. (2.3) in terms of γ and integrate to determine γ as a function of z.)

3.4 If the specific weight of a sludge can be expressed as $\gamma = 64.0 + 0.22h$, determine the pressure in psi at a depth of 14 ft below the surface. γ is in lb/ft^3, and h is in ft below the surface.

3.5 A bubble 4 in below the water surface contains 2×10^{-7} lb of air. If the temperature is 60°F and the barometric pressure is 14.7 psia, calculate the diameter of the bubble. Refer to Secs. 2.7 and 2.12, and ignore the partial pressure of water vapor inside the bubble.

3.6 The absolute pressure on a gas is 41 psia and the atmospheric pressure is 965 mb abs. Find the gage pressure in psi, kPa, and mb.

3.7 The tire of an airplane is inflated at sea level to 60 psi. Assuming the tire does not expand, what is the pressure within the tire at elevation 40,000 ft? Assume standard atmosphere. Express the answer in psig and psia.

3.8 The tire of an airplane is inflated at sea level to 350 kPa. Assuming the tire does not expand, what is the pressure within the tire at elevation (*a*) 10000 m; (*b*) 20000 m? Assume standard atmosphere. Express answers in both kPa gage and kPa abs.

3.9 In Fig. X3.5.8 assume the following: atmospheric pressure = 930 mb abs; vapor pressure of the alcohol = 110 mb abs; $x = 3.30$ m and $y = 1.60$ m. Compute the reading (*a*) on the pressure gage and (*b*) on the manometer.

3.10 The diameter of tube C in Fig. 3.11 is d_1, and that of tube B is d_2. Let z_0 be the elevation of the mercury above A when both mercury columns are at the same level. R is the distance the right-hand column of mercury rises above z_0 when the fluid in A is under pressure. Let γ' be the specific weight of the mercury (or any other measuring fluid), while γ is the specific weight of the fluid in A and the connecting tubing. Prove that

$$p_A = \gamma z_0 + \left[\gamma' + (\gamma' - \gamma)\left(\frac{d_1}{d_2}\right)^2\right]R$$

$$= M + NR$$

where M and N are constants. Note that this equation involves only one variable, which is the reading R on the scale for column C. It also shows the significance of having d_2 large compared with d_1.

3.11 What would be the manometer reading in Sample Prob. 3.4 if $p_B - p_A = 145$ kPa?

3.12 At a certain point the gage pressure in a pipeline containing gas ($\gamma = 0.05$ lb/ft^3) is 5.6 in of water. The gas is not flowing, and all temperatures are 60°F. What is the gage pressure in inches of water at another point in the line whose elevation is 650 ft greater than the first point? Make and state clearly any necessary assumptions.

3.13 A vertical semicircular area has its diameter in a liquid surface. Derive an expression for the depth to its center of pressure.

3.14 The Utah-shaped plate shown in Fig. P3.14 is submerged in oil ($s = 0.94$) and lies in a vertical plane. Find the magnitude and location of the hydrostatic force acting on one side of the plate.

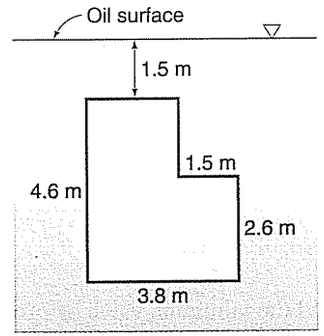


Figure P3.14

3.15 The common type of irrigation head gate shown in Fig. P3.15 is a plate that slides over the opening to a culvert. The coefficient of friction between the gate and its sliding ways is 0.6. Find the force required to slide open this 600-lb gate if it is set (*a*) vertically; (*b*) on a 2:1 slope ($n = 2$), as is common.

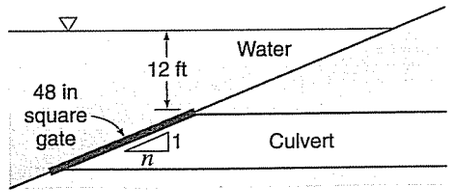


Figure P3.15

3.16 In the drainage of irrigated lands it is frequently desirable to install automatic flap gates to prevent a flood from backing up into the lateral drains from a river. Suppose a square flap gate, of side $b = 1.5$ m and weight 8 kN, is hinged 1 m above its center ($a = 0.25$ m), as shown in Fig. P3.16, and the face is sloped 4° from the vertical. To what depth will water rise behind the gate before it will open?

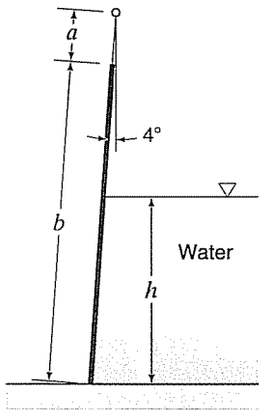


Figure P3.16

3.17 Find the minimum value of z for which the gate in Fig. P3.17 will rotate counterclockwise if the gate is (a) rectangular, 5 ft high by 4 ft wide; (b) triangular, 4 ft base as axis, height 5 ft. Neglect friction in bearings.

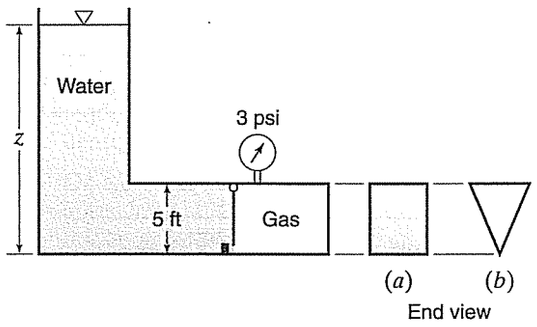


Figure P3.17

3.18 Repeat Exer. 3.7.12 for the case where a 2-m-thick layer of oil ($s = 0.8$) is resting on the water, and replace "water surface" by "oil surface."

3.19 Refer to Sample Prob. 3.7. If the oil depth were 2 ft rather than 1.5 ft, (a) compute the total force and (b) determine the depth of its center of pressure.

3.20 (a) Find the horizontal and vertical forces per foot of width acting on the Tainter gate in Fig. P3.20. (b) Locate the horizontal force and indicate the line of action of the vertical force without actually computing its location. (c) Locate the vertical force. (*Hint:* Consider the resultant.)

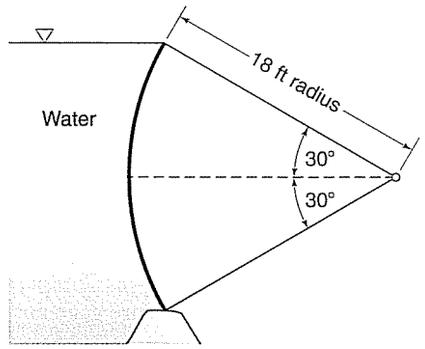


Figure P3.20

3.21 The cross section of a gate is shown in Fig. P3.21. Its dimension normal to the plane of the paper is 8 m, and its shape is such that $x = 0.2y^2$. The gate is pivoted about O. Develop analytic expressions in terms of the water depth y upstream of the gate for the following: (a) horizontal force; (b) vertical force; (c) clockwise moment acting on the gate. Compute (a), (b), and (c) for the case where the water depth is 2.5 m.

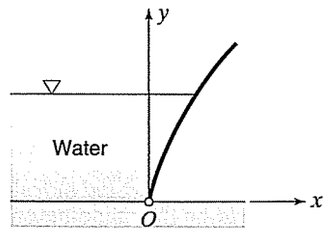


Figure P3.21

3.22 A tank has an irregular cross section as shown in Fig. P3.22. Determine as accurately as possible the magnitude and location of the horizontal- and vertical-force components on a 1-m length of the wall $ABCD$ when the tank contains water to a depth of 2 m. To determine areas, use a planimeter or count squares (0.25 m grid); make a cardboard cutout, or take approximate moments of the squares, to locate the centroid.

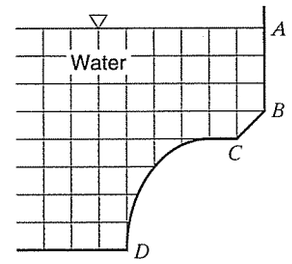


Figure P3.22

3.23 Repeat Exer. 3.8.2 where the tank contains 4 ft of water overlain with a gas that is under a pressure of 0.8 psi.

3.24 Find the approximate value of the maximum specific gravity of liquid for which the device of Exer. 3.9.8 will be stable.

3.25 A 2.0-ft^3 object weighing 650 lb is attached to a balloon of negligible weight and released in the ocean ($\gamma = 64$ lb/ft^3). The balloon was originally inflated with 5.0 lb of air to a pressure of 20 psi. To what depth will the balloon sink? Assume that air temperature within the balloon stays constant at 50°F.

3.26 Work Prob. 3.25 with all data the same except assume the balloon was originally inflated with 5.0 lb of air to a pressure of 10 psi. In this latter case the balloon is more elastic because a lower pressure is obtained with the same amount of air.

3.27 A wooden pole weighing 2 lb/ft has a cross-sectional area of 6.7 in^2 and is supported as shown in Fig. P3.27. The hinge is frictionless. Find θ.

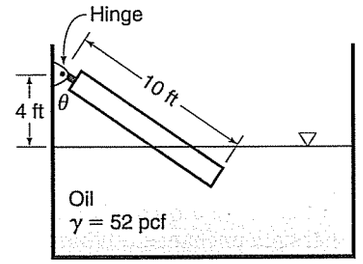


Figure P3.27

3.28 A rectangular block of uniform material and length $L = 3$ ft, width $b = 1.25$ ft, and depth $d = 0.20$ ft, is floating in a liquid. It assumes the position shown in Fig. P3.28 when a uniform vertical load of 1.30 lb/ft is applied at P. (a) Find the weight of the block. (b) If the load is suddenly removed, what is the righting moment before the block starts to move? (*Hint:* Refer also to Fig. 3.19.)

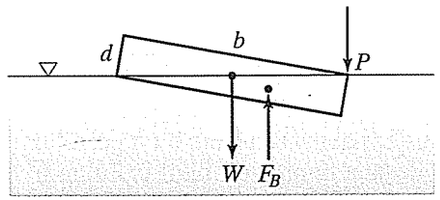


Figure P3.28

3.29 A rectangular block of uniform material and length $L = 800$ mm, width $b = 300$ mm, and depth $d = 50$ mm, is floating in a liquid. It assumes the position shown in Fig. P3.28 when a uniform vertical load of 20 N/m is applied at P. (a) Find the weight of the block. (b) If the load is suddenly removed, what is the righting moment before the block starts to move? (*Hint:* Refer also to Fig. 3.19.)

3.30 A solid block, 4 in wide by 4 in deep and 3 in high weighs 0.90 lb. It floats in liquid ($\gamma = 55$ lb/ft^3) inside a cubic container of side 5 in. Before immersion the liquid was 2 in deep. (*a*) At what level will the block float? Find the distance z from the bottom of the block to the bottom of the container. (*b*) If the block is tilted by a couple (no net force) to an angle of 15° so that two sides remain vertical, what will be the righting moment in lb·in?

3.31 Refer to Sample Prob. 3.10. Suppose the velocity of the airplane is 220 m/s, with all other data unchanged. What then would be the slope of the liquid surface in the tank?

3.32 At a particular instant an airplane is traveling upward at a velocity of 180 mph in a direction that makes an angle of 30° with the horizontal. At this instant the airplane is losing speed at the rate of 3.6 mph/sec. Also, it is moving on a concave-upward circular path of radius 5000 ft. Determine the slope of the free liquid surface in the airplane's fuel tank.

CHAPTER 4
Basics of Fluid Flow

In this chapter we shall deal with fluid velocities and accelerations and their variations in space without considering any forces involved. As we mentioned in Sec. 1.1, this subject, that deals with velocities and flow paths without considering forces or energy, is known as ***kinematics.***

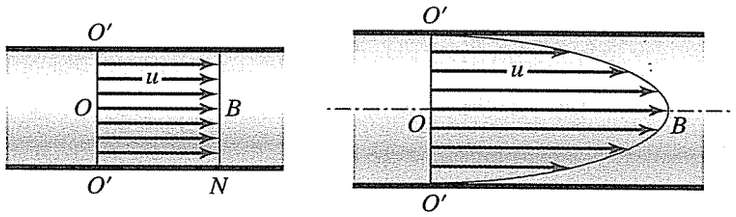
Because only certain types of flow can be treated by the methods of kinematics, and because there are many different types of flow, we summarize these first to provide perspective. We shall also introduce some related concepts, most notably the control volume and the flow net.

4.1 TYPES OF FLOW

When speaking of fluid flow, we often refer to the flow of an ***ideal fluid*** (Sec. 2.10). We presume that such a fluid has no viscosity. This is an idealized situation that does not exist; however, there are instances in engineering problems where the assumption of an ideal fluid is helpful. When we refer to the flow of a ***real fluid,*** the effects of viscosity are introduced into the problem. This results in the development of shear stresses between neighboring fluid particles when they are moving at different velocities. In the case of an ideal fluid flowing in a straight conduit, all particles move in parallel lines with equal velocity (Fig. 4.1*a*). In the flow of a real fluid the velocity adjacent to the wall will be zero; it will increase rapidly within a short distance from the wall and produce a velocity profile such as shown in Fig. 4.1*b*.

Flow can also be classified as that of an ***incompressible*** or ***compressible*** fluid. Since liquids are relatively incompressible, we generally treat them as wholly incompressible fluids. Under particular conditions where there is little pressure variation, we may also consider the flow of gases to be incompressible, though generally we should consider the effects of the compressibility of the gas. Basic concepts governing the flow of compressible fluids are discussed in Chap. 13.

In addition to the flow of different types of fluids, i.e., real, ideal, incompressible, and compressible, there are various classifications of flow. Flow may be ***steady*** or ***unsteady*** with respect to time (see Sec. 4.3). It may be ***laminar*** or



(*a*) Ideal fluid (*b*) Real fluid

Figure 4.1
Typical velocity profiles.

turbulent, as discussed in the following section. Other classifications of flow include ***rotational*** or ***irrotational*** (Chap. 14), ***supercritical*** or ***subcritical*** (Chap. 10), etc. These and other common ways in which we can classify flow are listed in Table 4.1, in many cases with definitions.

4.2 LAMINAR AND TURBULENT FLOW

Whether laminar or turbulent flow occurs in a given situation, or how much of each occurs, is very important because of the strongly different effects these two different types of flow have on a variety of flow features, including on energy losses, velocity profiles, and mixing of transported materials.

Osborne Reynolds[1] demonstrated in 1883 that there are two distinctly different types of fluid flow. He injected a fine, threadlike stream of colored liquid having the same density as water at the entrance to a large glass tube through which water was flowing from a tank. A valve at the discharge end permitted him to vary the flow. When the velocity in the tube was small, he saw this colored liquid as a straight line throughout the length of the tube, showing that the particles of water moved in parallel straight lines. As he gradually increased the velocity of the water by opening the valve further, at a certain velocity the flow changed. The line first became wavy, and then at a short distance from the entrance it broke into numerous vortices beyond which the color became uniformly diffused so that no streamlines could be distinguished. Later observations have shown that in this latter type of flow the velocities are continuously subject to irregular fluctuations.

The first type is known as ***laminar, streamline,*** or ***viscous*** flow. The significance of these terms is that the fluid appears to move by the sliding of laminations of infinitesimal thickness over adjacent layers, with relative motion of fluid particles occurring at a molecular scale; that the particles move in definite and observable paths or streamlines, as in Fig. 4.2; and also that the flow is characteristic of a viscous fluid or is one in which viscosity plays a significant part (Fig. 2.4 and Sec. 2.11).

[1] Other famous contributions by Reynolds are discussed in Sec. 7.4.

UNIVERSIDAD POLITECNICA SALESIANA
BIBLIOTECA

TABLE 4.1 Classification of types of flow[a]

One-dimensional, two-dimensional or ***three-dimensional flow***
See Sec. 4.8 for discussion.

Real fluid flow or ***ideal fluid flow*** (also referred to as ***viscid*** and ***inviscid flow***)
Real fluid flow implies frictional (viscous) effects. Ideal fluid flow is hypothetical; it assumes no friction (i.e., viscosity of fluid = 0).

Incompressible fluid flow or ***compressible fluid flow***
Incompressible fluid flow assumes the fluid has constant density (ρ = constant). Though liquids are slightly compressible we usually assume them to be incompressible. Gases are compressible; their density is a function of absolute pressure and absolute temperature [$\rho = f(p, T)$].

Steady or ***unsteady flow***
Steady flow means steady with respect to time. Thus all properties of the flow at every point remain constant with respect to time. In unsteady flow, the flow properties at a point change with time.

Pressure flow or ***gravity flow***
Pressure flow implies that flow occurs under pressure. Gases always flow in this manner. When a liquid flows with a free surface (for example, a partly full pipe), we refer to the flow as gravity flow, because gravity is the primary moving force. Liquids also flow under pressure (for example, a pipe flowing full).

Spatially constant or ***spatially variable flow***
Spatially constant flow occurs when the fluid density and the local average flow velocity are identical at all points in a flow field. If these quantities change along or across the flow lines, the flow is spatially variable. Examples of different types of spatially varied flow include the local flow field around an object, flow through a gradual contraction in a pipeline, and the flow of water in a uniform gutter of constant slope receiving inflow over the length of the gutter.

Laminar or ***turbulent flow***
See Sec. 4.2 for a discussion of the difference between these two types of flow.

Established or ***unestablished flow***
We discuss these in Sec. 8.8.

Uniform or ***varied flow***
We ordinarily use these classifications when dealing with open-channel (gravity) flow (Chap. 10). In uniform flow the cross section (shape and area) through which the flow occurs remains constant.

Subcritical or ***supercritical flow***
We use these classifications with open-channel flow (Chap. 10).

Subsonic or ***supersonic flow***
We use these classifications with compressible flow (Chap. 13).

Rotational or ***irrotational flow***
We use these in mathematical hydrodynamics (Chap. 14).

Other classifications of flow include ***converging*** or ***diverging, disturbed, isothermal*** (constant temperature), ***adiabatic*** (no heat transfer), and ***isentropic*** (frictionless adiabatic).

[a] Note that in a given situation these different types of flow may occur in combination. For example, we usually consider flow of a liquid in a pipe to be one-dimensional, incompressible, real fluid flow that may be steady or unsteady, and laminar or turbulent. Such flow is commonly spatially constant and established.

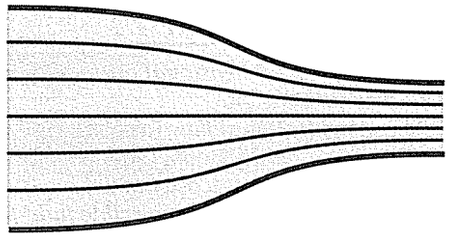
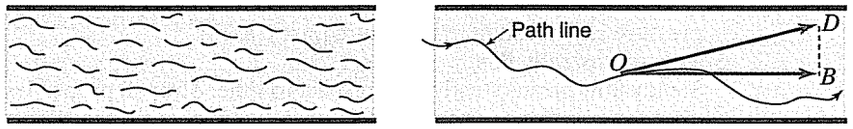


Figure 4.2



(*a*) (*b*)

Figure 4.3
Turbulent flow.

The second type is known as ***turbulent*** flow, and is illustrated in Fig. 4.3, where (*a*) represents the irregular motion of a large number of particles during a very brief time interval, while (*b*) shows the erratic path followed by a single particle during a longer time interval. A distinguishing characteristic of turbulence is its irregularity, there being no definite frequency as in wave action, and no observable pattern as in the case of large swirls.

Large swirls and irregular movements of large bodies of fluid, which can be traced to obvious sources of disturbances, do not constitute turbulence, but may be described as ***disturbed flow.*** By contrast, the far more common phenomenon of turbulence may often be found in what appears to be a very smoothly flowing stream and one in which there is no apparent source of disturbance. Turbulent flow is characterized by fluctuations in velocity at all points of the flow field (Figs. 4.6 and 8.6*b*). These fluctuations arise because the fluid moves as many small, discrete particles or "packets" called ***eddies,*** jostling each other around in a random manner. Although small, the smallest eddies are macroscopic in size, very much larger than the molecular sizes of the particles in laminar flow. The eddies interact with one another and with the general flow. They are the cause of the effective mixing action experienced with turbulent flow. They are often caused by rotation, particularly near boundaries, and so the eddies themselves often rotate. They change shape and size with time as they move along with the flow. Each eddy dissipates its energy through viscous shear with its surroundings and eventually disappears. New eddies are continuously forming. Large eddies (large-scale turbulence) have smaller eddies within them giving rise to small-scale turbulence. The resulting fluctuations in velocity are rapid and irregular, and often we can only detect them by a fast-acting probe such as a hot-wire or hot-film anemometer (Sec. 11.4).

At a certain instant the flow passing point *O* in Fig. 4.3*b* may be moving with the velocity *OD*. In turbulent flow *OD* will vary continuously both in

direction and in magnitude. Fluctuations of velocity are accompanied by fluctuations in pressure, which is the reason why manometers or pressure gages attached to a pipe containing flowing fluid usually show pulsations. In this type of flow an individual particle will follow a very irregular and erratic path, and no two particles may have identical or even similar motions. Thus a rigid mathematical treatment of turbulent flow is impossible, and instead we must use statistical methods of evaluation.

Criteria governing the conditions under which the flow will be laminar and those under which it will be turbulent are discussed in Sec. 8.2.

4.3 STEADY FLOW AND UNIFORM FLOW

A ***steady flow*** is one in which all conditions at any point in a stream remain constant with respect to *time,* but the conditions may be different at different points. A truly ***uniform flow*** is one in which the velocity is the same in both magnitude and direction at a given instant at every point in the fluid. Both of these definitions must be modified somewhat, since true steady flow is found only in laminar flow. In turbulent flow there are continual fluctuations in velocity and pressure at every point, as was just explained. But if the values fluctuate equally on both sides of a constant average value, we call the flow steady flow. However, a more precise term for this condition would be ***mean steady flow.***

Likewise, this strict definition of uniform flow can have little meaning for the flow of a real fluid where the velocity varies across a section, as in Fig. 4.1*b*. But when the size and shape of cross section are constant along the length of channel under consideration, we say the flow is ***uniform.***

Steady (or unsteady) and uniform (or nonuniform) flow can exist independently of each other, so that any of four combinations is possible. Thus the flow of liquid at a constant rate in a long straight pipe of constant diameter is ***steady uniform*** flow, the flow of liquid at a constant rate through a conical pipe is ***steady nonuniform*** flow, while at a changing rate of flow these cases become ***unsteady uniform*** and ***unsteady nonuniform flow,*** respectively.

Unsteady flow is a transient phenomenon, which may in time become either steady flow or zero flow. An example is given in Fig. 4.4, where *a* denotes the surface of a stream that has just been admitted to the bed of a canal by the sudden opening of a gate. After a time the water surface will be at *b,* later at *c,* and finally it reaches equilibrium at *d*. The unsteady flow has then become mean

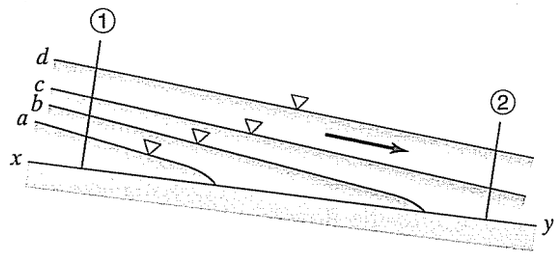


Figure 4.4
Unsteady flow in a canal.

steady flow. Another example of transient phenomenon is when a valve is closed at the discharge end of a pipeline (Sec. 12.6), thus causing the velocity in the pipe to decrease to zero. In the meantime, there will be fluctuations in both velocity and pressure within the pipe.

Unsteady flow may also include periodic motion such as that of waves on beaches, tidal motion in estuaries, and other oscillations. The difference between such cases and that of mean steady flow is that the deviations from the mean are very much greater and the time scale is also much longer.

EXERCISE

4.3.1 Classify the following cases of flow as to whether they are steady or unsteady, uniform or nonuniform: (*a*) water flowing from a tilted pail; (*b*) flow from a rotating lawn sprinkler; (*c*) flow through the hose leading to the sprinkler; (*d*) a natural stream during dry-weather flow; (*e*) a natural stream during flood; (*f*) flow in a city water-distribution main through a straight section of constant diameter with no side connections. (*Note:* There is room for legitimate argument in some of the above cases, which should stimulate independent thought.)

4.4 PATH LINES, STREAMLINES, AND STREAK LINES

A ***path line*** (Fig. 4.3*b*) is the trace made by a *single* particle over a *period* of time. If a camera were to take a time exposure of a flow in which a fluid particle was colored so it would register on the negative, the picture would show the course followed by the particle. This would be its path line. The path line shows the direction of the velocity of the particle at successive instants of time.

Streamlines show the mean direction of a *number* of particles at the *same* instant of time. If a camera were to take a very short time exposure of a flow in which there were a large number of particles, each particle would trace a short path, which would indicate its velocity during that brief interval. A series of curves drawn tangent to the means of the velocity vectors are streamlines. It follows that there can be no net velocity normal to a streamline.

Path lines and streamlines are identical in the steady flow of a fluid in which there are no fluctuating velocity components, in other words, for truly steady flow. Such flow may be either that of an ideal frictionless fluid or that of one so viscous and moving so slowly that no eddies are formed. This latter is the ***laminar*** type of flow, wherein the layers of fluid slide smoothly, one upon another. In turbulent flow, however, path lines and streamlines are not coincident, the path lines being very irregular while the streamlines are everywhere tangent to the local mean temporal velocity. The lines in Fig. 4.2 represent both path lines and streamlines if the flow is laminar; they represent only streamlines if the flow is turbulent.

In experimental fluid mechanics, a dye or other tracer is frequently injected into the flow to trace the motion of the fluid particles. If the flow is laminar, a ribbon of color results. This is called a ***streak line,*** or ***filament line.***

It is an instantaneous picture of the positions of all particles in the flow that have passed through a given point (namely, the point of injection). When using fluid-tracer techniques it is important to choose a tracer with physical characteristics (especially density) the same as those of the fluid being observed. Thus the smoke rising from an incense stick, while giving the appearance of a streak line, does not properly represent the movement of the ambient air in the room because it is less dense (warmer) than the air and therefore rises more rapidly.

4.5 FLOW RATE AND MEAN VELOCITY

We call the quantity of fluid flowing per unit time across any section the ***flow rate.*** We may express it (1) in terms of ***volume flow rate (discharge)*** using BG units such as cubic feet per second (cfs), gallons per minute (gpm), million gallons per day (mgd), or (2) in terms of ***mass flow rate*** (slugs per second), or (3) in terms of ***weight flow rate*** (pounds per second). In SI units, cubic meters per second (m^3/s), kilograms per second (kg/s), and kilonewtons per second (kN/s) are fairly standard for expressing volume, mass, and weight flow rates, respectively. When dealing with incompressible fluids, we commonly use volume flow rate, whereas weight flow rate or mass flow rate is more convenient with compressible fluids.

Figure 4.5 depicts a streamline in steady flow lying in the xz plane. Element of area dA lies in the yz plane. The mean velocity at point P is u. The volume flow rate passing through the element of area dA is

$$dQ = \mathbf{u}\cdot d\mathbf{A} = (u\cos\theta)\,dA = u(\cos\theta\, dA) = u\,dA' \qquad (4.1)$$

where dA' is the projection of dA on the plane normal to the direction of u. This indicates that the *volume flow rate is equal to the magnitude of the mean velocity multiplied by the flow area at right angles to the direction of the mean velocity.* We can compute the mass flow rate and the weight flow rate by multiplying the volume flow rate by the density and specific weight of the fluid respectively.

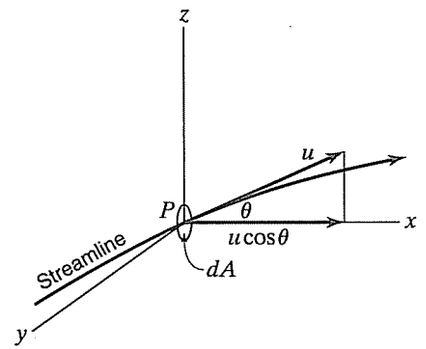


Figure 4.5

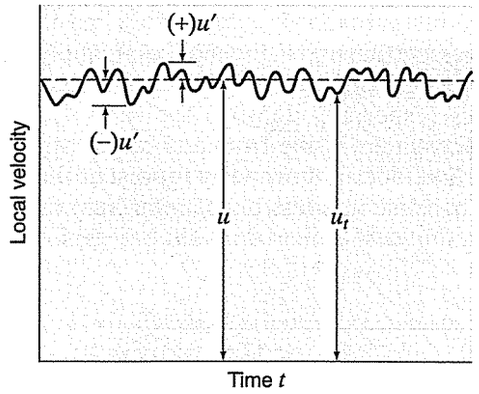


Figure 4.6
Fluctuating velocity at a point due to turbulence.

If the flow is turbulent, the ***instantaneous velocity component*** u_t along the streamline will fluctuate with time, even though the flow is nominally steady. A plot of u_t as a function of time is shown in Fig. 4.6. The average value of u_t over a period of time determines the time (temporal) mean value of velocity u at point P.

The difference between u_t and u, which we shall denote by u', is called the ***turbulent fluctuation*** of this component; it may be either positive or negative, but the time mean value of u' must be zero. Likewise, the time means of all velocity components perpendicular to the streamline must also be zero. At any instant, then,

$$u_t = u + u' \tag{4.2}$$

and we can evaluate u for any finite time t from $u = (1/t)\int_0^t u_t\,dt$.

In a real fluid the local time mean velocity u will vary across the section in some manner, such as that shown in Fig. 4.1*b*, and so we can express the flow rate as

$$Q = \int_A u\,dA = AV \tag{4.3}$$

or, for constant-density flow,

$$\dot{m} = \rho\int_A u\,dA = \rho AV = \rho Q \tag{4.4}$$

or

$$G = g\dot{m} = \gamma\int_A u\,dA = \gamma AV = \gamma Q \tag{4.5}$$

where u is the time mean velocity through an infinitesimal area dA, while V is the ***mean, or average, velocity*** over the entire sectional area A;[2] Q is the volume flow rate (cfs or m^3/s), $\dot{m}$ is the mass flow rate (slugs/sec or kg/s),[3] and G is the weight flow rate (lb/sec or kN/s).[4] If u is known as a function of A, we can integrate Eqs. (4.3), (4.4), and (4.5). If only an average value of V is known for each finite subarea of the total sectional area, then

$$Q = A_aV_a + A_bV_b + \cdots + A_nV_n = AV$$

We can write similar expressions for $\dot{m}$ and G. If we have determined the flow rate directly by some method, then from Eqs. (4.3)–(4.5) we can find the mean velocity,

$$V = \frac{Q}{A} = \frac{\dot{m}}{\rho A} = \frac{G}{\gamma A} \tag{4.6}$$

For the very common occurrence of flow through a circular pipe, we can substitute $A = \pi D^2/4$ into Eq. (4.6), to yield the mean velocity as

Circular pipe:
$$V = \frac{4Q}{\pi D^2} = \frac{4\dot{m}}{\pi D^2\rho} = \frac{4G}{\pi D^2\gamma} \tag{4.7}$$

SAMPLE PROBLEM 4.1 Air at 100°F and under a pressure of 40 psia flows in a 10-in-diameter ventilation duct at a mean velocity of 30 fps. Find the mass flow rate.

Solution

Table A.5 for air: $R = 1715$ ft·lb/(slug·°R)

From Eq. (2.4):
$$\rho = \frac{p}{RT} = \frac{40(144)}{1715(460 + 100)} = 0.00600 \text{ slug/ft}^3$$

Eq. (4.4):
$$\dot{m} = \rho AV = (0.00600)\frac{\pi}{4}\left(\frac{10}{12}\right)^2(30) = 0.0981 \text{ slug/sec} \quad \textit{ANS}$$

[2] Note that we define area A by the surface at right-angles to the velocity vectors.
[3] Here, as used on m, and subsequently, the overdot represents the time derivative, as is standard practice.
[4] In Eqs. (4.4) and (4.5) the ρ and γ should be to the right of the integral sign if the density of the fluid varies across the flow.

EXERCISES

4.5.1 In the laminar flow of a fluid through a pipe of circular cross section the velocity profile is exactly a true parabola. The volume of the paraboloid represents the rate of discharge. Prove that for this case the ratio of the mean velocity to the maximum velocity is 0.5.

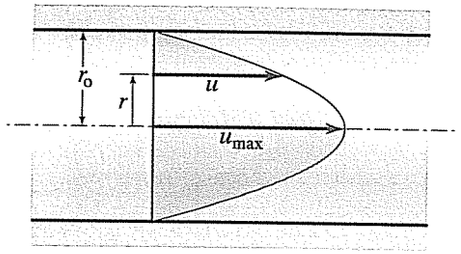


Figure X4.5.1

4.5.2 A gas ($\gamma = 0.05$ lb/ft^3) flows at the rate of 0.8 lb/sec past section A through a long rectangular duct of uniform cross section 1.2 ft by 1.8 ft. At section B some distance along the duct the gas weighs 0.08 lb/ft^3. What is the average velocity of flow at sections A and B?

4.5.3 The velocity of a liquid ($s = 1.4$) in a 150-mm pipeline is 0.8 m/s. Calculate the rate of flow in L/s, m^3/s, kg/s, and kN/s.

4.5.4 Water flows at 4 gal/min through a small circular hole in the bottom of a large tank. Assuming the water in the tank approaches the hole radially, what is the velocity in the tank at 2, 4, and 8 in from the hole?

4.5.5 Water flows at 0.25 L/s through a small circular hole in the bottom of a large tank. Assuming the water in the tank approaches the hole radially, what is the velocity in the tank at 50, 100, and 200 mm from the hole?

4.6 FLUID SYSTEM AND CONTROL VOLUME

The concept of a free body diagram, as used in the statics of rigid bodies and in fluid statics (e.g., Fig. 3.1), is usually inadequate for the analysis of moving fluids. Instead, we frequently find the concepts of a *fluid system* and a *control volume* to be useful in the analysis of fluid mechanics.

A ***fluid system*** refers to a *specific mass of fluid* within the boundaries defined by a closed surface. The shape of the system, and so the boundaries, may change with time, as when liquid flows through a constriction or when gas is compressed; as a fluid moves and deforms, so the system containing it moves and deforms. The size and shape of a system is entirely optional.

In contrast, a ***control volume*** (CV) refers to a *fixed region in space, which does not move or change shape* (Fig. 4.7). We usually choose it as a region that fluid flows into and out of. We call its closed boundaries the ***control surface.*** Again, the size and shape of a control volume is entirely optional, although we often choose the boundaries to coincide with some solid or other natural flow boundaries. Actually, the control surface may be in motion through space relative to an absolute frame of reference; this is acceptable provided the motion is limited to constant-velocity translation.

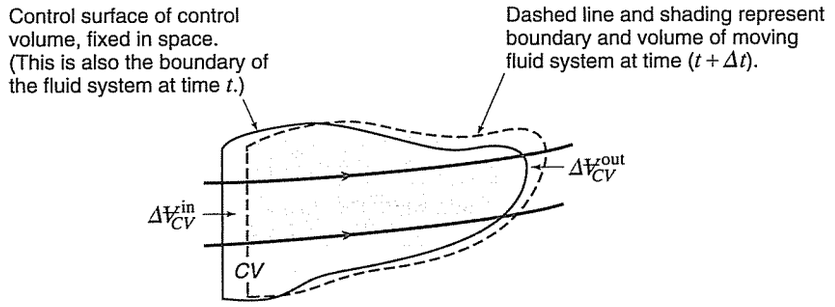


Figure 4.7
Fluid system, control volume, and differences.

We shall now derive a general relationship between a system and a control volume that provides an important basis for the equations of continuity, energy, and momentum for moving fluids. This relationship is derived from what we commonly call the ***control volume approach,*** more formally known as the ***Reynolds transport theorem.*** Addressing the motion of fluid as it moves through a given region, the control volume approach is also called the ***Eulerian approach,*** in contrast to the ***Lagrangian approach*** in which we describe the motion of each particle by its position as a function of time.

Let X represent the total amount of some fluid property (scalar or vector), such as mass, energy, or momentum, contained within specified boundaries at a specified time. It will probably help to think of X as mass for most of this section. The specified boundaries will be either those of a system, indicated by a subscript S, or those of a control volume, indicated by a subscript CV. Consider the general flow situation of Fig. 4.7. At time t, the boundaries of the system and the control volume were chosen to coincide, so $(X_S)_t = (X_{CV})_t$. At instant Δt later, the system has moved a little through the control volume and possibly slightly changed its shape; a small amount of new fluid $\Delta \mathcal{V}_{CV}^{in}$ has entered the control volume, and another small amount of system fluid $\Delta \mathcal{V}_{CV}^{out}$ has left the control volume, where $\mathcal{V}$ represents volume. These small volumes carry small amounts of property X (mass, etc.) with them, so that ΔX_{CV}^{in} enters and ΔX_{CV}^{out} leaves the control volume. Comparing X in the various volumes, we see that

$$(X_S)_{t+\Delta t} = (X_{CV})_{t+\Delta t} + \Delta X_{CV}^{out} - \Delta X_{CV}^{in}$$

Subtracting the equation for t from that for $t + \Delta t$, we obtain

$$(X_S)_{t+\Delta t} - (X_S)_t = (X_{CV})_{t+\Delta t} - (X_{CV})_t + \Delta X_{CV}^{out} - \Delta X_{CV}^{in}$$

or

$$\Delta X_S = \Delta X_{CV} + \Delta X_{CV}^{out} - \Delta X_{CV}^{in} \tag{4.8}$$

and dividing by Δt and letting $\Delta t \to 0$, we get

$$\frac{dX_S}{dt} = \frac{dX_{CV}}{dt} + \frac{dX_{CV}^{out}}{dt} - \frac{dX_{CV}^{in}}{dt} \tag{4.9}$$

These equations will be used in subsequent studies of continuity, energy, and momentum. The left-hand side of Eq. (4.9) is the rate of change of the total amount of any extensive property X within the moving system. The next term, dX_{CV}/dt, is the rate of change of that same property, but contained within the fixed control volume. The last two terms are the net rate of outflow of X passing through the control surface. So Eq. (4.9) states that *the difference between the rate of change of X* (e.g., mass) *within the system and that within the control volume is equal to the net rate of outflow from the control volume.*

4.7 EQUATION OF CONTINUITY

Although continuity is a strongly intuitive concept, it took early investigators a long time to formalize it. Pioneers in this effort include the Greek scientist Hero (or Heron) of Alexandria (*circa* 100 A.D.), the prodigious Italian artist and scientist Leonardo da Vinci (1452–1519), and Benedetto Castelli (1577–1643), a pupil of Galileo.

Let Fig. 4.8 represent a short length of a ***stream tube,*** which may be assumed, for practical purposes, to be a bundle of streamlines. Since the stream tube is bounded on all sides by streamlines and since there can be no net velocity normal to a streamline (Sec. 4.4), no fluid can leave or enter the stream tube except at the ends. The fixed volume between the two end sections is a control volume, of volume $\forall$ let us say. Using the relation of a system to a control volume developed in Sec. 4.6, and letting the general property X now be the mass m, Eq. (4.9) becomes

$$\frac{dm_S}{dt} = \frac{dm_{CV}}{dt} + \frac{dm_{CV}^{\text{out}}}{dt} - \frac{dm_{CV}^{\text{in}}}{dt} \tag{4.10}$$

But according to Newtonian physics (i.e., disregarding the possibility of converting mass to energy), the mass of a system must be conserved, so

$$\frac{dm_S}{dt} = 0 \tag{4.11}$$

In addition, because the volume $\forall$ of the control volume is fixed, $m_{CV} = \forall \bar{\rho}_{CV}$ where $\bar{\rho}_{CV}$ is the mean density within the control volume, so

$$\frac{dm_{CV}}{dt} = \forall \frac{d\bar{\rho}_{CV}}{dt} = \forall \frac{\partial \bar{\rho}_{CV}}{\partial t} \tag{4.12}$$

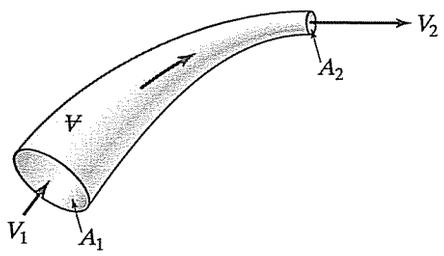


Figure 4.8
Portion of stream tube as control volume.

since $\bar{\rho}_{CV}$ can vary only with time within the control volume. Also, from Fig. 4.8, $\Delta m_{CV}^{\text{out}} = \rho_2 \Delta \forall_2 = \rho_2 A_2 V_2 \Delta t$, so that

$$\frac{dm_{CV}^{\text{out}}}{dt} = \rho_2 A_2 V_2 \tag{4.13}$$

and similarly,

$$\frac{dm_{CV}^{\text{in}}}{dt} = \rho_1 A_1 V_1 \tag{4.14}$$

Substituting Eqs. (4.11)–(4.14) into (4.10) and rearranging slightly, we obtain

$$\rho_1 A_1 V_1 - \rho_2 A_2 V_2 = \forall \frac{\partial \bar{\rho}_{CV}}{\partial t} \tag{4.15}$$

This is the general equation of continuity for flow through regions with fixed boundaries, in which $\partial \bar{\rho}_{CV}/\partial t$ is the time rate of change of the mean density of the fluid in $\forall$. The equation states that the net rate of mass inflow to the control volume is equal to the rate of increase of mass within the control volume.

For steady flow (Sec. 4.3), $\partial \bar{\rho}_{CV}/\partial t = 0$ in Eq. (4.15) and

Steady flow:

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 = \dot{m} \tag{4.16a}$$

$$\gamma_1 A_1 V_1 = \gamma_2 A_2 V_2 = g\dot{m} = G \tag{4.16b}$$

These are the continuity equations that apply to steady, compressible or incompressible flow within fixed boundaries.

If the fluid is incompressible, ρ = constant; hence $\rho_1 = \rho_2$ and $\partial \rho/\partial t = 0$ in Eq. (4.15), and thus

Incompressible flow:

$$A_1 V_1 = A_2 V_2 = Q \tag{4.17}$$

This is the continuity equation that applies to incompressible fluids for both steady and unsteady flow within fixed boundaries.[5]

Equations (4.16) and (4.17) are generally adequate for the analysis of flows in conduits with solid boundaries, but for the consideration of flow in space, as that of air around an airplane, for example, it is desirable to express the continuity equation in differential form, as indicated in Sec. 14.1. Or, for the case of unsteady flow of a liquid in an open channel (Fig. 4.4), the principle of conservation of mass indicates that the rate of flow past section 1 minus the rate of flow past section 2 is equal to the time rate of change of the volume of liquid $\forall$ contained in the channel between the two sections. Thus

$$Q_1 - Q_2 = \frac{d\forall}{dt} \tag{4.18}$$

[5] The continuity equations (4.16) and (4.17) apply to any stream tube in a flow system. Most commonly the continuity equation is applied to the stream tube that coincides with the boundaries of the flow.

EXERCISES

4.7.1 Water flows in a river. At 8 A.M. the flow past bridge 1 is 2150 cfs. At the same instant the flow past bridge 2 is 1800 cfs. At what rate is water being stored in the river between the two bridges at this instant? Assume zero seepage and negligible evaporation.

4.7.2 Water flows in a river. At 9 A.M. the flow past bridge 1 is 37.2 m^3/s. At the same instant the flow past bridge 2 is 26.9 m^3/s. At what rate is water being stored in the river between the two bridges at this instant? Assume zero seepage and negligible evaporation.

4.7.3 Gas is flowing in a long 9-in-diameter pipe from A to B. At section A the flow is 0.65 lb/sec while at the same instant at section B the flow is 0.72 lb/sec. The distance between A and B is 750 ft. Find the mean value of the time rate of change of the specific weight of the gas between sections A and B at that instant.

4.8 ONE-, TWO-, AND THREE-DIMENSIONAL FLOW

In true one-dimensional flow the velocity at all points has the same direction and (for an incompressible fluid) the same magnitude. Such a case is rarely of practical interest. However, we apply the term ***one-dimensional method of analysis*** to the flow between boundaries that are really three-dimensional, with the understanding that the "one dimension" is taken *along the central streamline* of the flow. We consider average values of velocity, pressure, and elevation across a section normal to this streamline to be typical of the flow as a whole. Thus we call the equation of continuity in Sec. 4.7 the one-dimensional equation of continuity, even though we can apply it to flow in conduits that curve in space and in which the velocity varies across sections normal to the flow. When we need high accuracy, in the following chapters, we will need to remember to refine the equations derived by the one-dimensional method of analysis in order to account for the variation in conditions across the flow section.

If all streamlines in the flow are plane curves and are identical in a series of parallel planes, we call the flow ***two-dimensional.*** In Fig. 4.9*a* the channel has a

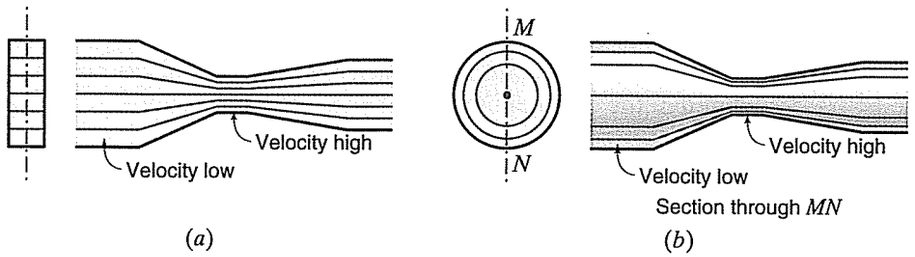


Figure 4.9
(*a*) Two-dimensional and (*b*) three-dimensional (axially symmetric) flow of an ideal fluid.

constant dimension perpendicular to the plane of the figure. Thus every cross section normal to the flow must be a rectangle of this constant width. The flow depicted in Fig. 4.9*b* is ***three-dimensional,*** although in this particular case the flow is also axially symmetric, which simplifies the analysis. A generalized three-dimensional flow, such as the flow of cool air from an air conditioning outlet into a room, is quite difficult to analyze. We often approximate such flows as two-dimensional or as axially symmetric flow. This offers the advantages that we can more easily draw diagrams describing the flow, and the mathematical treatment is much simpler.

4.9 THE FLOW NET

In the case of steady two-dimensional flow of an *ideal* fluid within any boundary configuration, we can represent and determine the streamlines and velocity distribution by a ***flow net,*** such as that shown in Fig. 4.10. This is a network of ***streamlines*** and lines normal (perpendicular) to them called ***equipotential lines.*** Flow nets help us visualize flow patterns, the streamlines indicate the mean flow directions, and the spacing between both sets of lines at any point is

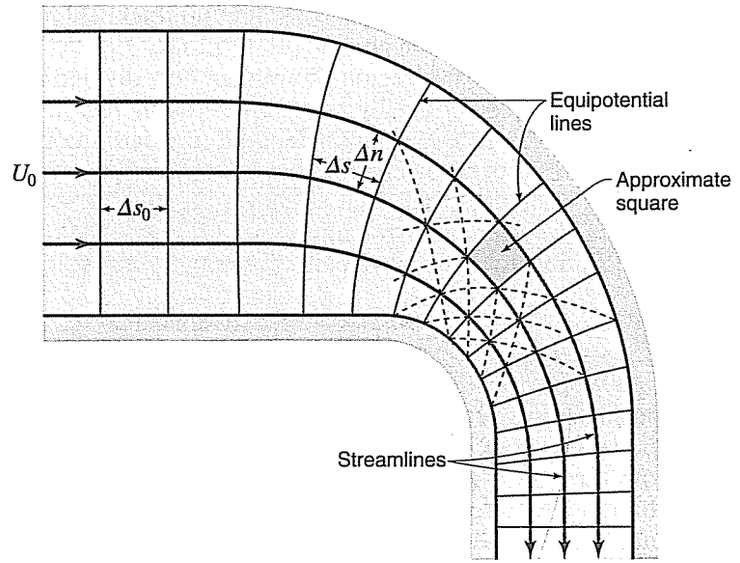


Figure 4.10
Flow net (two-dimensional flow).

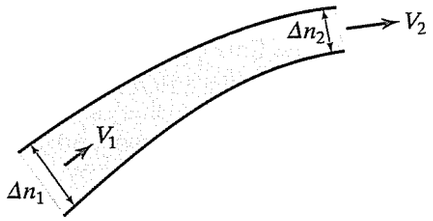


Figure 4.11

inversely proportional to the local flow velocity.[6] Furthermore, by sketching and adjusting streamlines and equipotential lines until they are approximately orthogonal at all points, we can graphically solve for flow patterns, and thus local flow directions and velocities. Thus, for example, the maximum velocity around the inside bend in Fig. 4.10 (at the smallest "square") is, from the ratio of the square sizes, about $2.7U_0$.

A fundamental property of the flow net is that it provides the one and only representation of the ideal flow within the given boundaries. It is also independent of the actual magnitude of the flow and, for the *ideal* fluid, it is the same whether the flow is in one direction or the reverse.

In a number of simple cases we can obtain mathematical expressions, known as ***stream functions*** (Sec. 14.4), from which we can plot streamlines. But even the most complex cases we can solve by plotting a flow net by a trial-and-error method. Although it is possible to construct nets for three-dimensional flow, we will restrict treatment here to the simpler two-dimensional net, which will more clearly illustrate the method. Consider the two-dimensional stream tube of Fig. 4.11. Assuming a constant unit thickness perpendicular to the paper, the continuity equation gives $V_1\Delta n_1 = V_2\Delta n_2$.

Consider next a region of uniform flow divided into a number of strips of equal width, separated by streamlines, as in Fig. 4.9*a*. Each strip represents a stream tube, and the flow is equally divided among the tubes. As the flow approaches a bend or obstruction, the streamline must curve so as to conform to the boundaries, but each stream tube still carries the same flow. Thus the spacing between all streamlines in the entire field is everywhere inversely proportional to the local velocities so that, for any section normal to the velocity,

$$V\Delta n = \text{constant} \tag{4.19}$$

When drawing the streamlines, we need to start by estimating not only the spacing between them but also their directions at all points. As an aid in the latter, we also draw the normal, or equipotential lines. As an analogy consider heat flow through a homogeneous material enclosed between perfectly insulated boundaries. We might consider the heat to flow along the equivalent of streamlines. As no heat can flow along a line of constant temperature, it follows that everywhere the heat must flow perpendicularly to isothermal lines. Likewise,

[6] See a mathematical basis for the flow net in Chap. 14, and particularly in Sec. 14.7.

streamlines must be everywhere perpendicular to equipotential lines. Because solid boundaries, across which there can be no flow, also represent streamlines, it follows that *equipotential lines must meet the boundaries everywhere at right-angles.*

If, as is usually most convenient, the equipotential lines are spaced the same distance apart as the streamlines in the region of uniform two-dimensional flow (as at the ends of Fig. 4.10), the flow net for that region is composed of perfect squares. In a region of deformed flow (as in the bend of Fig. 4.10) the quadrilaterals cannot remain square, but they will approach squares as the number of streamlines and equipotential lines are increased indefinitely by subdividing. It is frequently helpful, in regions where the deformation is marked, to create extra streamlines and equipotential lines spaced midway between the original ones.

In drawing a flow net, you will at first do a lot of erasing, but with some practice you will be able to sketch a net fairly easily to represent any boundary configuration. We can even construct an approximate flow net for cases where one solid boundary does not exist and the fluid extends laterally indefinitely, as in the flow around an immersed object. Such cases reveal an advantage of the flow net that is not evident from Fig. 4.10. For flow between confining solid boundaries we can always determine the mean velocity across any section by dividing the total flow by the section area. For flow around an immersed object, as in Fig. 4.12, there is no fixed area by which to divide a definite flow, but the flow net in combination with Eq. (4.19) provides a good means of estimating velocities in the surrounding region. With increasing distance from the body's centerline the deflection of streamlines around the body reduces, until the deflection becomes negligible; this distance or deflection must be estimated in order to

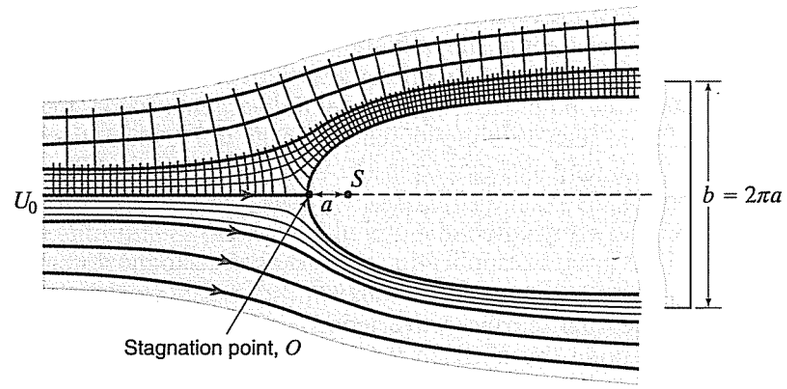


Figure 4.12
Two-dimensional flow of a frictionless fluid past a solid[7] whose surface is perpendicular to the plane of the paper. Streamlines or path lines for steady flow.

[7] This surface shape is the boundary between the given flow field and that issuing from a source of strength $Q = bdU_0$ located at *S*, where *d* is the source length perpendicular to the figure (see Prob. 14.14).

draw the flow net, and reasonable estimates will yield closely similar velocities near the body.

Where a channel is curved, the equipotential lines must diverge because they radiate from centers of curvature. The distance between the associated streamlines must vary in the same way as that between the equipotential lines. Therefore, as in Fig. 4.10, the areas are smallest along the inner radius of the bend and increase toward the outside.

We can check the accuracy of the final flow net by drawing diagonals, as indicated by a few dashed lines in Fig. 4.10. If the net is correct, these dashed lines will also form a network of lines that cross each other at right angles and produce areas that approach squares in shape.

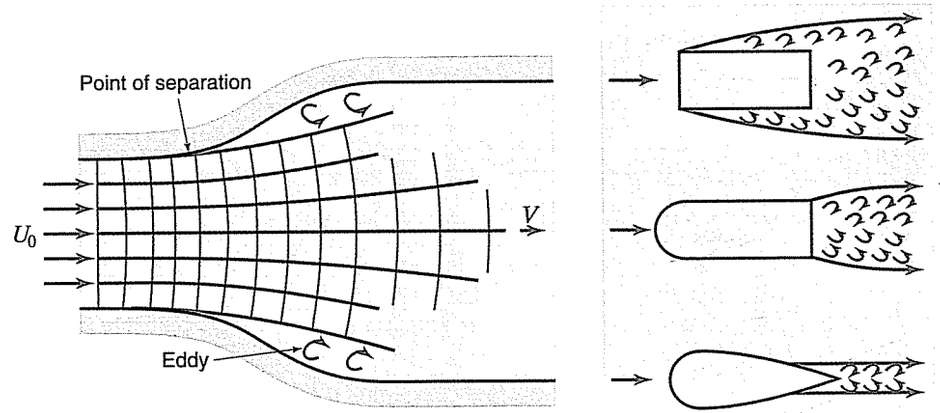
4.10 USE AND LIMITATIONS OF THE FLOW NET

Although the flow net is based on an ideal frictionless fluid, we can apply it to the flow of a real fluid within certain limits. Such limits are dictated by the extent that factors which the ideal-fluid theory neglects affect the real fluid. The principal factor of this type is fluid friction.

The viscosity effects of a real fluid are most pronounced at or near a solid boundary and diminish rapidly with distance from the boundary. Hence, for an airplane or a submerged submarine, we can consider the fluid as frictionless, except when very close to the object. The flow net always indicates a velocity next to a solid boundary, whereas a real fluid must have zero velocity adjacent to a wall due to the no-slip condition (Sec. 2.11). The region in which the velocity is so distorted, however, is confined to a relatively thin layer called the ***boundary layer*** (Secs. 8.8–8.10 and 9.2–9.4), outside of which the real fluid behaves very much like the ideal fluid.

The effect of the boundary friction is minimized when the streamlines are converging, but in a diverging flow there is a tendency for the streamlines not to follow the boundaries if the rate of divergence is too great. In a sharply diverging flow, such as is shown schematically in Fig. 4.13, there may be a ***separation*** of the boundary layer from the wall, resulting in ***eddies*** (Sec. 4.2) and even reverse flow in that region (Fig. 9.8). The flow is badly disturbed in such a case, and the flow net may then be of limited value.

A practical application of the flow net is to the flow around a body, as shown in Fig. 4.12. An example of this is the upstream portion of a bridge pier below the surface where surface wave action is not a factor. Except for a thin layer adjacent to the body, this diagram represents the flow in front of and around the sides of the body. The central streamline branches at the forward tip of the body to form two streamlines along the walls. At the forward tip the velocity must be zero, so we call this point a ***stagnation point.*** Other common applications are to flows over spillways, and to seepage flows through earth dams and through the ground under a concrete dam. In the first two of these cases, the flow has a ***free surface*** at atmospheric pressure. To draw flow nets for free surface flows, we must make use of more advanced principles that are not covered in this text.



(*a*) Eddy formation in a diverging channel (*b*) Turbulent wakes

Figure 4.13
Separation in diverging flow.

Considering the limitations of the flow net in diverging flow, we can see that, while the flow net gives a fairly accurate picture of the velocity distribution in the region near the upstream part of any solid body, it may give little information concerning the flow conditions near the rear because of the possibility of separation and eddies. We call the disturbed flow to the rear of a body a ***turbulent wake*** (Fig. 4.13*b*). We can greatly reduce the space occupied by the wake by streamlining the body, i.e., by giving the body a long slender tail, which tapers to a sharp edge for two-dimensional flow or to a point for three-dimensional flow.

SAMPLE PROBLEM 4.2 Figure 4.12 represents flow toward and around a bridge pier where $b = 5$ ft and $U_0 = 10$ fps. (*a*) Make a plot of the velocity along the flow centerline to the left of the solid, and along the boundary of the solid. (*b*) By what percentage does the maximum velocity along the boundary exceed the uniform velocity? (*c*) How far from the stagnation point does a velocity of 7.5 fps occur?

Solution

Eq. (4.19): $$V\Delta n = \text{const.} = U_0 \Delta n_0$$

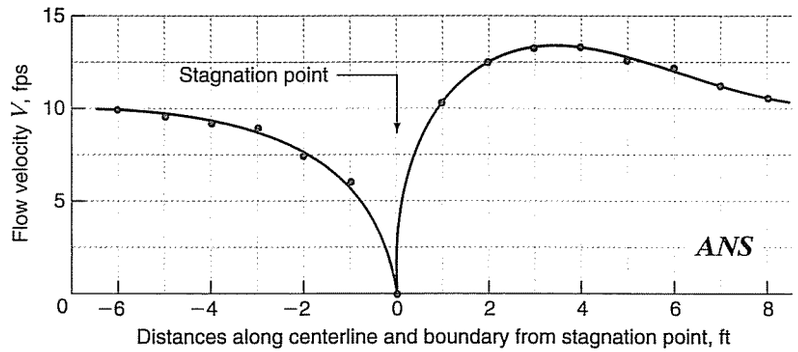
So $$V = (\Delta n_0/\Delta n)10 \text{ fps}.$$

Use $b = 5$ ft to scale 1 ft distances along the centerline and around the boundary of the solid. On Fig. 4.12 measure the net "square" sizes, in both the flow (ΔL) and perpendicular (ΔW) directions, using three or four squares where appropriate

and taking the average. Calculate Δn and V as shown in the table:

Distance from stagnation pt, ft	−6	−5	−4	−3	−2	−1	0	1	2	3	4	5	6	7	8
Average ΔL, mm	0.98	1.02	1.06	1.06	1.17	1.25	—	0.88	0.70	0.67	0.66	0.73	0.75	0.78	0.84
Average ΔW, mm	0.88	0.91	0.94	1.00	1.30	1.80	—	0.95	0.80	0.74	0.75	0.76	0.79	0.90	0.93
$\Delta n = \frac{1}{2}(\Delta L + \Delta W)$, mm	0.93	0.97	1.00	1.03	1.24	1.53	—	0.92	0.75	0.71	0.71	0.74	0.77	0.84	0.88
$\Delta n_0/\Delta n = 0.93/\Delta n$	1.00	0.96	0.93	0.90	0.75	0.61	—	1.02	1.24	1.32	1.32	1.25	1.21	1.11	1.05
$V = 10(\Delta n_0/\Delta n)$, fps	10.0	9.6	9.3	9.0	7.5	6.1	0	10.2	12.4	13.2	13.2	12.5	12.1	11.1	10.5

(*a*)



(*b*) $V_{max}/U_0 = 13.3/10.0 = 1.33$

Therefore V_{max} is 33% greater than U_0 ***ANS***

(*c*) From the plot shown, a velocity of 7.5 fps occurs at about −1.9 ft and +0.4 ft from the stagnation point ***ANS***

Note: If the flow net had been constructed perfectly, the respective average ΔL and ΔW values would have been identical. Even so, the results obtained here are quite accurate, because the respective values of ΔL and ΔW were averaged. This problem can also be solved analytically using principles of hydrodynamics, which yield $V_{max}/U_0 = 1.260$ (see, e.g., Prob. 14.14).

EXERCISES

4.10.1 An incompressible ideal fluid flows at 12 L/s through a circular 160-mm-diameter pipe into a conically converging nozzle like that of Sample Prob. 4.4 (diameter at B is 80 mm). Determine the average velocity of flow at sections D and B.

4.10.2 Figure X4.10.2 shows the flow net for two-dimensional flow from a rounded, long-slotted exit from a tank. If $U_0 = 1.8$ m/s, what is the approximate flow velocity at A?

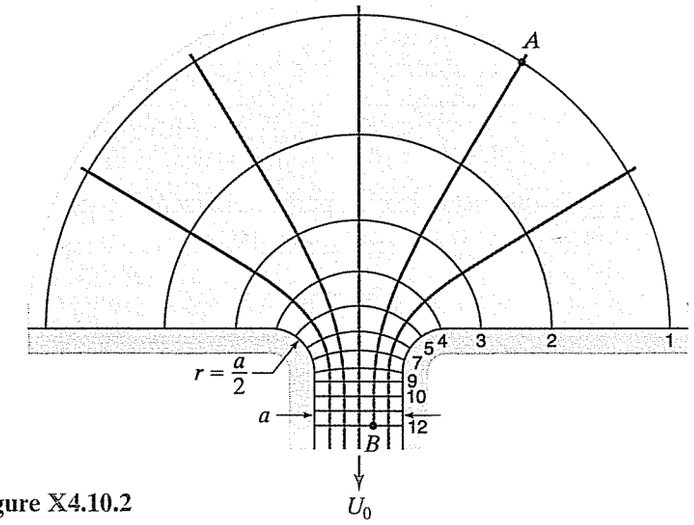


Figure X4.10.2

4.10.3 Given that U_0 in Fig. 4.10 is 6.0 fps, find approximately (*a*) the maximum velocity in the bend and (*b*) the uniform velocity in the downstream section.

4.10.4 Given that U_0 in Fig. 4.12 is 4 m/s, find approximately (*a*) the maximum and (*b*) the minimum velocity on the body surface.

4.11 FRAME OF REFERENCE IN FLOW PROBLEMS

In flow problems we are really concerned with the *relative* velocity between the fluid and the body. It makes no difference whether the body is at rest and the fluid flows past it or whether the fluid is at rest and the body moves through the fluid. There are thus two frames of reference. In one the observer (or the camera) is at rest with respect to the solid body. If the observer at rest with respect to a bridge pier views a steady flow past it or is on a ship moving at constant velocity through still water, the streamlines appear to him to be unchanging and therefore the flow is steady. But if he floats with the current past the pier or views a ship going by while he stands on the bank, the flow pattern that he observes is changing with time. Then the flow is unsteady.

The same flow may therefore be either steady or unsteady according to the frame of reference. The case that is usually of more practical importance is steady, ideal flow, for which the streamlines and path lines are identical. In unsteady flow streamlines and path lines are entirely different from each other, and they also bear no resemblance to those of steady flow.

4.12 VELOCITY AND ACCELERATION IN STEADY FLOW

In a typical three-dimensional flow field, the velocities may be everywhere different in magnitude and direction. Also, the velocity at any point in the field may change with time. Let us first consider the case where the flow is steady and

thus independent of time. If the velocity of a fluid particle has components u, v,[8] and w parallel to the x, y, and z axes, then, for steady flow,

$$u_{st} = u(x, y, z) \tag{4.20a}$$
$$v_{st} = v(x, y, z) \tag{4.20b}$$
$$w_{st} = w(x, y, z) \tag{4.20c}$$

Applying the chain rule of partial differentiation, the acceleration of the fluid particle for steady flow can be expressed as

$$\mathbf{a}_{st} = \frac{d}{dt}\mathbf{V}(x, y, z) = \frac{\partial \mathbf{V}}{\partial x}\frac{dx}{dt} + \frac{\partial \mathbf{V}}{\partial y}\frac{dy}{dt} + \frac{\partial \mathbf{V}}{\partial z}\frac{dz}{dt} \tag{4.21}$$

where $$|\mathbf{V}| = \sqrt{u^2 + v^2 + w^2}$$

Noting that $dx/dt = u$, $dy/dt = v$, and $dz/dt = w$,

$$\mathbf{a}_{st} = u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z} \tag{4.22}$$

This vector equation can be written as three scalar equations:

$$(a_x)_{st} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \tag{4.23a}$$
$$(a_y)_{st} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \tag{4.23b}$$
$$(a_z)_{st} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \tag{4.23c}$$

These equations show that even though the flow is steady, the fluid may possess an acceleration by virtue of a change in velocity with change in position. This type of acceleration we commonly refer to as ***convective acceleration.*** With incompressible fluid flow, there is a convective acceleration wherever the effective flow area changes along the flow path. This is also true for compressible fluid flow, but, in addition, convective acceleration of a compressible fluid occurs wherever the density varies along the flow path regardless of any changes in the effective flow area.

At times we find it convenient to superimpose the coordinate system on the streamline pattern in such a way that the x axis is tangential to the streamline at a particular point of interest. In such a case we shall let s indicate distance along the streamline. Thus $\mathbf{V} = \mathbf{V}(s)$, and, since the perpendicular velocity components in Eq. (4.22) are zero, we can conveniently express the acceleration of the fluid particle along the streamline at this point as

$$\mathbf{a}_{st} = V\frac{\partial \mathbf{V}}{\partial s} \tag{4.24}$$

[8] This text uses a rounded lower case v (vee) to help distinguish it from the capital V and from the Greek ν (nu) used for kinematic viscosity.

In the terminology of curvilinear motion, we refer to this as the ***tangential acceleration.*** In uniform flow with ρ = constant this acceleration is zero.

At this point in our discussion we should recall that a particle moving steadily along a curved path has a ***normal acceleration*** a_n toward the center of curvature of the path. From mechanics,

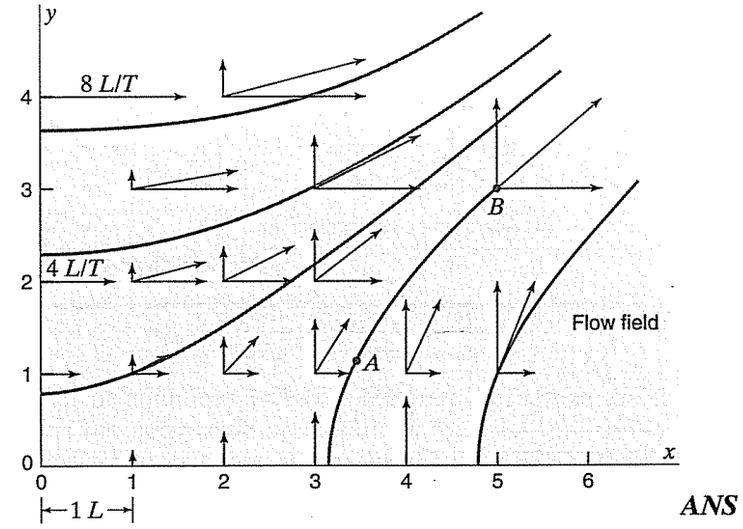
$$a_n = \frac{V^2}{r} \tag{4.25}$$

where r is the radius of the path. A particle moving on a curved path will always have a normal acceleration, regardless of its behavior in the tangential direction.

SAMPLE PROBLEM 4.3 A two-dimensional flow field is given by $u = 2y$, $v = x$. (*a*) Sketch the flow field. (*b*) Derive a general expression for the velocity and acceleration (x and y are in units of length L; u and v are in units of L/T). (*c*) Find the acceleration in the flow field at point A ($x = 3.5$, $y = 1.2$).

Solution

(*a*) Velocity components u and v are plotted to scale, and streamlines are sketched tangentially to the resultant velocity vectors. This yields the following general picture of the flow field:



ANS

(*b*)
$$V = (u^2 + v^2)^{1/2} = (4y^2 + x^2)^{1/2} \qquad \textbf{ANS}$$

Eq. (4.23*a*):
$$a_x = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = 2y(0) + x(2) = 2x$$

Eq. (4.23*b*):
$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = 2y(1) - x(0) = 2y$$
$$a = (a_x^2 + a_y^2)^{1/2} = (4x^2 + 4y^2)^{1/2} = 2(x^2 + y^2)^{1/2} \qquad \textbf{ANS}$$

(*c*) At A (3.5, 1.2),

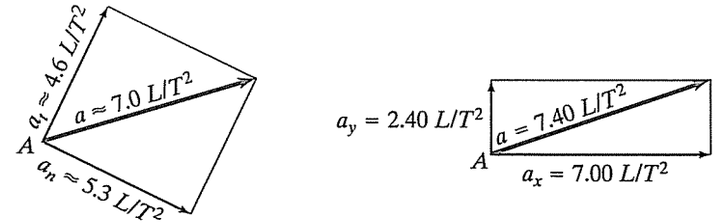
$$(a_A)_x = 2x = 2(3.5) = 7.00\ L/T^2; \quad (a_A)_y = 2y = 2(1.2) = 2.40\ L/T^2$$

and $$a_A = [(a_A)_x^2 + (a_A)_y^2]^{1/2} = [(7.00)^2 + (2.40)^2]^{1/2} = 7.40\ L/T^2 \qquad \textbf{ANS}$$

Rough check. Imagine a velocity vector at point A. This vector would have a magnitude approximately midway between that of the adjoining vectors, or $V_A \approx 4\ L/T$. The radius of curvature of the sketched streamline at A is roughly $3L$. Thus $(a_A)_n \approx 4^2/3 \approx 5.3\ L/T^2$. The tangential acceleration of the particle at A may be approximated by noting that the velocity along the streamline increases from about 3.2 L/T, where it crosses the x axis, to about 7.8 L/T at B. The distance along the streamline between these two points is roughly 4 L. Hence a very approximate value of the tangential acceleration at A is

$$(a_A)_t = V\frac{\partial V}{\partial s} \approx 4\left(\frac{7.8 - 3.2}{4}\right) \approx 4.6\ L/T^2$$

Vector diagrams of these roughly-computed normal and tangential acceleration components plotted below allow us to compare them with the true acceleration as given by the analytic expressions. In Chap. 14 we shall prove that the flow in this example is that of an incompressible fluid.



(*a*) Approximate vector diagram (*b*) True vector diagram

Acceleration at A

EXERCISES

4.12.1 A flow field is defined by $u = 2$, $v = 3$, and $w = 4$. What is the velocity of flow? Specify units in terms of L and T.

4.12.2 The velocity along a streamline lying on the x axis is given by $u = 6 + x^{0.4}$. What is the convective acceleration at $x = 5$? Specify units in terms of L and T. Assuming the fluid is incompressible, is the flow converging or diverging?

4.12.3 A flow field is defined by $u = 2x$ and $v = y$. Derive expressions for the x and y components of acceleration. Find the magnitude of the velocity and acceleration at the point (3, 2). Specify units in terms of L and T.

4.12.4 A flow field is defined by $u = 2y$ and $v = x$. Derive expressions for the x and y components of acceleration. Find the magnitude of the velocity and acceleration at the point (3, 1). Specify units in terms of L and T.

4.12.5 A flow field is defined by $u = 2y$ and $v = xy$. Derive expressions for the x and y components of acceleration. Find the magnitude of the velocity and acceleration at the point (2, 3). Specify units in terms of L and T.

4.12.6 The velocity along a circular streamline of radius 4 ft is 2.2 fps. Find the normal and tangential components of the acceleration if the flow is steady.

4.12.7 The velocity along a circular streamline of radius 1.5 m is 0.75 m/s. Find the normal and tangential components of the acceleration if the flow is steady.

4.12.8 A large tank contains an ideal liquid which flows out of the bottom of the tank through a 4-in-diameter hole. The rate of steady outflow is 5 cfs. Assume that the liquid approaches the center of the hole radially. Find the velocities and convective accelerations at points that are 2.5 and 5 ft from the center of the hole.

4.13 VELOCITY AND ACCELERATION IN UNSTEADY FLOW

For unsteady flow, Eqs. (4.20*a*–*c*) take the form

$$u = u(x, y, z, t)\ldots \tag{4.26}$$

Following a similar procedure to that of the preceding section, we obtain the vector equation

$$\mathbf{a} = \left(u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z}\right) + \frac{\partial \mathbf{V}}{\partial t} \tag{4.27}$$

and the following set of scalar equations

$$a_x = \left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) + \frac{\partial u}{\partial t} \tag{4.28a}$$

$$a_y = \left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) + \frac{\partial v}{\partial t} \tag{4.28b}$$

$$a_z = \left(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) + \frac{\partial w}{\partial t} \tag{4.28c}$$

In the set of equations (4.28*a*)–(4.28*c*) we recognize the three terms in parentheses as the convective accelerations (Sec. 4.12). The $\partial \mathbf{V}/\partial t$, $\partial u/\partial t$, $\partial v/\partial t$, and $\partial w/\partial t$ terms, however, represent the acceleration caused by the unsteadiness of the flow; we commonly refer to this latter type of acceleration as the ***local acceleration.***

If we let s represent distance along an instantaneous streamline, in the same manner as the previous section, we now have $\mathbf{V} = \mathbf{V}(s, t)$, and the tangential acceleration of a fluid particle along the streamline is

$$\mathbf{a} = V\frac{\partial \mathbf{V}}{\partial s} + \frac{\partial \mathbf{V}}{\partial t} \tag{4.29}$$

The first term on the right-hand side of this equation is the convective acceleration, which becomes zero in uniform flow (straight and parallel streamlines) with ρ = constant.

SAMPLE PROBLEM 4.4 Figure S4.4 is of a cross section along the centerline of a circular pipe with a conically converging nozzle. An incompressible ideal fluid flows through at $Q = (0.1 + 0.05t)$ cfs, where t is in sec. Find the average velocity and acceleration of the flow at points D and B when $t = 5$ sec.

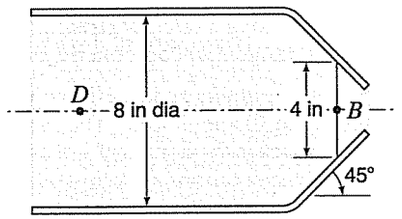
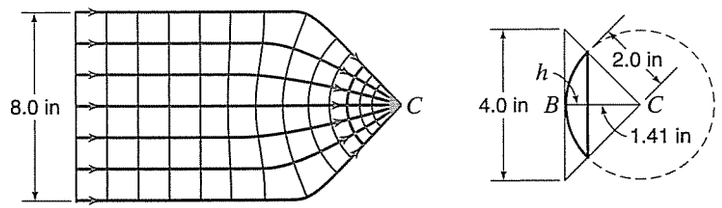


Figure S4.4

Solution

As a first step we sketch an approximate flow net to provide a general picture of the flow. We note that the flow is symmetric about the pipe axis (axisymmetric flow), so *the net is not a true two-dimensional flow net* (see Fig. 4.9).



Since D and B are both on the pipe axis, $v = 0$ and $w = 0$ due to symmetry, so Eqs. (4.28) for these points reduce to

$$a_x = u\frac{\partial u}{\partial x} + \frac{\partial u}{\partial t}, \qquad a_y = 0, \qquad a_z = 0$$

At section D the streamlines are parallel and hence the area at right angles to the velocity vectors is a plane circle,

$$A_D = \frac{\pi}{4}\left(\frac{8}{12}\right)^2 = 0.349 \text{ ft}^2$$

So
$$u = \frac{Q}{A_D} = \frac{0.1 + 0.05t}{0.349} = \frac{2 + t}{6.98}$$

and
$$\frac{\partial u}{\partial x} = 0, \quad \frac{\partial u}{\partial t} = \frac{1}{6.98}$$

Thus at $t = 5$ sec: $V_D = u = \dfrac{2 + 5}{6.98} = 1.003$ fps ***ANS***

and
$$a_D = a_x = 1.003(0) + \frac{1}{6.98} = 0.1432 \text{ ft/sec}^2 \qquad \textbf{\textit{ANS}}$$

At section B, however, the perpendicular flow area is the partial spherical surface through B, with center C and radius $r = 2$ in (see sketch). By table lookup, or by integration, this area is $2\pi rh$, where $h = r - r\cos 45° = 0.293r$. Thus $A_B = 2\pi r(0.293r) = 1.840r^2$.

On the centerline near B, $\quad u = \dfrac{Q}{A_B} = \dfrac{0.1 + 0.05t}{1.840r^2} = \dfrac{2 + t}{36.8r^2}$

and since x = constant $- r$,

$$\frac{\partial u}{\partial x} = -\frac{\partial u}{\partial r} = -\left[\frac{-2(2 + t)}{36.8r^3}\right] = \frac{2 + t}{18.40r^3}$$

and
$$\frac{\partial u}{\partial t} = \frac{1}{36.8r^2}$$

Thus at $r = 2$ in and $t = 5$ sec:

$$V_B = u = \frac{2 + 5}{36.8(2/12)^2} = 6.85 \text{ fps} \qquad \textbf{\textit{ANS}}$$

and
$$a_B = a_x = 6.85\left[\frac{2 + 5}{18.40(2/12)^3}\right] + \frac{1}{36.8(2/12)^2}$$

$$= 563 \text{ (convective)} + 0.978 \text{ (local)}$$

$$= 564 \text{ ft/sec}^2 \qquad \textbf{\textit{ANS}}$$

Note: For the flow net shown in the sketch, the velocity at C is infinite because the flow area at that point is zero. This, of course, cannot occur; in the real case a jet somewhat similar to that of Fig. 11.13 will form downstream of the nozzle opening.

EXERCISES

4.13.1 A flow is defined by $u = 2(1 + t)$, $v = 3(1 + t)$, $w = 4(1 + t)$. What is the velocity of flow at the point (3, 2, 4) at $t = 2$? What is the acceleration at that point at $t = 2$? Specify units in terms of L and T.

4.13.2 A two-dimensional flow field is given by $u = 2 + xy + 3t^2$, $v = 2xy^2 + t$. Find the velocity and acceleration of a particle of fluid at point (2, 3) at $t = 4$. Specify units in terms of L and T.

4.13.3 The flow velocity in fps along a circular streamline of radius 3 ft is $0.6 + 1.2t$. Find the normal and tangential components of the acceleration when $t = 1.5$ sec.

4.13.4 The flow velocity in m/s along a circular streamline of radius 1.5 m is $0.4 + 0.6t$. Find the normal and tangential components of the acceleration when $t = 1.2$ s.

PROBLEMS

4.1 The velocity of a liquid ($s = 1.26$) in a 3-in pipeline is 2.4 fps. Calculate the rate of flow in cfs, gal/min, slugs/sec, and lb/sec.

4.2 Carbon dioxide flows in a 2-in by 3-in duct at a pressure of 46 psi and a temperature of 80°F. If the atmospheric pressure is 13.8 psia and the flow velocity is 10 fps, calculate the weight flow rate.

4.3 Nitrogen at 40°C and under a pressure of 3000 mb abs flows in a 350-mm-diameter conduit at a mean velocity of 8 m/s. Find the mass flow rate.

4.4 Velocities in an 8-in-diameter circular conduit, measured at radii of 0, 1.44, 2.60, and 3.48 in, were 20.3, 19.7, 17.7, and 14.5 fps, respectively. Find approximate values (graphically) of the volume flow rate and the mean velocity. Also determine the ratio of the mean velocity to the maximum velocity.

4.5 Velocities in a 200-mm-diameter circular conduit, measured at radii of 0, 36, 65, and 87 mm, were 7.0, 6.8, 6.1, and 5.0 m/s, respectively. Find approximate values (graphically) of the volume flow rate and the mean velocity. Also determine the ratio of the mean velocity to the maximum velocity.

4.6 Gas flows at a steady rate in a 120-mm-diameter pipe that enlarges to a 180-mm-diameter pipe. (*a*) At a certain section of the 120-mm pipe the density of the gas is 165 kg/m^3 and the velocity is 15 m/s. At a certain section of the 180-mm pipe the velocity is 10 m/s. What must be the density of the gas at that section? (*b*) If these same data were given for the case of unsteady flow at a certain instant, could the problem be solved? Discuss.

4.7 A compressible fluid flows in a 20-in-diameter leaky pipe. Measurements are made simultaneously at two points A and B along the pipe that are 32,000 ft apart. Two sets of measurements are taken with an interval of exactly 45 min between them. The data are:

Time	ρ_1, slug/ft^3	V_1, ft/sec	ρ_2, slug/ft^3	V_2, ft/sec
0	0.520	65	0.608	54
45 min	0.616	51	0.727	40

Assuming ρ varies linearly with respect to time and distance, compute the approximate average mass rate of leakage between A and B.

4.8 Refer to Fig. X4.10.2. If a is 3 in and U_0 is 10 fps, approximately how long will it take a particle to move from point A to point B on the same streamline? (*Note:* Between each pair of equipotential lines, measure Δs, and then compute the average velocity and time increment.)

4.9 Repeat Prob. 4.8 using the following data: $a = 150$ mm and $U_0 = 0.5$ m/s. Find also the approximate velocity where the flow crosses equipotential line 3.

4.10 Make an approximate plot of the frictionless velocity (relative to U_0) along both the inner and the outer boundaries of Fig. 4.10. By what percent is the ideal maximum inner velocity greater than the ideal minimum outer velocity?

4.11 Consider the two-dimensional flow about a 2-in-diameter cylinder. Sketch the flow net for the ideal flow around one quarter of the cylinder. Start with a uniform net of $\frac{1}{2}$-in squares, and fill in with $\frac{1}{4}$-in squares where desirable. (*Note:* We can prove by classical hydrodynamics that the velocity tangent to the cylinder at a point 90° from the stagnation point is twice the uniform velocity.) From the flow net, determine the velocities (relative to U_0) along the center streamline from a point upstream where the velocity is uniform to the stagnation point, and then along the boundary of the cylinder from the stagnation point to the 90° point; plot them vs distance. By plotting a second curve on the same graph, compare the result thus obtained with the values given by the equation $V = 2U_0 \sin\theta$, where U_0 is the undisturbed stream velocity and θ is the angle subtended by the arc from the stagnation point to any point on the cylinder where V is desired.

4.12 An ideal liquid flows out the bottom of a large tank through a 100-mm-diameter hole at a steady rate of 0.80 m^3/s. Assume the liquid approaches the center of the hole radially. Find the velocities and convective accelerations at points 0.75 and 1.5 m from the center of the hole.

4.13 A flow field is defined by $u = 3y$, $v = 2xy$, and $w = 5z$. Derive expressions for the x, y, and z components of acceleration. Find the magnitude of the velocity and acceleration at the point (1, 2, 1). Specify units in terms of L and T.

4.14 Sketch the flow field defined by $u = 0$, $v = 3xy$, and derive expressions for the x and y components of acceleration. Find the acceleration at the point (2, 2). Specify units in terms of L and T.

4.15 Sketch the flow field defined by $u = 3y$, $v = 2$, and derive expressions for the x and y components of acceleration. Find the magnitude of the velocity and acceleration for the point having the coordinates (3, 4). Specify units in terms of L and T.

4.16 (*a*) Sketch the flow field defined by $u = -2y$, $v = 3x$, and derive expressions for the x and y components of acceleration. (*b*) As in Sample Prob. 4.3, find approximate values of the normal and tangential accelerations of the particle at the point (2, 3). Specify units in terms of L and T. (*c*) Compare the value of $(a_n^2 + a_t^2)^{1/2}$ with the computed value $(a_x^2 + a_y^2)^{1/2}$.

4.17 The steady flow rate in each of the four stream tubes of Fig. 4.10 is 15 cfs per foot perpendicular to the plane of the figure. By scaling, the dimensions of the shaded "square" have been found to be 1.65 ft wide on the upstream face, 1.53 ft wide on the downstream face, and 1.67 ft along the flow line through its center; the radius of that flow line measures 11.1 ft. Find the normal, tangential, and resultant accelerations of a fluid particle at the center of the shaded area.

4.18 Figure P4.18 represents a two-dimensional stream tube drawn to scale. If the flow rate is 25 m^3/s per meter perpendicular to the plane of the sketch, determine approximate values of the normal and tangential accelerations of a fluid particle at C. What is the resultant acceleration of a particle at C?

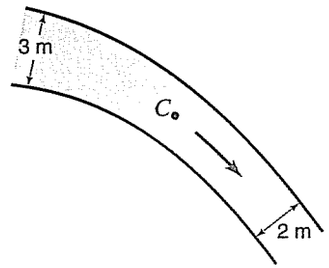


Figure P4.18

4.19 A large tank contains an ideal liquid which flows out of the bottom through a 4-in-diameter hole. The outflow rate $Q = 8 - 0.5t$, where Q is in cfs and t is in sec. Assume the liquid approaches the center of the hole radially. Find the local acceleration at a point 2 ft from the center of the hole at times $t = 5$ sec and 10 sec. What is the total acceleration at a point 3 ft from the center of the hole at $t = 10$ sec?

4.20 An ideal liquid flows out of the bottom of a large tank through an 80-mm-diameter hole. The outflow rate $Q = 0.4 - 0.02t^{0.5}$, where Q is in m^3/s and t is in s. Assume the liquid approaches the center of the hole radially. Find the local, convective, and total accelerations at a point 0.5 m from the center of the hole at time = 12 s.

4.21 Refer to the two-dimensional stream tube drawn to scale in Fig. P4.18. If the flow rate is $(18 - 4t)$ m^3/s per meter perpendicular to the plane of the sketch, with t in s, find approximate values of the normal, tangential, and total accelerations of a fluid particle at A when $t = 3$ s.

4.22 Assume that the streamlines for a two-dimensional flow of a frictionless incompressible fluid against a flat plate normal to the initial velocity may be represented by the equation xy = constant and that the flow is symmetrical about the plane through $x = 0$. A different streamline may be plotted for each value of the constant. Plot streamlines for values of the constant of 16, 64, and 128.

4.23 For the case in Prob. 4.22, we can show that the velocity components at any point are $u = ax$, and $v = -ay$, where a is a constant. Thus the actual velocity is $V = a\sqrt{x^2 + y^2} = ar$, where r is the radius to the origin. Let $a = \frac{1}{3}$. Draw curves of equal velocity for values of 2, 4, 6, 8, and 10 fps. How does the velocity vary along the surface of the plate?

4.24 For three-dimensional flow with the y axis as the centerline, assume that the equation for the bounding streamline of a jet impinging vertically downward on a flat plate is $x^2y = 64$. (*a*) Plot the flow showing the centerline and bounding streamlines of the jet. (*b*) What is the approximate average velocity in the vertical jet at $y = 10$ if the average velocity in the vertical jet is 5.0 m/s at $y = 16$? (*c*) For the above conditions find the approximate velocity along the plate at $r = 12$, 24, and 36.

CHAPTER 5
Energy in Steady Flow

In this chapter we shall approach flow from the viewpoint of energy considerations. The first law of thermodynamics tells us that energy can be neither created nor destroyed. But it can, of course, be changed in form. It follows that all forms of energy are equivalent.

In Secs. 5.5–5.7 we shall derive flow equations based on such energy considerations, but before this, in Secs. 5.2 and 5.3 we shall see how we can derive some of these equations from Newton's second law.

First we shall introduce the various forms of energy present in fluid flow.

5.1 ENERGIES OF A FLOWING FLUID

Kinetic Energy

A body of mass m when moving at a velocity V possesses a kinetic energy, KE = $\frac{1}{2}mV^2$. Thus if a fluid were flowing with all particles moving at the same velocity, its kinetic energy would also be $\frac{1}{2}mV^2$; for unit *weight* of the fluid we can write this as

$$\frac{\text{KE}}{\text{Weight}} = \frac{\frac{1}{2}mV^2}{\gamma\forall} = \frac{\frac{1}{2}(\rho\forall)V^2}{\rho g\forall} = \frac{V^2}{2g} \tag{5.1a}$$

where $\forall$ represents the volume of the fluid mass. In BG units we express $V^2/2g$ in ft·lb/lb = ft and in SI units as N·m/N = m. Similarly,

$$\frac{\text{KE}}{\text{Mass}} = \frac{\frac{1}{2}mV^2}{m} = \frac{V^2}{2} \tag{5.1b}$$

and

$$\frac{\text{KE}}{\text{Volume}} = \frac{\frac{1}{2}mV^2}{\forall} = \frac{\frac{1}{2}(\rho\forall)V^2}{\forall} = \frac{\rho V^2}{2} \tag{5.1c}$$

The units of $V^2/2$ of course are ft^2/sec^2 in BG units or m^2/s^2 in SI units. The units of $\rho V^2/2$ are lb/ft^2 or N/m^2, which are units of pressure.

In most situations the velocities of the different fluid particles crossing a section are not the same, so it is necessary to integrate all portions of the stream to obtain the true value of the kinetic energy. It is convenient to express the true value in terms of the mean velocity V and a factor α (alpha), known as the ***kinetic-energy correction factor.*** Then

$$\frac{\text{True KE}}{\text{Weight}} = \alpha \frac{V^2}{2g} \tag{5.2}$$

In order to obtain an expression for α, consider the case where the axial components of the velocity vary across a section, as in Fig. 4.1*b*. If u is the local axial velocity component at a point, the mass flow per unit of time through an elementary area dA is $\rho dQ = \rho u\, dA$. Thus the true flow of kinetic energy per unit of time across area dA is $\frac{1}{2}(\rho u\, dA)u^2 = \frac{1}{2}\rho u^3 dA$. The weight rate of flow through dA is $\gamma dQ = \rho g u\, dA$. Thus, for the entire section,

$$\frac{\text{True KE/time}}{\text{Weight/time}} = \frac{\text{true KE}}{\text{weight}} = \frac{\frac{1}{2}\rho \int u^3 dA}{\rho g \int u\, dA} = \frac{\int u^3 dA}{2g \int u\, dA} \tag{5.3}$$

Comparing Eq. (5.3) with Eqs. (5.2) and (4.3), we get

$$\alpha = \frac{1}{V^2}\frac{\int u^3 dA}{\int u\, dA} = \frac{1}{AV^3}\int u^3\, dA \tag{5.4}$$

and we get the same result if we use True KE/Mass, where mass flow rate is $\rho \int dQ = \rho \int u\, dA$, or if we use True KE/Volume, where the volume flow rate is $\int dQ = \int u\, dA$.

As the average of cubes is always greater than the cube of the average, the value of α will always be more than 1. The greater the variation in velocity across the section, the larger will be the value of α. For laminar flow in a circular pipe, $\alpha = 2$ (see Sample Prob. 5.1); for turbulent flow in pipes, α ranges from 1.01 to 1.15, but it is usually between 1.03 and 1.06.

In some instances it is very desirable to use the proper value of α, but in most cases the error made in neglecting its divergence from 1.0 is negligible. As precise values of α are seldom known, it is customary in the case of turbulent flow to assume that $\alpha = 1$, i.e., that the kinetic energy is $V^2/2g$ per unit weight of fluid, measured in units of ft·lb/lb = ft or N·m/N = m. In laminar flow the velocity is usually so small that the kinetic energy per unit weight of fluid is negligible.

Potential Energy

The potential energy of a particle of fluid depends on its elevation above an arbitrary datum plane. We are usually interested only in *differences* of elevation, and therefore the location of the datum plane used is determined solely by

convenience. A fluid particle of weight W situated a distance z above datum possesses a potential energy of Wz. Thus its potential energy per unit *weight* is z, again measured in units of ft·lb/lb = ft or N·m/N = m.

The particle's potential energy per unit mass is gz, again measured in units of ft^2/sec^2 or m^2/s^2; its potential energy per unit volume is ρgz, again measured in units of lb/ft^2 or N/m^2.

Pressure Head

A particle of fluid has energy due to its pressure above datum, most usually its pressure above atmospheric, although we normally do not refer to this as pressure energy. From Eq. (3.4) this pressure is $p = \gamma h$, and so the depth of liquid that would produce this pressure, or the "pressure head" (Sec. 5.8), is $h = p/\gamma$. We see that the units of p/γ are ft = ft·lb/lb or m = N·m/N, or once again energy per unit *weight*.

Internal Energy

Internal energy is stored energy that is associated with the molecular, or internal state of matter; it may be stored in many forms, including thermal, nuclear, chemical, and electrostatic. Here we shall only consider internal *thermal* energy (heat), which is due to the motion of molecules and forces of attraction between them. Texts on thermodynamics describe this more fully. Experiments indicate that the internal thermal energy is primarily a function of temperature. For liquids and solids, the only exception occurs as they approach the vapor phase, when the internal thermal energy also depends on specific volume, or pressure. When a gas behaves as a perfect gas (Sec. 2.7), this also implies that the internal thermal energy is a function of temperature only. We can express internal thermal energy in terms of energy i per unit of mass[1] or in terms of energy I per unit of weight. Note therefore that $i = gI$.

We can take the zero of internal energy at any arbitrary temperature, since we are usually concerned only with differences. For a unit mass of substance at a constant volume, $\Delta i = c_v \Delta T$, where c_v is the specific heat at constant volume, whose units are ft·lb/(slug·°R) in the BG system or N·m/(kg·K) in SI units. Thus we express Δi in ft·lb/slug (N·m/kg in SI units). We usually express internal energy I per unit of weight in ft·lb/lb = ft (N·m/N = m in SI units).[2]

[1] The technical literature commonly represents internal energy per unit mass by the symbol u. In this text, however, we use i for internal energy per unit mass since we use u in several situations for velocity.

[2] In the BG system of measurement, scientists sometimes express internal energy I in Btu/lb; however, we rarely use those units today. Nevertheless, it is important that we are familiar with such units when reading technical papers that were written a number of years ago. 1 Btu ≈ 778 ft·lb.

SAMPLE PROBLEM 5.1 In laminar flow through a circular pipe the velocity profile is a parabola (Fig. S5.1), the equation of which is $u = u_m[1 - (r/r_0)^2]$, where u is the velocity at any radius r, u_m is the maximum velocity in the center of the pipe where $r = 0$, and r_0 is the radius to the wall of the pipe. Find α.

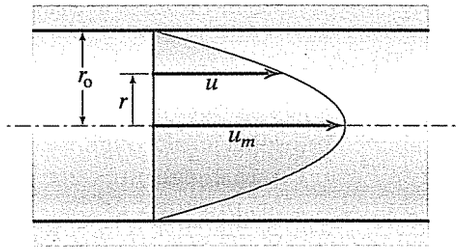


Figure S5.1

Solution

$$u = u_m\left[1 - \left(\frac{r}{r_0}\right)^2\right], \quad dA = 2\pi r\,dr$$

For Eq. (5.4):
$$\int u^3 dA = 2\pi u_m^3 \int_0^{r_0}\left[1 - \left(\frac{r}{r_0}\right)^2\right]^3 r\,dr$$

$$= 2\pi u_m^3 \int_0^{r_0}\left(r - 3\frac{r^3}{r_0^2} + 3\frac{r^5}{r_0^4} - \frac{r^7}{r_0^6}\right)dr$$

$$= 2\pi u_m^3\left[\frac{r^2}{2} - \frac{3}{4}\frac{r^4}{r_0^2} + \frac{3}{6}\frac{r^6}{r_0^4} - \frac{1}{8}\frac{r^8}{r_0^6}\right]_0^{r_0}$$

$$= 0.25\pi r_0^2 u_m^3$$

and
$$Q = AV = \int u\,dA = 2\pi u_m \int_0^{r_0}\left[1 - \left(\frac{r}{r_0}\right)^2\right] r\,dr$$

$$= 2\pi u_m \int_0^{r_0}\left(r - \frac{r^3}{r_0^2}\right)dr = 2\pi u_m\left[\frac{r^2}{2} - \frac{r^4}{4r_0^2}\right]_0^{r_0} = 0.5\pi u_m r_0^2$$

So
$$V = \frac{Q}{A} = \frac{0.5\pi u_m r_0^2}{\pi r_0^2} = 0.5u_m$$

Eq. (5.4):
$$\alpha = \frac{1}{AV^3}\int u^3 dA = \frac{0.25\pi r_0^2 u_m^3}{(\pi r_0^2)(0.5u_m)^3} = 2 \qquad \textbf{\textit{ANS}}$$

EXERCISES

5.1.1 Assume the velocity profile for turbulent flow in a circular pipe to be approximated by a parabola from the axis to a point very close to the wall where the local velocity is $u = 0.6u_m$, where u_m is the maximum velocity at the axis (Fig. X5.1.1). The equation for this parabola is $u = u_m[1 - 0.4(r/r_0)^2]$. Find α.

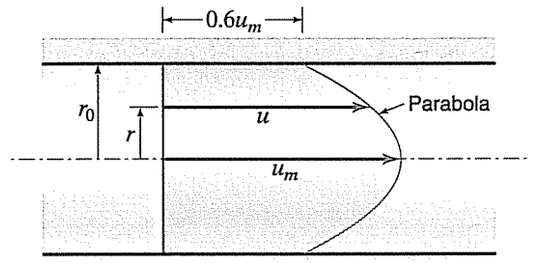


Figure X5.1.1

5.1.2 Assume flow in an open rectangular channel with the velocity at the surface twice that at the bottom, and with the velocity varying as a straight line from top to bottom. Find α.

5.1.3 Find α for the case of a two-dimensional laminar flow, as between two flat plates, for which the velocity profile is parabolic.

5.2 EQUATION FOR STEADY MOTION OF AN IDEAL FLUID ALONG A STREAMLINE, AND BERNOULLI'S THEOREM

Referring to Fig. 5.1, let us consider frictionless steady flow of an ideal fluid (Sec. 2.10) along the streamline. We shall consider the forces acting in the direction of the streamline on a small element of the fluid in the stream tube, and we shall apply Newton's second law, that is $F = ma$. The cross-sectional area of the element at right angles to the streamline may have any shape and varies from A to $A + dA$. We recall from Secs. 4.12 and 4.13 that in steady flow the velocity does not vary at a point (local acceleration = 0), but that it may vary with position (convective acceleration $\neq$ 0).

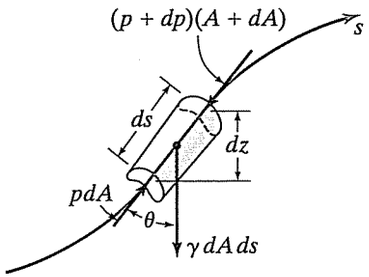


Figure 5.1
Element moving along streamline (ideal fluid).

The mass of the fluid element is $m = \rho ds(A + \frac{1}{2}dA) = \rho dsA$ when we neglect second-order terms. The forces tending to accelerate or decelerate this mass along s are (a) the pressure forces

$$pA + \left(p + \frac{1}{2}dp\right)dA - (p + dp)(A + dA) = -dpA$$

and (b) the weight component in the direction of motion, which is

$$-\gamma ds\left(A + \frac{1}{2}dA\right)\cos\theta = -\rho g\, dsA\frac{dz}{ds} = -\rho gA\, dz$$

Applying $\Sigma F = ma$ along the streamline, we get

$$-dpA - \rho gA\, dz = (\rho dsA)a$$

Dividing by the volume dsA,

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

This states that the pressure gradient along the streamline combined with the weight component in that direction causes the acceleration a of the element. Recalling from Eq. (4.24) that $a = V(dV/ds)$ for steady flow, we get

$$-\frac{dp}{ds} - \rho g\frac{dz}{ds} = \rho V\frac{dV}{ds}$$

Multiplying by ds/ρ and rearranging,

$$\frac{dp}{\rho} + g\,dz + V\,dV = 0 \tag{5.5}$$

We commonly refer to this equation as the ***one-dimensional Euler*[3] *equation,*** because Leonhard Euler (1707–1783), a Swiss mathematician, first derived it in about 1750. It applies to both compressible and incompressible flow, since the variation of ρ over the elemental length ds is small. Dividing through by g, we can also express Eq. (5.5) as

$$\frac{dp}{\gamma} + dz + d\frac{V^2}{2g} = 0 \tag{5.6}$$

Compressible Fluid

For the case of a compressible fluid, since $\gamma \neq$ constant, we must introduce an equation relating γ (or ρ) to p and T before integrating Eq. (5.5) or (5.6). We discussed stationary compressible fluid in Secs. 2.7–2.9, and flowing compressible fluid is treated further in Sec. 5.7 and Chap. 13.

[3] Euler is pronounced (oi'lər), to rhyme with boiler.

Incompressible Fluid

For the case of an incompressible fluid (γ = constant), we can integrate Eq. (5.6) to give

$$\text{Energy per unit weight:} \quad \frac{p}{\gamma} + z + \frac{V^2}{2g} = \text{constant (along a streamline)} \tag{5.7}$$

We know this famous equation as ***Bernoulli's theorem,*** in honor of Daniel Bernoulli (1700–1782), the Swiss physicist who presented this theorem in 1738. If we multiply each term first by g and then by ρ, we obtain the following alternate forms:

$$\text{Energy per unit mass:} \quad \frac{p}{\rho} + gz + \frac{V^2}{2} = \text{constant (along a streamline)} \tag{5.8}$$

and

$$\text{Energy per unit volume:} \quad p + \gamma z + \frac{1}{2}\rho V^2 = \text{constant (along a streamline)} \tag{5.9}$$

Terms in these three equations represent various energies of the flow, as discussed in Secs. 5.1 and 5.8. As noted, in Eq. (5.7) they are in units of energy per unit weight, in Eq. (5.8) they are in units of energy per unit mass, and in Eq. (5.9) they are in units of energy per unit volume. The constant (of integration) is known as the ***Bernoulli constant.***

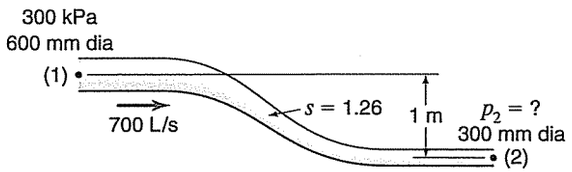
Because there are so many basic assumptions involved in the derivation of Bernoulli's equation, it is important to remember them all when applying it. They are:

1. It assumes viscous (friction) effects are negligible;
2. It assumes the flow is steady;
3. The equation applies along a streamline;
4. It assumes the fluid to be incompressible; and
5. It assumes no energy is added to or removed from the fluid along the streamline.

If we do not comply with any of these restrictions, serious errors can result. However, we do sometimes apply the Bernoulli equation to *real* fluids with good results in situations where the frictional effects are very small. The streamline, shown as two dimensional in Fig. 5.1, may also be three dimensional. If enough is known about the flow at some point on the streamline, we can find the Bernoulli constant; for Eq. (5.7) this constant is known as the total head, which we will discuss further in Sec. 5.8. Note that certain special flows do occur (Sec. 14.2) for which Bernoulli's equation holds *throughout* the flow field, not just along a streamline.

If the fluid is not moving, then we see that Eq. (5.7) reduces to Eq. (3.6).

SAMPLE PROBLEM 5.2 Glycerin (specific gravity 1.26) in a processing plant flows in a pipe at a rate of 700 L/s. At a point where the pipe diameter is 600 mm, the pressure is 300 kPa. Find the pressure at a second point where the pipe diameter is 300 mm if the second point is 1.0 m lower than the first point. Neglect head loss.



Solution

Facing inside cover: $\gamma_{water} = 9810\ \text{N/m}^3 = 9.81\ \text{kN/m}^3$

Eq. (4.6): $V_1 = \dfrac{0.70\ \text{m}^3/\text{s}}{\pi(0.3)^2\ \text{m}^2} = 2.48\ \text{m/s}, \qquad V_2 = 4V_1 = 9.90\ \text{m/s}$

Eq. (5.15): $$\frac{300}{1.26(9.81)} + 0 + \frac{(2.48)^2}{2(9.81)} = \frac{p_2}{1.26(9.81)} - 1.0 + \frac{(9.90)^2}{2(9.81)}$$

from which $p_2 = 254\ \text{kN/m}^2 = 254\ \text{kPa}$ ***ANS***

EXERCISES

5.2.1 Assume frictionless flow in a long, horizontal, conical pipe, which has a diameter of 3.6 ft at entrance and 2.4 ft at exit. The pressure head at the smaller end is 15 ft of water. If water flows through this cone at the rate of 95 cfs, find the velocities at the two ends and the pressure head at the larger end.

5.2.2 Assume the flow to be frictionless in the siphon shown in Fig. X5.2.2, where $a = 3$ ft, $b = 12$ ft. Find the rate of discharge in cfs and the pressure head at B if the pipe has a uniform diameter of 3 in.

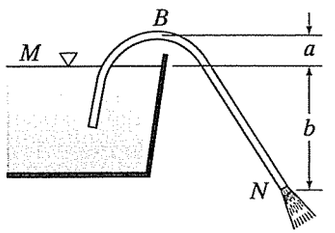


Figure X5.2.2

5.2.3 Refer to Fig. X5.2.2. Assume $a = 1$ m, $b = 4$ m, and the flow to be frictionless in the siphon. Find the rate of discharge in m^3/s and the pressure head at B if the pipe has a uniform diameter of 150 mm.

5.2.4 From point 1, a 25-mm-diameter pipe runs horizontally under the floor and then a 12.5-mm-diameter line runs 1 m up the wall to point 2. To maintain a pressure of 300 kPa at point 2, when 15°C water is flowing at 0.5 L/s, what pressure must be provided at point 1? Neglect friction.

5.2.5 A straight horizontal pipe changes diameter from 6 in at inlet to 3 in at outlet. If the water pressures are 7.5 psi at inlet and 5.0 psi at outlet, find the flow rate of water at 70°F. Neglect friction.

5.3 EQUATION FOR STEADY MOTION OF A REAL FLUID ALONG A STREAMLINE

Let us follow the same procedure as in the previous section, except that now we shall consider a real fluid. The real fluid element in a stream tube depicted in Fig. 5.2 is similar to that of Fig. 5.1, except that now with the real fluid there is an additional force acting because of fluid friction, namely $\tau(P + \frac{1}{2}dP)ds$, where τ (tau) is the shear stress at the boundary of the element and $(P + \frac{1}{2}dP)ds$ is the area over which the shear stress acts, P being the perimeter of the end area A, which may have any shape. Writing $\sum F = ma$ along the streamline and neglecting second-order terms, for steady flow we now get

$$-dpA - \rho gA\,dz - \tau P\,ds = (\rho\,ds\,A)V\frac{dV}{ds}$$

Dividing through by ρA and rearranging gives

$$\frac{dp}{\rho} + g\,dz + V\,dV = -\frac{\tau P}{\rho A}ds \tag{5.10}$$

This equation is similar to Eq. (5.5), except that it has an extra term. The extra term $-\tau P ds/(\rho A)$ accounts for fluid friction.

As before, we can also express Eq. (5.10) as

$$\frac{dp}{\gamma} + dz + d\frac{V^2}{2g} = -\frac{\tau P}{\gamma A}ds \tag{5.11}$$

These equations apply to steady flow of both compressible and incompressible real fluids.

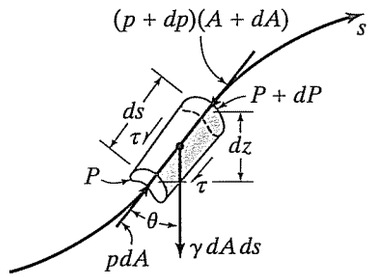


Figure 5.2
Element moving along streamline (real fluid).

Compressible Fluid

Once again, when we are dealing with a compressible fluid we must introduce an equation of state relating γ or ρ to p and T before integrating. Energy equations for the flow of compressible real fluid are further developed in Sec. 5.7 and Chap. 13.

Incompressible Fluid

For an incompressible fluid (γ = constant), we can integrate Eq. (5.11) directly. Integrating from some point 1 to another point 2 on the same streamline, where the distance between them is L, we get for an incompressible real fluid

$$\frac{p_2}{\gamma} - \frac{p_1}{\gamma} + z_2 - z_1 + \frac{V_2^2}{2g} - \frac{V_1^2}{2g} = -\frac{\tau PL}{\gamma A}$$

or

$$\text{Energy per unit weight:} \quad \left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) - \frac{\tau PL}{\gamma A} = \left(\frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}\right) \tag{5.12}$$

As we did in Sec. 5.2, we may easily convert Eq. (5.12) to represent energy per unit mass, or energy per unit volume. The basic assumptions involved in the derivation of this equation, that we need to bear in mind, are (1) steady flow, (2) of incompressible fluid, (3) along a streamline, (4) with no energy added or removed.

If we compare Eq. (5.12) with Bernoulli Eq. (5.7) for ideal flow we see again the only difference is the additional term $-\tau PL/(\gamma A)$, which represents the loss of energy per unit weight due to fluid friction between points 1 and 2. The dimensions of this energy loss term are length only, which agrees with all the other terms in Eq. (5.12), and so this term is a form of head (Sec. 5.8).

As we noted at the outset, the friction causing this loss of energy occurs over the boundary or surface of the element, of area PL. When, as occurs often, we consider the stream tube to fill the conduit, pipe, or duct conveying the fluid, PL becomes the inside surface area of the conduit wall, and τ becomes the shear stress at the wall, τ_0. Then we can call this energy loss term the

wall friction head loss: $$h_f = \frac{\tau_0 PL}{\gamma A} \tag{5.13}$$

Inserting this into Eq. (5.12), we obtain

$$\text{Energy per unit weight:} \quad \left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) - h_f = \left(\frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}\right) \tag{5.14}$$

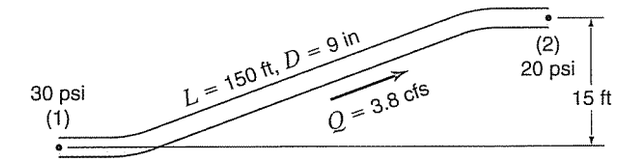
If, as is most common, the conduit is a circular pipe of diameter D, then $P/A = \pi D/(\pi D^2/4) = 4/D$, and Eq. (5.13) becomes the

pipe friction head loss: $$h_f = \frac{4\tau_0 L}{\gamma D} \tag{5.15}$$

Later we shall see how fluid friction can dissipate energy in many other ways besides through shear stress over the stream tube surface; examples are in the extra turbulence caused when discharging into still water (Sec. 5.12) and by flow through valves and orifices and the like (Chaps. 8 and 11). Fluid friction loss from *any* such cause, *including* wall or pipe friction, we commonly refer to as ***head loss,*** denoted by h_L. So wall friction head loss is usually a part of, but it may be all of, the total head loss. In a given conduit, then, $h_L \geq h_f$. We shall discuss head loss further in Sec. 5.6.

SAMPLE PROBLEM 5.3 Water flows through a 150-ft-long, 9-in-diameter pipe at 3.8 cfs. At the entry point, the pressure is 30 psi; at the exit point, 15 ft higher than the entry point, the pressure is 20 psi. Between these two points, find (*a*) the pipe friction head loss, (*b*) the wall shear stress, and (*c*) the friction force on the pipe.

Solution



(*a*) From Eq. (5.14): $$h_f = \left(\frac{30(144)}{62.4} + 0 + \frac{V^2}{2g}\right) - \left(\frac{20(144)}{62.4} + 15 + \frac{V^2}{2g}\right)$$

$V_1 = V_2$, so terms in V cancel, and

$$h_f = 8.08 \text{ ft} \qquad \textbf{\textit{ANS}}$$

(*b*) From Eq. (5.15): $$\tau_0 = \frac{h_f \gamma D}{4L} = \frac{8.08(62.4)0.75}{4(150)} = 0.630 \text{ lb/ft}^2 \qquad \textbf{\textit{ANS}}$$

(*c*) Friction force $= \tau_0 PL = \tau_0(\pi D)L = 0.630\pi(0.75)150 = 223$ lb **ANS**

EXERCISES

5.3.1 A vertical pipe of 4 ft diameter and 60 ft long has a pressure head of 22.7 ft of water at its upper end. When the flow of water through it is such that the mean velocity is 16 fps, the pipe friction head loss is $h_f = 2.8$ ft. Find the pressure head at the lower end of the pipe when the flow is (*a*) downward; (*b*) upward.

5.3.2 A vertical pipe of 1.5 m diameter and 20 m long has a pressure head of 6.3 m of water at its upper end. When the flow of water through it is such that the mean velocity is 5.6 m/s, the pipe friction head loss is $h_f = 1.09$ m. Find the pressure head at the lower end of the pipe when the flow is (*a*) downward; (*b*) upward.

5.3.3 A conical pipe has diameters at the two ends of 1.2 and 4.2 ft and is 48 ft long. It is vertical, and the pipe friction head loss is $h_f = 7.6$ ft for flow of water in either direction when the velocity at the smaller section is 28 fps. If the smaller section is at the top and the pressure head there is 6.4 ft of water, find the pressure head at the lower end when the flow is (*a*) downward; (*b*) upward.

5.3.4 In Fig. X5.3.4 the pipe AB is of uniform diameter and $h = 28$ ft. The pressure at A is 30 psi and at B is 40 psi. In which direction is the flow, and what is the pipe friction head loss in feet of the fluid if the liquid has a specific weight of (*a*) 35 lb/ft^3, (*b*) 92 lb/ft^3?

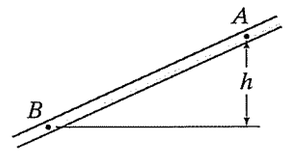


Figure X5.3.4

5.3.5 If $h = 10.5$ m in Fig. X5.3.4 and the pressures at A and B are 170 and 275 kPa respectively, find the direction of flow and the pipe friction head loss in meters of liquid. Assume the liquid has a specific gravity of 0.85.

5.3.6 Water flows through a pipe at 14 cfs. At a point where the pipe diameter is 18 in, the pressure is 30 psi; at a second point, further along the flow path and 2 ft lower than the first, the diameter is 9 in and the pressure is 18 psi. Find the pipe friction head loss between the two points. Neglect other head losses.

5.3.7 Water at 20°C flows up a straight 180-mm-diameter pipe that slopes at 12° to the horizontal. Find the shear stress at the wall, if the pressure is 100 kPa at point 1, and 25 kPa at higher point 2 that is 30 m further along the pipe.

5.4 PRESSURE IN FLUID FLOW

Pressure in Conduits of Uniform Cross Section

Let us now consider how pressure varies over a cross section of flow in a uniform conduit. Figure 5.3 shows a small prism of a flowing fluid. Perpendicular to the motion and in the plane of the sketch, the forces acting on the faces of the prism are p_1A and p_2A as shown. Forces parallel to the direction of motion, namely the pressure and friction forces and the weight component, must balance out if the flow is steady and parallel. Summing the perpendicular forces, we get $p_1A +$

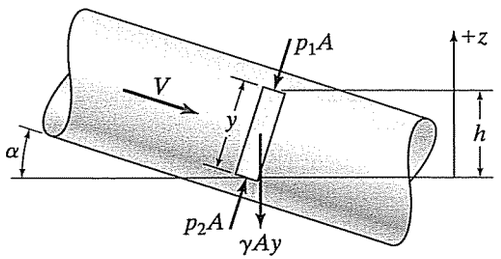


Figure 5.3

$\gamma Ay\cos\alpha - p_2A = 0$, where y is the dimension of the prism as shown, and A is its cross-sectional area. From this, we get

$$p_2 - p_1 = \gamma y\cos\alpha = \gamma h = \gamma(z_1 - z_2) = -\gamma(\Delta z) \tag{5.16}$$

which is similar to Eq. (3.3). Therefore *in any plane perpendicular to the direction of a parallel and steady flow the pressure varies according to the hydrostatic law.* The average pressure is then the pressure at the centroid of such an area. The pressure is lowest near the top of the conduit, and cavitation (Sec. 5.10), if it were to occur, would appear there first. Equation (5.16) tells us that on a horizontal axis through the conduit and perpendicular to its centerline the pressure is everywhere the same. Since the velocity is higher near the center than near the walls, it follows that the local energy head is also higher near the center. This emphasizes the fact noted earlier that a flow equation such as Eq. (5.7) or (5.12) applies along the same streamline, but not between two different streamlines, just as they do not apply between two streams in two separate channels.

Static Pressure

In a flowing fluid, we call the fluid pressure p the ***static pressure*** because it is the pressure that an instrument would measure if it were *static with respect to the fluid,* i.e., moving *with* the fluid. We measure it with piezometer tubes (Sec. 3.5) and other devices that attempt to minimize disturbance to the flow (see Sec. 11.2).

Stagnation Pressure

The center streamline in Fig. 4.12 shows that the velocity becomes zero at the ***stagnation point.*** If p/γ denotes the static-pressure head at some distance away where the velocity is V, while p_0/γ denotes the pressure head at the stagnation point, then, applying Eq. (5.7) to these two points, $p/\gamma + 0 + V^2/2g = p_0/\gamma + 0 + 0$, or the ***stagnation pressure*** is

$$p_0 = p + \gamma\frac{V^2}{2g} = p + \rho\frac{V^2}{2} \tag{5.17}$$

Some scientists call the quantity $\gamma V^2/2g$, or $\rho V^2/2$, the ***dynamic pressure.***

Equation (5.17) applies to a fluid where we may disregard compressibility. From Secs. 13.3 and 13.5 we can show that for a compressible fluid,

$$p_0 = p + \rho\frac{V^2}{2}\left(1 + \frac{V^2}{4c^2} + \cdots\right) \tag{5.18}$$

where c is the sonic (acoustic) velocity. For air at 68°F (20°C), $c \approx 1130$ fps (345 m/s). If $V = 226$ fps (69 m/s) the error in using 1.00 for the compressibility factor, which is the quantity in parentheses, is only one percent. But for higher values of V, the effect becomes much more important. Equation (5.18) is, however, restricted to values of V/c less than one.

EXERCISES

5.4.1 Find the stagnation pressure on the nose of a submarine moving at 12 knots in seawater ($\gamma = 64$ lb/ft^3) when it is 70 ft below the surface.

5.4.2 Find the stagnation pressure on the nose of a submarine moving at 6 m/s in seawater ($\gamma = 10\,050$ N/m^3) when it is 20 m below the surface.

5.4.3 Find the stagnation pressure on the nose of a fish swimming at 22 fps in fresh water ($\gamma = 62.4$ lb/ft^3) when it is 8 ft below the surface.

5.5 GENERAL ENERGY EQUATION FOR STEADY FLOW OF ANY FLUID

Let us now use the principles of Sec. 4.6 to consider the energy of the fluid system and control volume defined within the stream tube of Fig. 5.4. The fixed control volume lies between sections 1 and 2, and the (colored) moving fluid system consists of the fluid mass that was contained in the control volume at time t. During a short time interval Δt we shall assume that the fluid moves a short distance Δs_1 at section 1 and Δs_2 at section 2. As we are restricting ourselves in these discussions to steady flow, from Eq. (4.16b) $\gamma_1 A_1 \Delta s_1 = \gamma_2 A_2 \Delta s_2 = g\,\Delta m$, the weight of fluid entering and leaving the control volume during Δt. Recalling the analysis of Sec. 4.6, and letting the general property X now be the energy E, Eq. (4.8) becomes

$$\Delta E_S = \Delta E_{CV} + \Delta E_{CV}^{\text{out}} - \Delta E_{CV}^{\text{in}}$$

where, as before, subscript S denotes the moving fluid system and subscript CV denotes the fixed control volume. Because the flow is steady, conditions within the control volume do not change so $\Delta E_{CV} = 0$ and

$$\Delta E_S = \Delta E_{CV}^{\text{out}} - \Delta E_{CV}^{\text{in}} \tag{5.19}$$

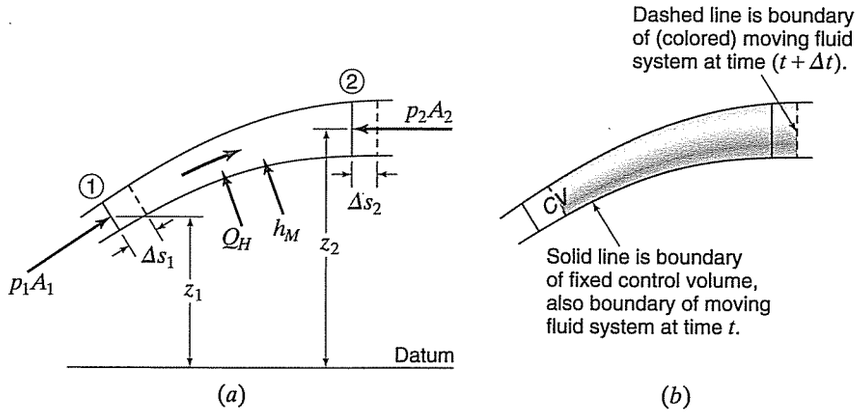


Figure 5.4

Let us now apply the ***first law of thermodynamics*** to the fluid system. This law states that for steady flow, the external work done on any system plus the thermal energy transferred into or out of the system is equal to the change of energy of the system. In other words, for steady flow during time Δt,

$$\text{External work done} + \text{heat transferred} = \Delta E_S$$

Note that work, heat, and energy all have the same units, and thus are interchangeable under certain conditions.

External work can be done on the moving fluid system in various ways. One way is when a force moves through a distance. So here, when the pressure forces acting on the boundaries move, in our case when p_1A_1 and p_2A_2 at the end sections move through Δs_1 and Δs_2, respectively, external work is done. This work is referred to as ***flow work.*** It can be expressed as

$$\begin{aligned}\text{Flow work} &= p_1A_1\Delta s_1 - p_2A_2\Delta s_2 \\ &= \frac{p_1}{\gamma_1}(\gamma_1A_1\Delta s_1) - \frac{p_2}{\gamma_2}(\gamma_2A_2\Delta s_2) \\ &= \left(\frac{p_1}{\gamma_1} - \frac{p_2}{\gamma_2}\right)g\Delta m\end{aligned}$$

The minus signs in the second terms indicate that the force and displacement are in opposite directions.

In addition to flow work, if there is a machine between sections 1 and 2 then there will be ***shaft work.*** During the short time interval Δt, we can write

$$\begin{aligned}\text{Shaft work} &= \frac{\text{weight}}{\text{time}} \times \frac{\text{energy}}{\text{weight}} \times \text{time} \\ &= \left(\gamma_1A_1\frac{ds_1}{dt}\right)h_M\Delta t = (\gamma_1A_1\Delta s_1)h_M \\ &= (\gamma_1\Delta\forall_1)h_M = (g\Delta m)h_M\end{aligned}$$

where h_M is the energy *added* to the flow by the machine per unit *weight* of flowing fluid. If the machine is a pump, which adds energy to the fluid, h_M is positive; if the machine is a turbine, which removes energy from the fluid, h_M is negative. Note that frictional shear stresses at the boundary of the fluid system also do work on the fluid within the system. These shear stresses are not external to the system and the work they do transforms into heat, which increases the temperature of the fluid within the system (usually very slightly).

The heat transferred from an external source into the fluid system over a time interval Δt is

$$\text{Heat transferred} = \left(\gamma_1A_1\frac{ds_1}{dt}\right)Q_H\Delta t = (\gamma_1A_1\Delta s_1)Q_H = (g\Delta m)Q_H$$

where Q_H is the amount of energy put *into* the flow by the external heat source per unit *weight* of flowing fluid. If the heat flow is out of the fluid, the value of Q_H is negative. Note that because the fluid is flowing through the control volume at

some rate (weight/sec), and Q_H is added to each unit weight of fluid, Q_H here does correspond to a *rate* of flow of heat.

So the total energy added to (or removed from) the fluid system during time Δt [the left side of Eq. (5.19)], is

$$\begin{aligned}\Delta E_S &= \text{external work done + heat transferred}\\ &= \text{flow work + shaft work + heat transferred}\\ &= \left(\frac{p_1}{\gamma_1} - \frac{p_2}{\gamma_2} + h_M + Q_H\right) g\,\Delta m \end{aligned} \tag{5.20}$$

To evaluate the right-hand side of Eq. (5.19), we first recall, as we noted initially, that for steady flow during time interval Δt, the weights of fluid entering the control volume at section 1 and leaving at section 2 are both equal to $g\,\Delta m$. From Sec. 5.1, we see that the energy (kinetic + potential + internal) carried across the boundary by $g\,\Delta m$ is

$$\Delta E = g\,\Delta m\left(z + \alpha\frac{V^2}{2g} + I\right)$$

Thus the change in energy of the control volume during Δt is

$$\Delta E_{CV}^{\text{out}} - \Delta E_{CV}^{\text{in}} = g\Delta m\left(z_2 + \alpha\frac{V_2^2}{2g} + I_2\right) - g\Delta m\left(z_1 + \alpha\frac{V_1^2}{2g} + I_1\right) \tag{5.21}$$

Substituting Eqs. (5.20) and (5.21) into Eq. (5.19), at the same time factoring out $g\,\Delta m$, we get

$$\frac{p_1}{\gamma_1} - \frac{p_2}{\gamma_2} + h_M + Q_H = \left(z_2 + \alpha_2\frac{V_2^2}{2g} + I_2\right) - \left(z_1 + \alpha_1\frac{V_{10}^2}{2g} + I_1\right)$$

or

$$\left(\frac{p_1}{\gamma_1} + z_1 + \alpha_1\frac{V_1^2}{2g} + I_1\right) + h_M + Q_H = \left(\frac{p_2}{\gamma_2} + z_2 + \alpha_2\frac{V_2^2}{2g} + I_2\right) \tag{5.22}$$

This energy equation applies to liquids, gases, and vapors, and to ideal fluids as well as to real fluids with friction, both compressible and incompressible. The only restriction is that it is for steady flow. The new features of this general equation are that it takes into account density changes (via γ), energy changes due to machines (h_M) and due to heat transfer to or from outside the fluid (Q_H), and it accounts for the conversion of other forms of fluid energy into internal heat (I).

The p/γ terms (pressure head, see Sec. 5.1) represent energy possessed by the fluid per unit weight of fluid by virtue of the pressure under which the fluid exists. Under proper circumstances, this pressure can be released and will transform into other forms of energy, i.e., kinetic, potential, or internal energy. Likewise, it is possible for these other forms of energy to transform into pressure head.

In turbulent flow there are other forms of kinetic energy besides that of translation described in Sec. 5.1. These other forms are the rotational kinetic energy of eddies initiated by fluid friction (Sec. 4.2) and the kinetic energy of the turbulent fluctuations of velocity (Sec. 4.5). No specific terms in Eq. (5.22) represent them because their effect appears indirectly. While the kinetic energy of translation can transform into increases in p/γ or z, the kinetic energy due to eddies and turbulent fluctuations can never transform into anything but thermal energy. Thus they appear as an increase in the numerical value of I_2 over the value it would have if there were no friction.

The general energy equation (5.22) and the continuity equation (4.16) are two important keys to the solution of many problems in fluid mechanics. For compressible fluids, we need a third equation, which is the equation of state, Eq. (2.4), which provides a relationship between density (or specific volume) and the absolute values of the pressure and temperature.

In many cases Eq. (5.22) simplifies greatly because certain quantities are equal and thus cancel each other, or are zero. Thus, if two points are at the same elevation, $z_1 - z_2 = 0$. If the conduit is well insulated or if the temperature of the fluid and that of its surroundings are practically the same, Q_H may effectively be zero. On the other hand, Q_H may be very large, as in the case of flow of water through a boiler tube. If there is no machine between sections 1 and 2 then the term h_M drops out. If there is a machine present, we can determine the rate of shaft work it does or is done on it by first solving Eq. (5.22) for h_M.

5.6 Energy Equations for Steady Flow of Incompressible Fluids, Bernoulli's Theorem

For liquids, and even for gases and vapors where the change in pressure is very small, we can consider the fluid as incompressible for all practical purposes, and thus we can take $\gamma_1 = \gamma_2 = \gamma =$ constant. In turbulent flow the value of α is only a little more than 1.0 (Sec. 5.1), and, as a simplifying assumption, we will assume it equal to 1.0. If the flow is laminar, $V^2/2g$ is usually very small compared with the other terms in Eq. (5.22); hence we introduce little error if we set α equal to 1.0 rather than 2.0, its true value for laminar flow in circular pipes. Thus, for an incompressible fluid, Eq. (5.22) with $\gamma =$ constant and $\alpha = 1.0$ becomes

$$\left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) + h_M + Q_H = \left(\frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}\right) + (I_2 - I_1) \tag{5.23}$$

Fluid friction produces eddies and turbulence (Sec. 4.2), and these forms of kinetic energy eventually transform into thermal energy. If there is no heat transfer, friction results in an increase in temperature, so that I_2 becomes greater than I_1. Or if the flow is isothermal (T and I both constant), there must be a loss of heat Q_H from the system at a rate equal to the rate at which friction is converting mechanical energy into thermal energy.

A change in the internal energy of a fluid coincides with a change in temperature. If c is the specific heat[4] of the incompressible fluid then, on a *mass* basis,

$$\frac{\Delta(\text{internal energy})}{\text{Unit of mass}} = \Delta i = i_2 - i_1 = c(T_2 - T_1) \tag{5.24}$$

On a unit *weight* basis, the change of internal energy is equal to the heat added to or removed from the fluid plus the heat generated by fluid friction, i.e.,

$$\frac{\Delta(\text{internal energy})}{\text{Unit of weight}} = \Delta I = \frac{\Delta i}{g} = I_2 - I_1 = \frac{c}{g}(T_2 - T_1) = Q_H + h_L \tag{5.25}$$

where h_L is the fluid-friction energy loss from all causes (Sec. 5.3) per unit weight of fluid (ft·lb/lb = ft or N·m/N = J/N = m); we commonly refer to h_L as ***head loss.***

The occurrence of head loss follows directly from the ***second law of thermodynamics*** (the law of degradation of energy). This states that some forms of energy, such as kinetic and potential energies, which will completely convert to other forms, are "superior" to other "inferior" forms, such as heat and internal energy, which will only partially convert to the superior forms. Thus, while it is possible for a given amount of mechanical energy to completely transform into heat, the opposite is only possible in part, resulting in the mechanical energy (head) loss that always occurs with viscous flow.

We see in Eq. (5.25) that if the loss of heat (Q_H negative) is greater than h_L then T_2 will be less than T_1. On the other hand, if there is any absorption of heat (Q_H positive), T_2 will be greater than the value which would have resulted from friction alone. Note that, because the specific heat c of water and other liquids is numerically very large (footnote 4 and Table A.4), Eq. (5.25) dictates that changes in heat energy ΔI, from heat exchange or head loss, cause relatively *small* temperature changes.

We can rewrite Eq. (5.25) as

$$h_L = (I_2 - I_1) - Q_H = \frac{c}{g}(T_2 - T_1) - Q_H \tag{5.26}$$

This states that the head loss is equal to the total internal heat gain minus any heat added from external sources, per unit weight of fluid; in other words, it is the gain in thermal energy *from internal sources only.*

[4] For water, c = 1 Btu/(mass of standard lb·°R) = 1 Btu(32.2 ft/s^2)/(lb·°R) = 32.2 Btu/(slug·°R). We define the Btu (British thermal unit) on the pages facing the covers of this book, and the slug in Sec. 1.5. In SI units, c for water = 1 cal/(g·K). We can also express these values as 25,000 ft·lb/(slug·°R) and 4187 N·m/(kg·K), equivalent to 25,000 ft^2/(s^2·°R) and 4187 m^2/(s^2·K), respectively. For the specific heats of various liquids, refer to Appendix A, Table A.4.

If we use Eq. (5.26) to substitute h_L for $(I_2 - I_1) - Q_H$, the general steady flow equation (5.23) for an incompressible fluid becomes

$$\left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) + h_M - h_L = \left(\frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}\right) \tag{5.27}$$

This states that the increase in the total mechanical energy of the fluid, between sections 1 and 2, is equal to that added by a machine minus that dissipated in head loss. If there is no machine between sections 1 and 2, the energy equation for an incompressible fluid becomes

$$\left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) - h_L = \left(\frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}\right) \tag{5.28}$$

When the head loss is caused only by wall or pipe friction, h_L becomes h_f (Sec. 5.3), and then we see that Eq. (5.28), derived from the energy of the fluid system and control volume, is the same as Eq. (5.14), derived from Newton's second law.

The head loss h_L may be very large in some cases, such as in very long pipelines or almost-closed valves. Although for any real fluid the head loss can never be zero, there are cases when it is so small that it may be neglected with small error.[5] In such special cases

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} \tag{5.29}$$

from which it follows that

$$\frac{p}{\gamma} + z + \frac{V^2}{2g} = \text{constant} \tag{5.30}$$

Readers will recognize the energy equation in either of these two last forms as ***Bernoulli's theorem,*** Eq. (5.7), previously discussed in Sec. 5.2. The constant will be discussed further in Sec. 5.8. We recall that Bernoulli's equation strictly holds for steady flow of an ideal (frictionless) incompressible fluid along a streamline, and along which no energy is added to or removed from the fluid. However, we can apply it to *real* incompressible fluids with good results if the friction effects are very small.

With this, we have now developed the Bernoulli equation from two standpoints: previously from Newton's second law, and now from energy considerations. In Sec. 5.2 we presented the Bernoulli equation in three alternate forms,

[5] It is important to recognize that we can assume frictionless flow when frictional effects are very small. For example, we can determine the pressure around the nose of a streamlined body (Fig. 4.12) quite accurately by assuming frictionless flow; however, we must consider frictional effects if we wish to determine the shear stresses at the boundary.

depending on whether the terms represent energy per unit weight as in Eqs. (5.29) and (5.30), energy per unit mass, or energy per unit volume.

SAMPLE PROBLEM 5.4 Water flows at 10 m^3/s in a 1.5-m-diameter aqueduct; the head loss in a 1000 m length of this pipe is 20 m. Find the increase in water temperature assuming no heat enters or leaves the pipe.

Solution

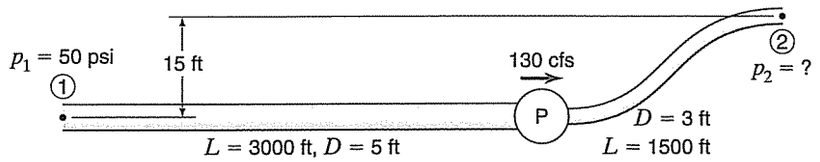
Using Eq. (5.26):
$$20\text{ m} = h_L = \frac{c}{g}(T_2 - T_1)$$

From footnote 4: c for water = 4187 N·m/(kg·K), and, rearranging,

$$\Delta T = T_2 - T_1 = \frac{gh_L}{c} = \frac{(9.81\text{ m/s}^2)(20\text{ m})}{4187[(\text{kg}\cdot\text{m/s}^2)\cdot\text{m}]/(\text{kg}\cdot\text{K})}$$

SAMPLE PROBLEM 5.5 Water is pumped through a pipeline to a treatment plant at a rate of 130 cfs. The 5-ft-diameter suction line is 3000 ft long, and the 3-ft-diameter discharge line is 1500 ft long. The pump adds energy to the water at a rate of 40 ft·lb/lb, and the total head loss is 56 ft. If the water pressure at the pipeline entrance is 50 psi, what is the pressure at the exit, which is 15 ft higher than the entrance?

Solution



Eq. (4.6):
$$V_1 = \frac{130}{(\pi/4)(5)^2} = 6.62\text{ fps}, \qquad V_2 = \frac{130}{(\pi/4)(3)^2} = 18.39\text{ fps}$$

Eq. (5.27):

$$\left(\frac{50(144)}{62.4} + 0 + \frac{(6.62)^2}{2(32.2)}\right) + 40 - 56 = \left(\frac{p_2(144)}{62.4} + 15.0 + \frac{(18.39)^2}{2(32.2)}\right)$$

from which
$$p_2 = 34.6\text{ psi} \qquad \textbf{\textit{ANS}}$$

EXERCISES

5.6.1 Water is flowing in a pipeline. Due to heat input from the environment and energy dissipation (head loss), the water temperature rises by 3°F between intake and outlet. Find the gain in heat in (*a*) ft·lb/lb, and (*b*) Btu/lb.

5.6.2 Head loss and sunshine striking a pipeline cause the temperature of the water flowing inside to rise by 2°C between two measuring points. Find the heat gain in (*a*) J/N, and (*b*) cal/N.

5.6.3 The pipeline shown in Fig. X5.6.3 supplies water to a hydroelectric power plant, the elevation of which is 2200 ft below the level of the water surface at intake to the pipe. If 8% of this total, or 176 ft, is the head loss in the line, what will be the value of ΔI in Btu/lb if there is no heat transfer; and what will be the rise in temperature?

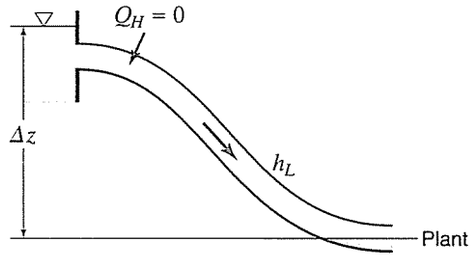


Figure X5.6.3

5.6.4 The pipeline shown in Fig. X5.6.3 supplies water to a hydroelectric power plant, the elevation of which is 650 m below the level of the water surface at intake to the pipe. If 8% of this total, or 52 m, is the head loss in the line, what will be the value of ΔI in J/N if there is no heat transfer, and what will be the rise in temperature?

5.7 ENERGY EQUATION FOR STEADY FLOW OF COMPRESSIBLE FLUIDS

If we choose sections 1 and 2 so that there is no machine between them, and if we assume α as 1.0, Eq. (5.22) becomes

$$\left(\frac{p_1}{\gamma_1} + I_1 + z_1 + \frac{V_1^2}{2g}\right) + Q_H = \left(\frac{p_2}{\gamma_2} + I_2 + z_2 + \frac{V_2^2}{2g}\right) \qquad (5.31)$$

For most compressible fluids, i.e., gases or vapors, the quantity p/γ is usually very large compared with $z_1 - z_2$ because of the small value of γ, and therefore we usually omit the z terms. But we should not ignore $z_1 - z_2$ unless we know it is negligible compared with the other quantities.

For gases and vapors, we usually combine the p/γ and the I terms into a single term called ***enthalpy.*** Thus enthalpy represents a composite energy property possessed by a given mass (or weight) of gas or vapor. In thermodynamics we

usually express enthalpy in terms of energy per unit mass (h) rather than energy per unit weight. Thus[6]

$$h = i + \frac{p}{\rho} = gI + \frac{p}{\rho} \tag{5.32}$$

and so

$$I + \frac{p}{\gamma} = \frac{h}{g} \tag{5.33}$$

With these changes, Eq. (5.31) becomes

$$\frac{h_1}{g} + \frac{V_1^2}{2g} + Q_H = \frac{h_2}{g} + \frac{V_2^2}{2g} \tag{5.34}$$

This equation is valid for any gas or vapor and for any process. We will need some knowledge of thermodynamics to evaluate the enthalpies, and in the case of vapors we will need to use vapor tables or charts, because we cannot express their properties by any simple equations. We discuss many more aspects of the flow of compressible fluids in Chap. 13.

SAMPLE PROBLEM 5.6 In an air conditioning system, air flows without heat gain or loss through a horizontal pipe of uniform diameter. At section 1 the pressure is 150 psia, the velocity is 80 fps, and the temperature is 70°F; at section 2 the pressure is 120 psia and the temperature is 50°F. Find (*a*) the change in kinetic energy of the air; (*b*) the head (mechanical energy) loss in Btu/lb; (*c*) the change in enthalpy; all between sections 1 and 2. Assume the air to be a perfect gas.

Solution

Eq. (2.4): $$\frac{p}{\rho T} = R = \text{constant}, \quad \text{so} \quad \frac{p_1}{\rho_1 T_1} = \frac{p_2}{\rho_2 T_2} \tag{1}$$

From Eq. (4.16*a*): $$\frac{\dot{m}}{A} = \rho_1 V_1 = \rho_2 V_2 \tag{2}$$

Multiplying (1) by (2) to eliminate ρ:

$$\frac{p_1 V_1}{T_1} = \frac{p_2 V_2}{T_2}$$

So $$V_2 = V_1\left(\frac{p_1 T_2}{p_2 T_1}\right) = 80\left(\frac{150}{120}\right)\frac{(460 + 50)}{(460 + 70)} = 96.2 \text{ fps}$$

[6] Values of enthalpy h for vapors commonly used in engineering, such as steam, ammonia, freon, and others, are given in vapor tables or charts. For a perfect gas and practically for real gases, $\Delta h = c_p \Delta T$, where c_p is specific heat at constant pressure. For air at usual pressures, c_p has a value of 6000 ft·lb/(slug·°R) [or 1003 N·m/(kg·K)]. These are equivalent to 6000 ft²/(sec²·°R) [or 1003 m²/(s²·K)].

(*a*) $$\Delta KE = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} = \frac{96.2^2 - 80^2}{2(32.2)} = 44.4 \text{ ft·lb/lb increase} \quad \textbf{\textit{ANS}}$$

(*b*) From Eq. (5.28) with $z_1 = z_2$:

$$h_L = \frac{p_1}{\rho_1 g} - \frac{p_2}{\rho_2 g} + \frac{V_1^2 - V_2^2}{2g}$$

Eq. (2.4): $$\frac{p}{\rho} = RT$$

Table A.5 for air: $R = 1715 \text{ ft}^2/(\text{s}^2\cdot^\circ\text{R})$

So $$h_L = \frac{R}{g}(T_1 - T_2) + \frac{V_1^2 - V_2^2}{2g}$$

$$= \frac{1715}{32.2}(460 + 70 - 460 - 50) - 44.4 = 1021 \text{ ft}$$

$$h_L = \frac{1021 \text{ ft·lb/lb}}{778 \text{ ft·lb/Btu}} = 1.312 \text{ Btu/lb} \quad \textbf{\textit{ANS}}$$

(*c*) From Eq. (5.34) with $Q_H = 0$:

$$\frac{h_1 - h_2}{g} = \frac{V_2^2 - V_1^2}{2g} = \Delta KE = 44.4 \text{ ft·lb/lb increase}$$

$$\Delta h = h_2 - h_1 = -g(\Delta KE) = -32.2(44.4) = 1493 \text{ ft·lb/slug decrease} \quad \textbf{\textit{ANS}}$$

EXERCISES

5.7.1 Gas flows at a constant temperature through a uniform, horizontal pipe. At section 1 the pressure is 125 psia and the velocity is 50 fps; at section 2 the pressure is 105 psia and the velocity is 65 fps. Between sections 1 and 2, find (*a*) the change in enthalpy; (*b*) the gain or loss of heat per lb. Assume the gas is perfect. [*Hint:* Recall Eq. (2.4).]

5.7.2 Air flows isothermally (constant temperature) through a horizontal duct of constant cross section. At station 1 the pressure is 860 kPa abs and the velocity is 22 m/s; at station 2 the pressure is 1040 kPa abs and the velocity is 18 m/s. Between stations 1 and 2, find (*a*) the change in enthalpy; (*b*) the gain or loss of heat per newton. Assume the air is a perfect gas. [*Hint:* Recall Eq. (2.4).]

5.7.3 Oxygen flows without gain or loss of heat through a horizontal pipe of constant cross section. At section 1 the pressure is 170 psia, the velocity is 75 fps, and the temperature is 50°F; at section 2 the pressure is 125 psia, the velocity is 98 fps, and the temperature is 30°F. Between sections 1 and 2, find (*a*) the head (mechanical energy) loss in Btu/lb; (*b*) the change in enthalpy. Assume the oxygen is a perfect gas. [*Hint:* Recall Eq. (2.4).]

5.7.4 Air flows without heat gain or loss through a uniform horizontal pipe. At station 1 the pressure is 1135 kPa abs, the velocity is 25 m/s, and the temperature is 10°C; at station 2 the pressure is 830 kPa abs, the velocity is 33 m/s, and the temperature is 0°C. Between sections 1 and 2, find (*a*) the head (mechanical energy) loss in J/N; (*b*) the change in enthalpy. Assume the air is a perfect gas. [*Hint:* Recall Eq. (2.4).]

5.8 HEAD

In Eq. (5.28) each term has the dimensions of *length*. Thus p/γ, called the ***pressure head,*** represents the energy per unit weight stored in the fluid by virtue of the pressure under which the fluid exists; z, called the ***elevation head*** or ***potential head,*** represents the potential energy per pound of fluid; and $V^2/2g$, called the ***velocity head,*** represents the kinetic energy per pound of fluid. We call the sum of these three terms the ***total head,*** usually denoted by H, so that

$$H = \frac{p}{\gamma} + z + \frac{V^2}{2g} \tag{5.35}$$

Although we usually express each term in this equation in feet (or meters), it actually represents *foot pounds of energy per pound of fluid flowing* (newton meters of energy per newton of fluid flowing in SI units). Note also that we call the sum of the middle two terms above, $(p/\gamma + z)$, the ***piezometric head*** or the ***static (pressure) head*** (see Sec. 5.11).

For an ideal (frictionless) incompressible fluid with no machine between 1 and 2, $H_1 = H_2$, but for a real fluid,

$$H_1 - h_L = H_2 \tag{5.36}$$

This is merely a brief way of writing Eq. (5.28), in which the total head loss h_L (Sec. 5.3) includes the pipe or wall friction head loss h_f and possibly other losses, to be discussed later. For a real fluid, it is obvious that if there is no input of energy head h_M by a machine between sections 1 and 2, the total head must decrease in the direction of flow.

If there *is* a machine between sections 1 and 2, then

$$H_1 + h_M - h_L = H_2 \tag{5.37}$$

If the machine is a pump, $h_M = h_p$, where h_p is the energy head put *into* the flow by the pump. If the machine is a turbine, $h_M = -h_t$, where h_t is the energy head *extracted* from the flow by the turbine.

5.9 POWER CONSIDERATIONS IN FLUID FLOW

We recall from mechanics that the power P developed when a force F acts on a translating body, or when a torque T acts on a rotating body, is given by

$$\text{Rate of energy transfer} = \text{Power} = P = FV = T\omega \tag{5.38}$$

where V is linear velocity in feet per second (or meters per second) and ω is angular velocity in radians per second. The force F represents the component force in the direction of the velocity V.

Substituting ΔpA for F and γh for Δp we can write

$$FV = (\Delta pA)V = (\gamma h)AV$$

and noting from Eq. (4.3) that $AV = Q$, we get

$$P = Q\Delta p = \gamma Q h \tag{5.39}$$

where γ = the unit weight of fluid, lb/ft^3 (N/m^3 in SI units)
Q = the rate of flow, ft^3/sec (m^3/s in SI units)
h = the energy head, ft (m in SI units)
p = the pressure, lb/ft^2 (Pa in SI units)

We will refer to these equations in Chap. 6, where we discuss the dynamic forces exerted by moving fluids, and again in Chaps. 15 and 16, in our discussion of turbomachinery.

In an alternative approach, we recall that every term of Eq. (5.28) represents energy per unit weight (i.e., energy head). If we multiply the energy head by the weight rate of flow, the resulting product represents rate of energy transfer, or power, since

$$\text{Power} = \frac{\text{energy}}{\text{time}} = \frac{\text{energy}}{\text{weight}} \times \frac{\text{weight}}{\text{time}} = h \times G = h \times g\dot{m}$$

from which, again, $$P = \gamma Q h$$

Noting, from the inside cover, that 1 hp = 550 ft·lb/sec (exactly) = 0.745 700 kW, we can obtain the convenient conversions

in BG units, $$\text{Horsepower} = P = \frac{\gamma Q h}{550} = \frac{Q\Delta p}{550} \tag{5.40}$$

while in SI units, $$\text{Kilowatts} = P = \frac{\gamma Q h}{1000} = \frac{Q\Delta p}{1000} \tag{5.41}$$

In these equations h may be any head (difference) and Δp any pressure difference for which we desire the corresponding power. For example, to find the power extracted from the flow by a turbine (i.e., the rate at which shaft work is done on a turbine, see Sec. 5.5), substitute h_t for h; to find the power of a jet, substitute $V_j^2/2g$ for h, where V_j is the jet velocity; and to find the power lost because of fluid friction, substitute h_L (or h_f) for h.

When power is transmitted through a process or machine, some power is lost in the process due to friction. The ***efficiency*** η (eta) of the transmission is the fraction of the power input that appears in the output, i.e.,

$$\text{Efficiency } \eta = \frac{\text{power output}}{\text{power input}} \tag{5.42}$$

We discuss the efficiency of pumps and turbines in more detail in Secs. 15.3 and 16.9, respectively.

SAMPLE PROBLEM 5.7 Find the rate of energy loss due to pipe friction for the pipe of Sample Prob. 5.4.

Solution

Eq. (5.41): $$\text{Rate of energy loss} = \frac{\gamma Q h}{1000}, \quad \text{where} \quad h = h_L$$

$$= \frac{(9810\ \text{N/m}^3)(10\ \text{m}^3\text{/s})(20\ \text{m})}{1000}$$

$$= 1962\ \text{kW} \qquad \textbf{\textit{ANS}}$$

SAMPLE PROBLEM 5.8 A liquid with a specific gravity of 1.26 is being pumped from A to B through the pipeline of Fig. S5.8. At A the pipe diameter is 24 in (600 mm) and the pressure is 45 psi (300 kPa). At B the pipe diameter is 12 in (300 mm) and the pressure is 50 psi (330 kPa). Point B is 3 ft (1.0 m) lower than A. Find the flow rate if the pump puts 22 hp (16 kW) into the flow. Neglect head loss.

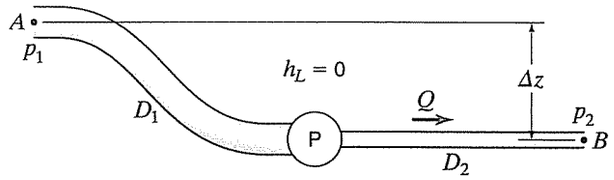


Figure S5.8

Solution (BG units)

Eq. (5.40): $$\text{Horsepower} = 22 = \frac{(1.26 \times 62.4) Q h_p}{550}$$

Rearranging: $$h_p = \frac{153.9}{Q}$$

Eq. (5.27) with elevation A as datum, and with $h_L = 0$ (given), using $V = Q/A$, gives

$$\frac{45(144)}{1.26(62.4)} + 0 + \frac{(Q/\pi)^2}{2(32.2)} + \frac{153.9}{Q} = \frac{50(144)}{1.26(62.4)} - 3 + \frac{[Q/(0.25\pi)]^2}{2(32.2)}$$

i.e., $$\frac{Q^2}{42.37} + 6.158 - \frac{153.9}{Q} = 0$$

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Without a polynomial solver (Appendixes B and C.1) or an equation solver (Appendixes C.1–C.3 and D), we can solve this cubic equation by trials (see Sample Prob. 3.5) as follows:

Trial Q:	10.0	20.0	17.0	14.0	14.15
Left side:	−22.26	7.903	3.925	−0.210	0.006

Thus $Q = 14.15$ cfs ***ANS***

Note: The other two roots of this equation in Q involve imaginary numbers. We could "automate" the trial calculations by using a spreadsheet (Appendix C.2).

Solution (SI Units)

Eq. (5.41): $$\text{kW} = 16 = \frac{(1.26 \times 9810) Q h_p}{1000}$$

Rearranging: $$h_p = \frac{1.294}{Q}$$

Eq. (5.27) with elevation A as datum, and with $h_L = 0$ (given), using $V = Q/A$, gives

$$\frac{300 \times 10^3}{1.26(9810)} + 0 + \left[\frac{Q}{\pi(0.3)^2}\right]^2 \frac{1}{2(9.81)} + \frac{1.294}{Q} = \frac{330 \times 10^3}{1.26(9810)} - 1.0$$
$$+ \left[\frac{Q}{\pi(0.15)^2}\right]^2 \frac{1}{2(9.81)}$$

By trials, $Q = 0.418\ \text{m}^3\text{/s}$ ***ANS***

Note: The other two roots of this cubic equation in Q involve imaginary numbers.

EXERCISES

5.9.1 A turbine, located at an elevation 750 ft below that of the water surface at intake (Fig. X5.9.1), carries a flow of 120 cfs. The head loss in the pipeline leading to it is 25 ft. Find the horsepower delivered by the turbine if its efficiency is 90%.

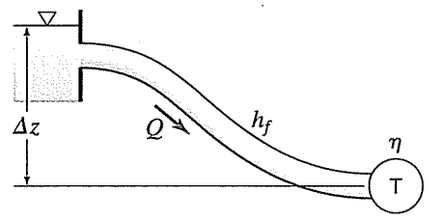


Figure X5.9.1

5.9.2 A turbine, located 255 m below the water surface at intake (Fig. X5.9.1), carries a flow of 3.5 m³/s. The head loss in the pipeline leading to it is 10 m. Find the power (kW) delivered by the turbine if its efficiency is 92%.

5.9.3 Water entering a pump through an 8-in-diameter pipe at 4 psi has a flow rate of 3.5 cfs. It leaves the pump through a 4-in-diameter pipe at 15 psi. Assuming that the suction and discharge sides of the pump are at the same elevation, find the horsepower delivered to the water by the pump.

5.9.4 After entering a pump through an 180-mm-diameter pipe at 35 kN/m^2, oil ($s = 0.82$) leaves the pump through a 120-mm-diameter pipe at 120 kN/m^2. The suction and discharge sides of the pump are at the same elevation. Find the rate at which energy is delivered to the oil by the pump if the flow rate is 70 L/s.

5.9.5 Water from a reservoir is being supplied to a powerhouse that is located at an elevation 935 ft below that of the reservoir surface. Discharging through a nozzle, the water has a jet velocity of 240 fps and a jet diameter of 6 in. Find the horsepower lost to friction between the reservoir and the jet, and find the horsepower of the jet.

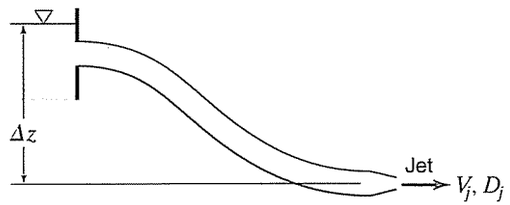


Figure X5.9.5

5.9.6 Water from a reservoir is being supplied to a powerhouse that is located at an elevation 325 m below that of the reservoir surface (Fig. X5.9.5). Discharging through a nozzle, the water has a jet velocity of 75 m/s and a jet diameter of 250 mm. Find the kW lost to friction between the reservoir and the jet, and find the power of the jet in kW.

5.10 CAVITATION

The rapid vaporization and recondensation of liquid as it briefly flows through a region of low absolute pressure we call ***cavitation,*** as we first noted in Sec. 2.13. This phenomenon is not possible in gas flow, because a gas does not change state at low pressure, whereas a liquid will change to a gas (vapor) if the pressure is low enough. We must investigate the possibility of cavitation occurring in liquid flows because it can cause serious damage.

The dangerous, temporary low-pressure conditions associated with cavitation result from temporary high velocities, in accordance with Bernoulli's theorem, Eq. (5.7). In view of that theorem, at a given location (elevation z = constant) in a liquid flow where no energy is added or removed, if the velocity head increases, there must be a corresponding decrease in the pressure head. However, so long as there is some liquid present to evaporate, there is a minimum absolute pressure possible, namely, the vapor pressure of the liquid. The (absolute) vapor pressure depends on the liquid and its temperature (see, e.g., Table A.1 for water), and it is usually less than atmospheric (Sec. 2.13). If conditions are such that calculations indicate the absolute pressure of a liquid is lower than its vapor pressure, this simply means that the assumptions upon

which the calculations are based no longer apply. Thus the critical condition for cavitation is

$$(p_{crit})_{abs} = p_v$$

But

$$(p_{crit})_{abs} = p_{atm} + (p_{crit})_{gage}$$

so that

$$(p_{crit})_{gage} = -(p_{atm} - p_v) \tag{5.43}$$

where p_{atm}, p_v, and p_{crit} represent the (absolute) atmospheric pressure, the (absolute) vapor pressure, and the critical (or minimum) possible pressure, respectively, in liquid flow. Equation (5.43) states that the gage pressure head in a flowing liquid can be negative, but no more negative than $p_{atm} - p_v$. Note that the same equations can of course be expressed in terms of pressure *head,* by dividing all pressures by γ.

If at any point in a liquid the local velocity is so high that the pressure falls to its vapor pressure, the liquid will then vaporize (or boil) at that point, and bubbles of vapor will form. As the fluid moves on into a region of higher pressure, the bubbles of vapor will suddenly condense; in other words, they ***collapse*** or ***implode.*** When this occurs adjacent to solid walls, the collapse begins as a jet of liquid entering the bubble from the side opposite the wall. Figure 5.5 is a microphotograph of such a jet. Investigators have estimated that jet velocities reach 360 ft/sec (110 m/s), and that they cause pressures of up to 500 atmospheres (7350 psi, or 50700 kPa) when the jet strikes the wall.[7] They also estimated that the implosion heats the liquid immediately surrounding the cavity to about 3800°F (2100°C) for less than a microsecond. Although the jets are very small, they occur continuously with a high frequency; so combined with the high temperatures and the shock waves caused by bubble collapse, they may damage the wall material.

Such action often severely and quickly damages turbine runners, pump impellers, and ship screw propellers, because it rapidly makes holes in the metal

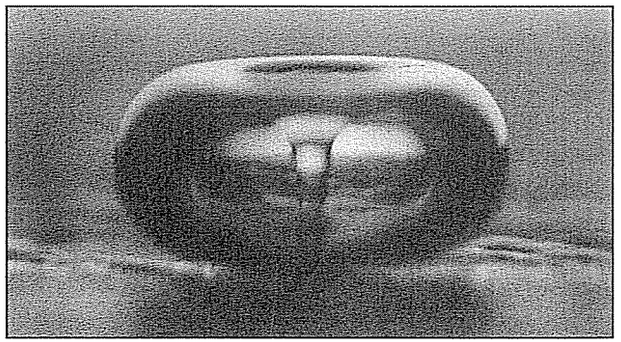


Figure 5.5
Photomicrograph of imploding bubble, with liquid jet moving downward through the center. (Bubble diameter is about 0.006 in, or 0.15 mm.)(Courtesy of Dr. Larry Crum)

[7] K.S. Suslick, The Chemical Effects of Ultrasound, *Scientific American,* Vol. 260, No. 2, pp. 80–86, 1989.

(see Fig. 16.15). Similar damage can occur immediately downstream of partly open valves. Overflow spillways (Sec. 11.13), stilling basins (Sec. 10.18), and other types of hydraulic structures built of concrete also may experience damage by cavitation. The damaging action is known as ***pitting.*** Not only is cavitation destructive, but it may cause a drop in efficiency of the machine or propeller or other device, and it may produce undesirable cavitation noise and vibration.

In order to avoid cavitation, we need to keep the absolute pressure at every point above the vapor pressure. There are various ways we can ensure this. In one way, we can raise the general pressure level, by placing the device below the intake level so that the liquid flows to it by gravity rather than being drawn up by suction. In another way, we can design the machine so that there are no local velocities high enough to produce such a low pressure. In a third way, we can admit atmospheric air into the low-pressure zone; we often do this downstream of partly open valves and on overflow spillways (see Sec. 11.13).

Figure 5.6 shows photographs of blades for an axial-flow pump set up in a transparent-lucite working section where the pressure level was varied. For *a*, *b*, and *c*, the water velocity was the same around the same vane but with decreasing absolute pressures. We see that the vapor pocket under the vane became larger at lower pressures. For *d*, the stream flow and the pressure were the same as for *b*, but the nose of the blade was slightly different in shape, which gave a different type of bubble formation. This shows the effect of a slight change in design.

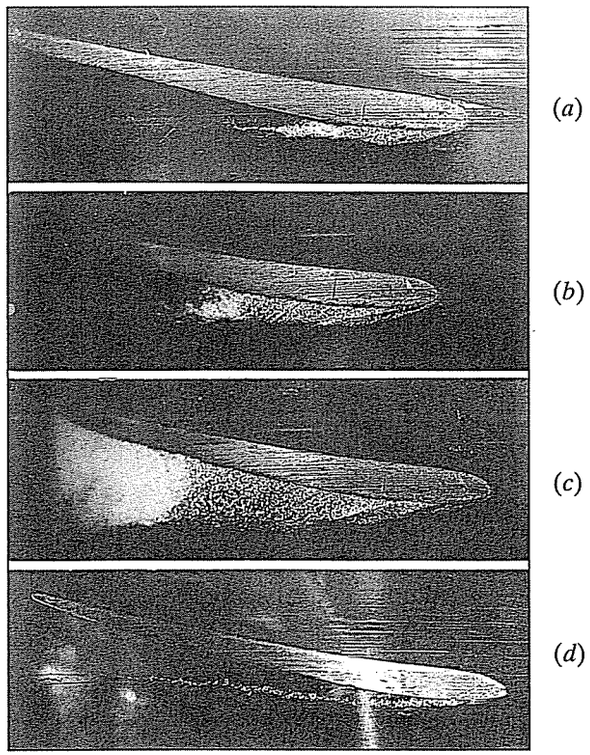


Figure 5.6
Cavitation phenomena: flow around a blade of an axial-flow pump, illustrating the effect of reducing absolute pressure in (*a*), (*b*), and (*c*), and the effect of a slight change of shape in (*d*). (Courtesy of the Archives, California Institute of Technology)

SAMPLE PROBLEM 5.9 A liquid ($s = 0.86$) with a vapor pressure of 3.8 psia flows through the horizontal constriction in Fig. S5.9. Atmospheric pressure is 26.8 inHg. Find the maximum theoretical flow rate, i.e., at what minimum Q does cavitation occur in the ***throat*** (narrowest section)? Neglect head loss.

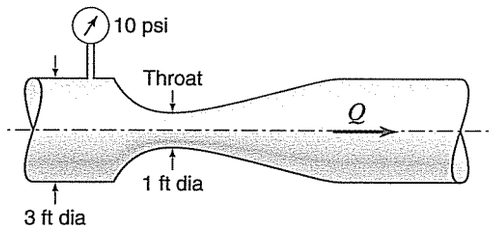


Figure S5.9

Solution

Since the standard atmosphere is equivalent to 29.92 inHg and 14.70 psia (Sec. 3.5),

$$p_{\text{atm}} = \frac{26.8}{29.92}(14.70) = 13.16 \text{ psia}$$

From Eq. (5.43): $$\left(\frac{p_{\text{crit}}}{\gamma}\right)_{\text{gage}} = -\left[\frac{13.16 - 3.8}{0.86(62.4)}\right]144 = -25.1 \text{ ft}$$

Eq. (4.7): $$V_1 = \frac{Q}{A_1} = \frac{4Q}{\pi D_1^2} = \frac{4Q}{\pi 3^2} = \frac{Q}{7.07}; \quad V_2 = \frac{4Q}{\pi 1^2} = \frac{Q}{0.785}$$

Eq. (5.29): $$\frac{10(144)}{0.86(62.4)} + 0 + \left(\frac{Q}{7.07}\right)^2 \frac{1}{2(32.2)} = -25.1 + 0 + \left(\frac{Q}{0.785}\right)^2 \frac{1}{2(32.2)}$$

$$Q = 45.7 \text{ cfs} \quad \textit{ANS}$$

EXERCISES

5.10.1 Water at 170°F flows horizontally through the constriction of Fig. S5.9 when the atmospheric pressure is 28.2 inHg. Neglecting head loss, find the flow rate at which cavitation begins.

5.10.2 Water at 40°C flows horizontally through a constriction similar to that in Fig. S5.9 when the atmospheric pressure is 715 mmHg. The gage reading is 35 kPa, $d_1 = 0.5$ m, and $d_2 = 0.15$ m. Neglecting head loss, find the flow rate at which cavitation begins.

5.10.3 Water at 80°F flows horizontally through the constriction of Fig. S5.9 at a rate of 65 cfs when the atmospheric pressure is 27.9 inHg. Neglecting head loss, find the largest throat constriction diameter d_2, that will cause cavitation.

5.10.4 Water at 50°C flows horizontally at a rate of 2 m^3/s through a constriction similar to that in Fig. S5.9 when the atmospheric pressure is 750 mmHg. The gage reading is 30 kPa and $d_1 = 0.5$ m. Neglecting head loss, find the largest throat constriction diameter d_2 that will cause cavitation.

5.10.5 Referring to Fig. S5.9, find the maximum theoretical flow rate for water at 70°C. Neglect head loss. The diameters are 800 mm and 400 mm respectively, the upstream pressure is 20 kPa, and the atmospheric pressure is 750 mmHg.

5.11 DEFINITION OF HYDRAULIC GRADE LINE AND ENERGY LINE

When dealing with flow problems involving liquids, the concepts of energy line and hydraulic grade line are usually advantageous. Even with gas flow, these concepts can be useful.

We refer to the quantity $p/\gamma + z$ as the ***piezometric head,*** because it represents the level to which liquid will rise in a ***piezometer tube,*** which has its connecting end in a plane parallel to the flow (Fig. 5.7 and Sec. 3.5). The ***piezometric line,*** or ***hydraulic grade line*** (HGL), is a line drawn through the liquid surfaces in the piezometers (Fig. 5.7); on the HGL the pressure is that of the surroundings, usually atmospheric. Let us consider point A on the stream tube in Fig. 5.7. The z portion of the piezometric head represents the elevation of point A. Therefore, the vertical distance from point A to the corresponding point on the HGL represents the remainder, which is the ***static pressure head*** p/γ in the flow at point A (see also Sec. 5.4). This is the difference in pressure *head* between the streamline and the surroundings, and of course this also indicates the difference in *pressure*.

A ***pitot tube,*** a small tube with its open end in the flow pointing upstream (Fig. 5.7 and Sec. 11.3), will intercept the kinetic energy of the flow in addition to the piezometric head, and so its liquid level indicates the ***total energy head,*** $p/\gamma + z + u^2/2g$. Therefore the vertical distance between the liquid surface in the piezometer tube and that in the pitot tube is $u^2/2g$, from which we can easily

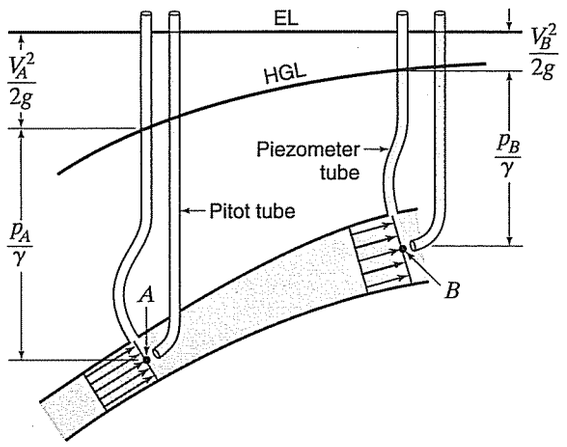


Figure 5.7
Ideal fluid.

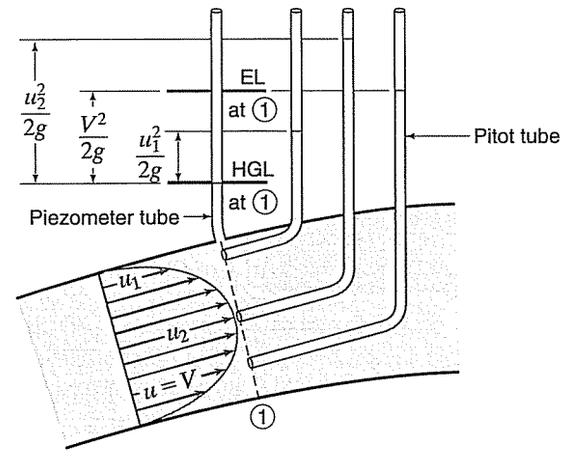


Figure 5.8
Real fluid.

compute the local flow velocity, u. The line drawn through the pitot-tube liquid surfaces (Fig. 5.7) is known as the ***energy line*** (EL). For the flow of an ideal fluid, as depicted in Fig. 5.7, the energy line is horizontal, because there is no head loss; for a real fluid, the energy line must slope downward in the direction of flow because of head loss due to fluid friction.

Because the local velocity u usually varies across a flow cross section, as shown in Fig. 5.8, the reading given by a pitot tube will depend on the precise location of its submerged open end. So a pitot tube will indicate the true level of the energy line only when we place it in the flow at a point where $u^2/2g = \alpha(V^2/2g)$, or, in other words, where $u = \sqrt{\alpha}V$. If we assume α (Sec. 5.1) has a value of 1.0, then, to indicate the true energy line, we must place the tube in the flow at a point where $u = V$. We rarely know ahead of time where in the flow $u = V$ (or $= \sqrt{\alpha}V$); so the correct positioning of a pitot tube, in order that it indicate the true position of the energy line, is generally unknown.

SAMPLE PROBLEM 5.10 Water flows in a wide open channel as shown in Fig. S5.10. Two pitot tubes are connected to a differential manometer containing a liquid ($s = 0.82$). Find u_A and u_B.

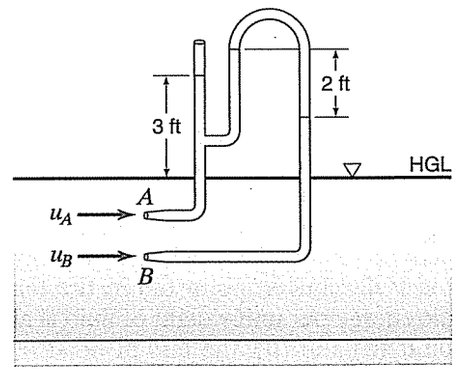


Figure S5.10

Solution

The water surface in the channel is the HGL, and the water surface in the pitot tube is at the EL, so the difference is

$$\frac{u_A^2}{2g} = 3 \text{ ft}$$

from which $$u_A = \sqrt{2(32.2)3} = 13.90 \text{ fps} \qquad \textit{ANS}$$

From Sec. 3.5, Fig. 3.14*b*, and Eq. (3.13*a*) for the manometer,

$$\frac{p_A}{\gamma} - \frac{p_B}{\gamma} = z_B - z_A + \left(1 - \frac{s_M}{s_F}\right)R_m \qquad (1)$$

The tip of piezometer A is a stagnation point (Sec. 4.10) where $V = 0$, so, considering Eq. (5.29) for the approaching streamline, with y_A as the depth of point A, we obtain

$$y_A + z_A + \frac{u_A^2}{2g} = \frac{p_A}{\gamma} + z_A + 0, \quad \text{i.e.,} \quad \frac{p_A}{\gamma} = \frac{u_A^2}{2g} + y_A$$

and, subtracting a similar equation for point B, we obtain

$$\frac{p_A}{\gamma} - \frac{p_B}{\gamma} = \frac{u_A^2}{2g} - \frac{u_B^2}{2g} + y_A - y_B$$

Substituting for $\Delta p/\gamma$ from manometer equation (1),

$$z_B - z_A + \left(1 - \frac{s_M}{s_F}\right)R_m = \frac{u_A^2}{2g} - \frac{u_B^2}{2g} + y_A - y_B$$

and, noting that $y_A + z_A = y_B + z_B =$ the elevation of the HGL, this simplifies to

$$\frac{u_A^2}{2g} - \frac{u_B^2}{2g} = \left(1 - \frac{s_M}{s_F}\right)R_m$$

i.e., $$3 - \frac{u_B^2}{2g} = (1 - 0.82)2 = 0.360 \text{ ft}$$

from which $$u_B = \sqrt{2(32.2)(3 - 0.360)} = 13.04 \text{ fps} \qquad \textit{ANS}$$

5.12 LOSS OF HEAD AT SUBMERGED DISCHARGE

When a fluid discharges with velocity V from the end of a pipe into a tank or reservoir that is so large that the velocity within it is negligible, the entire kinetic energy of the flow dissipates. We can see that this is so by examining Fig. 5.9. In the pipe up to point (*a*) the kinetic energy of the flowing fluid per unit weight of fluid is $V^2/2g$, but at point (*b*) in the tank the velocity is zero and hence the kinetic energy per unit weight of fluid is also zero. Because p and z are the same at points (*a*) and (*b*), the loss of head in this case, with ***submerged discharge,***

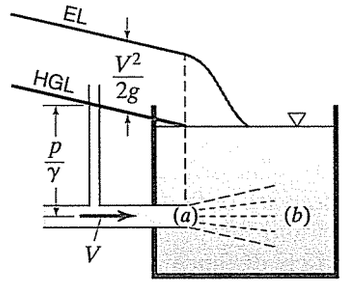


Figure 5.9

must be $V^2/2g$. The loss occurs after the fluid leaves the end of the pipe. This is a situation where fast-moving fluid impinges on stationary fluid. It is an impact situation not unlike that in which a fast-moving mudball collides with an immovable wall. The loss of head at submerged discharge into still water is $V^2/2g$, regardless of whether the fluid is ideal or real, compressible or incompressible. We shall consider this topic in more detail in Sec. 8.22.

5.13 APPLICATION OF HYDRAULIC GRADE LINE AND ENERGY LINE

Familiarity with the concepts of the energy line and hydraulic grade line is useful in the solution of flow problems involving incompressible fluids. In the piezometer tube at B in Fig. 5.10, the liquid in it will rise to a height BB' equal to the pressure head existing at that point. If the end of the pipe at E were closed so that no flow would take place, the water would rise in this column to M. The drop from M to B' when flow occurs is due to two factors, one being that part of the pressure head has transformed into the velocity head which the liquid has at

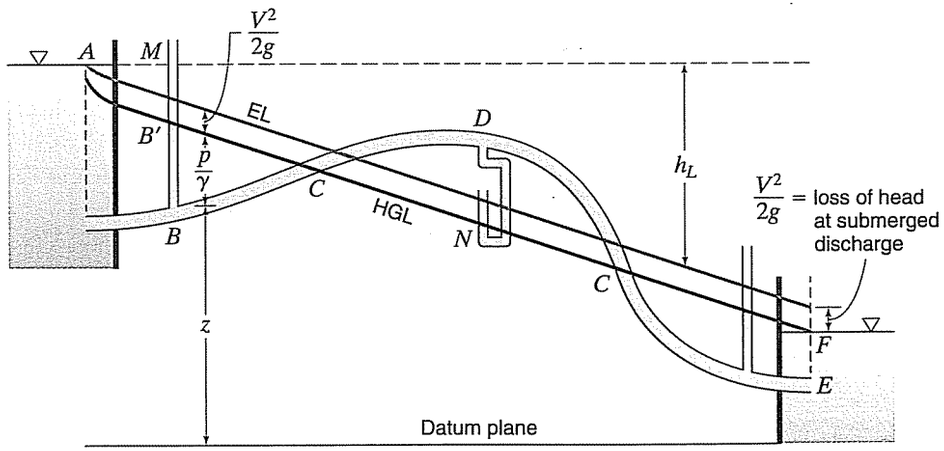


Figure 5.10
Hydraulic and energy grade lines.

B, and the other being that there is a loss of head due to fluid friction between A and B.

As noted in Sec. 5.11, if we connected a series of piezometers all along the pipe, the liquid would rise in them to various levels along what is called the ***hydraulic grade line*** (Figs. 5.7 and 5.10). We can see that the hydraulic grade line represents what would be the free surface if one could exist and maintain the same conditions of flow.

The hydraulic grade line indicates the *pressure* along the pipe, since at any point the vertical distance from the pipe to the hydraulic grade line is the pressure head at that point, assuming the profile is drawn to scale. At C this distance is zero, indicating that the absolute pressure in the pipe there is atmospheric. At D the pipe is above the hydraulic grade line, indicating that the pressure head there is $-DN$, or a vacuum of DN ft (or m) of liquid.

If we draw the profile of a pipeline to scale, not only does the hydraulic grade line enable us to determine the pressure head (and so the pressure) at any point by measuring the diagram, but it shows by mere inspection how the pressure varies over the entire length of the pipe. The hydraulic grade line is a straight line only if the pipe is straight and of uniform diameter and roughness (friction). But for the gradual curvatures that are often found in long pipelines, the deviation from a straight line will be small. Of course, if there are local losses of head, in addition to those due to normal pipe friction, there may be abrupt drops in the hydraulic grade line. Changes in diameter with resulting changes in velocity will also cause abrupt changes in the hydraulic grade line.

If the velocity head is constant, as in Fig. 5.10, the drop in the hydraulic grade line between any two points is equal to the loss of head between those two points, and so the slope of the hydraulic grade line is a measure of the rate of loss. In Fig. 5.11, for example, the rate of loss in the larger pipe (lower velocity) is much less than in the smaller pipe (higher velocity). If the velocity changes, the hydraulic grade line might actually rise in the direction of flow, as in Figs. 5.11 and 5.12.

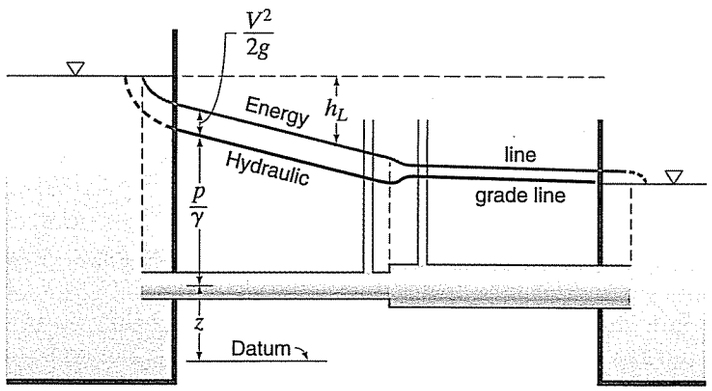


Figure 5.11
(Plotted to scale).

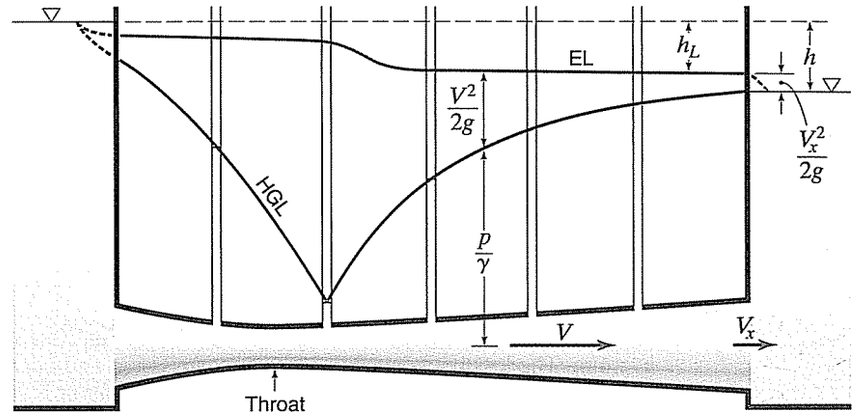


Figure 5.12
(Plotted to scale).

The vertical distance from the level of the surface at A in Fig. 5.10 down to the hydraulic grade line for any point represents the h_L from A to the point in question plus $V^2/2g$ at that point. Thus the position of the grade line does not depend on the position of the pipe. Therefore we need not compute pressure heads at various points in the pipe to plot the hydraulic grade line. Instead, we can set off values of $V^2/2g + h_L$ from A to various points, below the horizontal line through A, and this procedure is often more convenient. If the pipe diameter is uniform, we need only locate a few points, and often two are sufficient.

If Fig. 5.10 represents to scale the profile of a pipe of uniform diameter, we can draw the hydraulic grade line as follows. At the intake to the pipe there will be a drop below the surface at A, which we should set off equal to $V^2/2g$ plus a local entrance loss. (This latter we explain in Sec. 8.21.) At E the pressure is EF, and hence the grade line must end at the surface at F. If the pipe discharged freely into the air at E, the line would pass through E. We can compute the location of other points, such as B' and N, if desired. In the case of a *long* pipe of uniform diameter the error is very small if we draw the hydraulic grade line as a straight line from A to F for a submerged discharge, or from A to E for a free discharge into the atmosphere.

If we set off values of h_L below the horizontal line through A, the resulting line represents values of the total energy head H measured above any arbitrary datum plane inasmuch as the line is above the hydraulic grade line by an amount equal to $V^2/2g$. This line is the ***energy grade line,*** usually known as simply the ***energy line*** (see also Sec. 5.11). It shows the rate at which the energy decreases, and it must always drop downward in the direction of flow unless energy is added by a pump. The energy line *also* does not depend on the position of the pipeline.

Energy lines are shown in Figs. 5.10–5.12. The last one, plotted to scale, shows that the chief loss of head is in the diverging portion and just beyond the ***throat*** (section of minimum diameter). In all three of these cases the discharge

is submerged and so the velocity head is lost at discharge (Sec. 5.12). But note in Fig. 5.12 how the conical ***diffuser*** (diverging pipe) greatly reduces this loss, because the enlarged discharge area reduces the velocity at discharge. The large pressure changes that occur in converging–diverging pipes similar to Fig. 5.12 provide a very convenient means of measuring flow rates, which we will discuss in Sec. 11.7.

EXERCISES

5.13.1 Assume there is friction head loss in the siphon of Fig. X5.13.1, where $a = 1$ m, $b = 4$ m. The loss between the intake and B is 0.6 m and between B and N is 0.9 m. What is the rate of discharge and pressure head at B when the diameter is 150 mm?

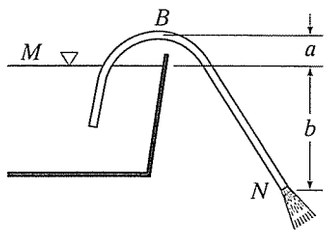


Figure X5.13.1

5.13.2 Assume there is friction head loss in the siphon of Fig. X5.13.1, where $a = 3$ ft, $b = 12$ ft. The loss between the intake and B is 2.5 ft and that between B and N is 3 ft. What is the rate of discharge and pressure head at B when the diameter is 6 in?

5.13.3 Refer to Fig. X5.13.1. Find the maximum value for b if $a = 1.1$ m. Assume friction is negligible and the minimum pressure allowable in the siphon is a vacuum of -9.8 m of water.

5.13.4 Refer to Fig. X5.13.1. Find the maximum value for b if $a = 3.5$ ft. Assume friction is negligible and the minimum pressure allowable in the siphon is a vacuum of -32.8 ft of water.

5.13.5 A pump, having an efficiency of 90%, lifts water to a height of 465 ft at the rate of 250 cfs. The friction head loss in the pipe is 35 ft. What is the required horsepower? Also sketch the energy line and the hydraulic grade line of this system.

5.13.6 A pump, having an efficiency of 90%, lifts water to a height of 155 m at the rate of 7.5 m^3/s. The friction head loss in the pipe is 13 m. What is the required pump power in kW? Also sketch the energy line and the hydraulic grade line of this system.

5.13.7 In Fig. X5.13.7 let $a = 25$ ft, $b = 60$ ft, $c = 40$ ft, and $d = 2$ ft. All the losses of energy are to be ignored when the stream discharging into the air at E has a diameter of 4 in. What are pressure heads at B, C, and D if the diameter of the vertical pipe is 5 in?

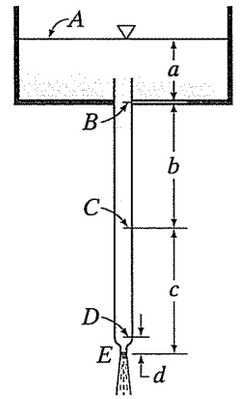


Figure X5.13.7

5.13.8 In Fig. X5.13.7 let $a = 7.5$ m, $b = c = 15$ m, and $d = 300$ mm. All the losses of energy are to be ignored when the stream discharging into the air at E has a diameter of 80 mm. What are pressure heads at B, C, and D if the diameter of the vertical pipe is 120 mm?

5.14 METHOD OF SOLUTION OF LIQUID FLOW PROBLEMS

For the solutions of problems of (incompressible) liquid flow, there are two fundamental equations: the equation of continuity (4.17) and the energy equation in one of the forms from Eqs. (5.22)–(5.29). We may employ the following procedure:

1. Choose a datum plane through any convenient (lower) point.
2. Note at what sections we know or must assume the velocity. If at any point the section area is great compared with its value elsewhere, the velocity head is so small that we may disregard it.
3. Note at what points we know or must assume the pressure. In a body of liquid at rest with a free surface we know the pressure at every point within the body. The pressure in a jet is the same as that of the medium surrounding the jet.
4. Note whether or not there is any point where we know all three terms, pressure, elevation, and velocity.
5. Note whether or not there is any point where there is only one unknown quantity.

Generally we can write an energy equation that will fulfill conditions 4 and 5. If there are two unknowns in the equation then we must also use the continuity equation. For the application of these principles see Sample Probs. 5.11 and 5.12.

SAMPLE PROBLEM 5.11 In a fire fighting system, a pipeline with a pump leads to a nozzle as shown in Fig. S5.11. Find the flow rate when the pump develops a head of 80 ft, given that we may express the friction head loss in the 6-in-diameter pipe by $h_f = 5V_6^2/2g$, and the friction head loss in the 4-in-diameter pipe by $h_f = 12V_4^2/2g$. (*a*) Sketch the energy line and hydraulic grade line. (*b*) Find the pressure head at the suction side of the pump. Find (*c*) the power delivered to the water by the pump, and (*d*) the power of the jet.

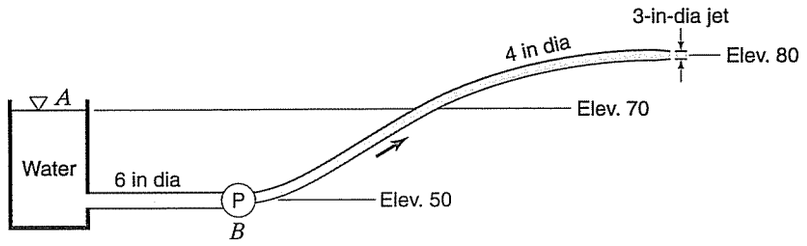


Figure S5.11

Solution

(*a*) Select the datum as the elevation of the water surface in the reservoir. If V_3 is the jet velocity, note from continuity Eq. (4.17) that

$$V_6 = \left(\tfrac{3}{6}\right)^2 V_3 = 0.25V_3, \qquad V_4 = \left(\tfrac{3}{4}\right)^2 V_3 = 0.563V_3$$

Writing energy equation (5.27) from the surface of the reservoir (point 1) to the jet (point 3),

$$\left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) - h_{f_6} + h_p - h_{f_4} = \frac{p_3}{\gamma} + z_3 + \frac{V_3^2}{2g}$$

$$0 + 0 + 0 - 5\frac{V_6^2}{2g} + 80 - 12\frac{V_4^2}{2g} = 0 + 10 + \frac{V_3^2}{2g}$$

Use the continuity relations to express all velocities in terms of V_3:

$$-\frac{5(0.25V_3)^2}{2g} + 80 - 12\frac{(0.563V_3)^2}{2g} = 10 + \frac{V_3^2}{2g}$$

from which $$V_3 = 29.7 \text{ fps}$$

Eq. (4.17): $$Q = A_3V_3 = \frac{\pi}{4}\left(\frac{3}{12}\right)^2 29.7 = 1.458 \text{ cfs} \qquad \textbf{\textit{ANS}}$$

Friction head loss in suction pipe:

$$h_{f_6} = 5\frac{V_6^2}{2g} = \frac{5(0.25V_3)^2}{2g} = \frac{0.312V_3^2}{2g} = 4.28 \text{ ft}$$

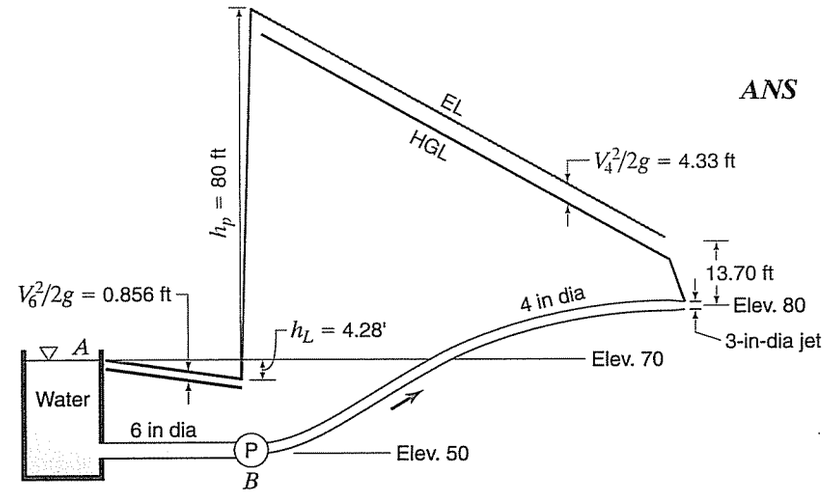
Friction head loss in discharge pipe:

$$h_{f_4} = 12\frac{V_4^2}{2g} = \frac{12(0.563V_3)^2}{2g} = 52.0 \text{ ft}$$

$$\frac{V_3^2}{2g} = 13.70 \text{ ft}, \quad \frac{V_4^2}{2g} = 4.33 \text{ ft}, \quad \frac{V_6^2}{2g} = 0.856 \text{ ft}$$

Drawing the energy line and hydraulic grade line on the figure to scale:

ANS



(*b*) From the figure we see that the pressure head on the suction side of the pump is

$$p_B/\gamma = 70 - 50 - 4.28 - 0.856 = 14.86 \text{ ft} \qquad \textbf{\textit{ANS}}$$

Likewise, we can find the pressure head at any point in the pipe if the figure is to scale.

(*c*) Eq. (5.40): $$P_{\text{deliv. by pump}} = \frac{\gamma Q h_p}{550} = \frac{62.4(1.458)80}{550} = 13.23 \text{ hp} \qquad \textbf{\textit{ANS}}$$

(*d*) Eq. (5.40) and Sec. 5.9: $$P_{\text{jet}} = \frac{\gamma Q(H_j - z_j)}{550} = \frac{\gamma Q}{550}\left[\frac{V_3^2}{2g}\right]$$

$$= \frac{62.4(1.458)13.70}{550} = 2.27 \text{ hp} \qquad \textbf{\textit{ANS}}$$

SAMPLE PROBLEM 5.12 Find the flow rate per meter width for the two-dimensional channel flow shown in Fig. S5.12. Assume no head loss.

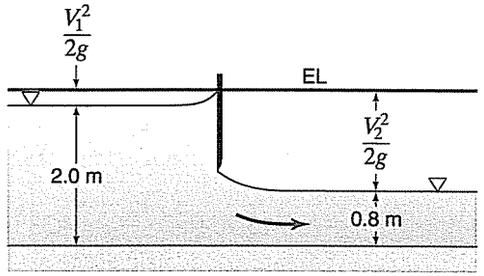


Figure S5.12

Solution

Select the datum as the (effectively horizontal) channel bed. The water surface represents the hydraulic grade line in the region where the streamlines are parallel. The energy line is a distance $V^2/2g$ above the water surface, assuming $\alpha = 1.0$. If there is a no head loss, the energy line is horizontal. Writing the energy equation (5.29) from section 1 to 2, we have

$$0 + 2.0 + \frac{V_1^2}{2g} = 0 + 0.8 + \frac{V_2^2}{2g} \tag{1}$$

Note that this applies either (*a*) between points on the water surface, with $p_1 = p_2 = 0$, $z_1 = 2.0$, and $z_2 = 0.8$, or (*b*) between points on the bed, with $z_1 = z_2 = 0$, $p_1/\gamma = 2.0$, and $p_2/\gamma = 0.8$.

But from the continuity equation (4.17), for 1 m of channel width perpendicular to the figure,

$$(2 \times 1)V_1 = (0.8 \times 1)V_2 \tag{2}$$

Substituting Eq. (1) into Eq. (2), and using $g = 9.81$ m/s^2, we obtain

$$V_1 = 2.12 \text{ m/s}, \quad V_2 = 5.29 \text{ m/s}, \quad \frac{V_1^2}{2g} = 0.229 \text{ m}, \quad \frac{V_2^2}{2g} = 1.429 \text{ m}$$

and $Q = A_1V_1 = (2 \times 1)2.12 = 4.24$ m^3/s (for 1 m of channel width) ***ANS***

EXERCISES

5.14.1 Refer to Fig. S5.12. If the depths upstream and downstream of the gate were 7.5 ft and 3.0 ft respectively, find the flow rate per foot of channel width. Assume no head loss.

5.14.2 Refer to Fig. S5.12. If the depths upstream and downstream of the gate were 1.5 m and 0.6 m respectively, find the flow rate per meter of channel width. Assume no head loss.

5.14.3 Refer to Fig. S5.12. Suppose the gate opening is set so the depth downstream is 2.0 ft. Find the upstream depth under these conditions if the flow rate is 45 ft^3/sec per ft of width. Assume no head loss.

5.14.4 Refer to Fig. S5.12. Suppose the gate opening is set so the depth downstream is 0.7 m. Find the upstream depth under these conditions if the flow rate is 4.24 m^3/s per m of width. Assume no head loss.

5.15 JET TRAJECTORY

A free liquid jet in air will describe a ***trajectory,*** or path under the action of gravity, with a vertical velocity component that is continually changing. The trajectory is a streamline, and consequently, if we neglect air friction, we can apply Bernoulli's theorem to it, with all the pressure terms zero. Thus the sum of the elevation and velocity head is the same for all points of the curve. The energy grade line is a horizontal line at distance $V_0^2/2g$ above the nozzle, where V_0 is the initial velocity of the jet as it leaves the nozzle (Fig. 5.13).

We can obtain the equation for the trajectory by applying Newton's equations of uniformly accelerated motion to a particle of the liquid traveling in time t from the nozzle to point P, whose coordinates are (x, z). Then $x = V_{x0}t$ and $z = V_{z0}t - \frac{1}{2}gt^2$. Solving for t from the first equation and substituting it into the second gives

$$z = \frac{V_{z0}}{V_{x0}}x - \frac{g}{2V_{x0}^2}x^2 \tag{5.44}$$

By setting $dz/dx = 0$, we find that z_{max} occurs when $x = V_{x0}V_{z0}/g$. Substituting this value for x in Eq. (5.44) gives $z_{max} = V_{z0}^2/2g$. Thus Eq. (5.44) is that of an inverted parabola having its vertex at $x = V_{x0}V_{z0}/g$ and $z = V_{z0}^2/2g$. Since the velocity at the

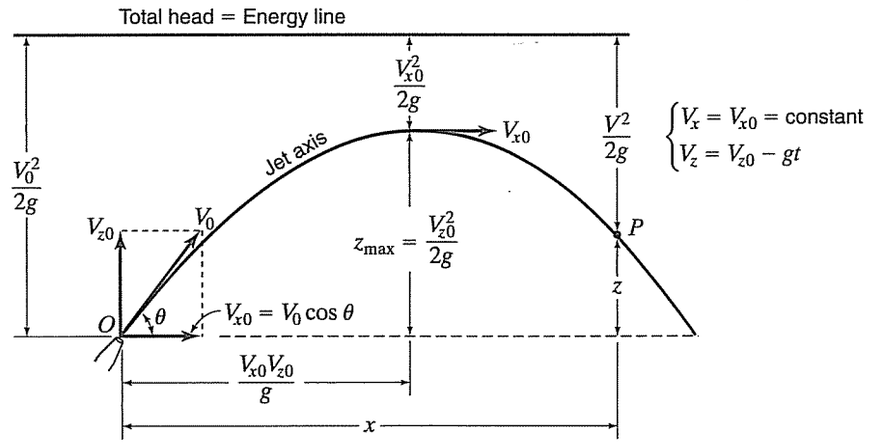


Figure 5.13
Jet trajectory.

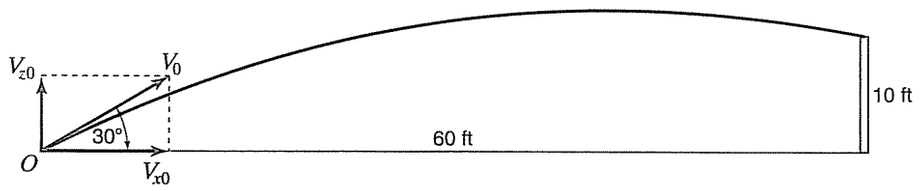
top of the trajectory is horizontal and equal to V_{x0}, the distance from this point to the energy line is evidently $V_{x0}^2/2g$. We can obtain this in another way by considering that $V_0^2 = V_{x0}^2 + V_{z0}^2$. Dividing each term by $2g$ gives the relations shown in Fig. 5.13.

If the jet is initially horizontal, as in the flow from a vertical orifice, $V_{x0} = V_0$ and $V_{z0} = 0$. Equation (5.44) then readily reduces to an expression for the initial jet velocity in terms of the coordinates from the vena contracta (Fig. 11.14) to any point of the trajectory, z now being positive downward:

$$V_0 = x\sqrt{\frac{g}{2z}} \tag{5.45}$$

SAMPLE PROBLEM 5.13 If a water jet is inclined upward 30° from the horizontal, what minimum initial velocity will enable it to reach over a 10-ft wall at a horizontal distance of 60 ft, neglecting friction?

Solution



$$V_{x0} = V_0 \cos 30° = 0.866V_0$$
$$V_{z0} = V_0 \sin 30° = 0.5V_0$$

From Newton's laws,

$$x = 0.866V_0 t = 60 \tag{1}$$

$$z = 0.5V_0 t - 0.5gt^2 = 10 \tag{2}$$

From (1), $t = 69.3/V_0$. Substituting this into (2),

$$0.5V_0\frac{69.3}{V_0} - \frac{32.2}{2}\left(\frac{69.3}{V_0}\right)^2 = 10$$

from which $V_0^2 = 3140$, or $V_0 = 56.0$ fps ***ANS***

Plumes

When one fluid (specific gravity s_1) discharges into a second fluid (s_2) with a similar density, a plume of the first fluid forms. We are familiar with smoke and steam plumes; similar plumes form when treated sewage effluent discharges under the ocean from outfall sewers (see Exer. 5.15.3). Such plumes rise because $s_1 < s_2$. To a first approximation, neglecting fluid friction and mixing, if the second fluid is not moving then we may compute the path of the plume as a jet trajectory. However, we must then replace the gravitational acceleration in the trajectory equations by the force per unit mass on the plume fluid, which is

$$g' = \frac{(\rho_1 - \rho_2)g\forall}{\rho_1\forall} = \frac{(s_1 - s_2)g}{s_1} = g\left(1 - \frac{s_2}{s_1}\right) \tag{5.46}$$

For a rising plume, both g' and z will be negative.

EXERCISES

5.15.1 A jet issues horizontally from an orifice in the vertical wall of a large tank (Fig. X5.15.1). Neglecting air resistance, determine the velocity of the jet at the orifice for the following variety of trajectories: (*a*) $x = 1.0$ m, $y = 1.0$ m; (*b*) $x = 2.0$ m, $y = 2.0$ m; (*c*) $x = 3.0$ m, $y = 3.0$ m; (*d*) $x = 4.0$ m, $y = 4.0$ m. Express the answers in m/s.

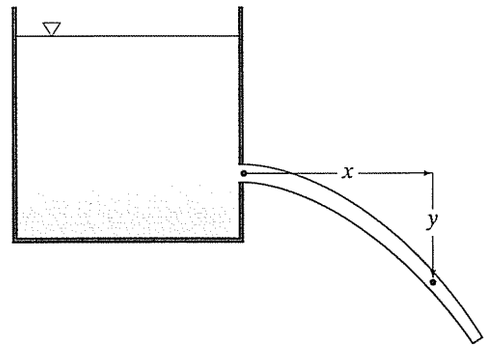


Figure X5.15.1

5.15.2 A water jet must reach the window in the wall shown in Fig. X5.15.2. Assuming a jet velocity of 25 m/s at the nozzle and neglecting air friction, find the angle (or angles) of inclination θ which will achieve this result, given $h = 14$ m, $d = 23$ m, and $a = 2$ m.

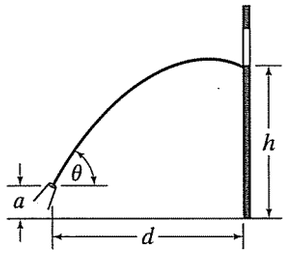


Figure X5.15.2

5.15.3 Freshwater sewage effluent discharges from a horizontal outfall pipe on the floor of the ocean at a point where the depth is 120 ft. When the ocean is still, the jet is observed to rise to the surface at a point 95 ft horizontally from the end of the pipe. Assuming the ocean water to have a specific gravity of 1.03 and neglecting fluid friction and mixing of the jet with the ocean water, find the velocity at the end of the outfall.

5.16 FLOW IN A CURVED PATH

The energy equations we developed previously apply fundamentally to flow along a streamline or along a stream of large cross section if we use certain average values. Now we will investigate conditions in a direction normal to a streamline. Figure 5.14 represents an element of fluid moving in a *horizontal* plane[8] with a velocity V along a curved path of radius r. The element has a linear dimension dr in the plane of the paper and an area dA normal to the plane of the paper. The mass of this fluid element is $\rho\, dA\, dr$, and the normal component of acceleration is V^2/r. Thus the centripetal force acting on the element toward the center of curvature is $\rho\, dA\, dr\, V^2/r$. As the radius increases from r to $r + dr$, the pressure will change from p to $p + dp$. Therefore the resultant force in the direction of the center of curvature is $dp dA$. Equating these two forces and dividing by dA,

$$dp = \rho \frac{V^2}{r} dr \tag{5.47}$$

When horizontal flow is in a straight line for which r is infinity, the value of dp is zero. That is, no difference in pressure can exist in the horizontal direction perpendicular to horizontal flow in a straight line.

Because dp is positive if dr is positive, Eq. (5.47) shows that pressure increases from the concave to the convex side of the stream, but the exact way in which it increases depends on the way in which V varies with the radius. If we can express V as a function of r, or if V is constant, we can integrate Eq. (5.47) to find $p_{\text{outer}} - p_{\text{inner}}$. Usually V is *not* constant. In Secs. 5.17 and 5.18 we will consider two important practical cases in which V varies with the radius in two different ways.

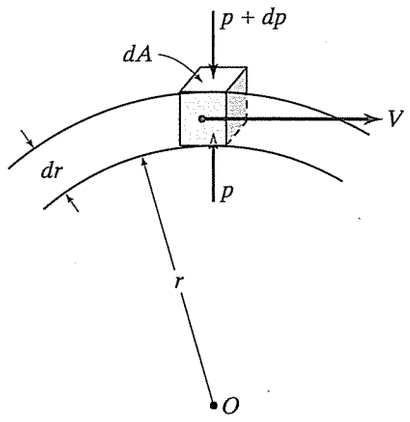


Figure 5.14
Circular motion in a horizontal plane.

[8] A more generalized analysis of flow along a curved path in a vertical or inclined plane leads to a result that includes z terms.

EXERCISES

5.16.1 Figure X5.16.1 shows a two-dimensional ideal flow in a vertical plane. Data are as follows: $r = 12$ ft, $b = 5$ ft, $\gamma = 62.4$ lb/ft^3, $V = 24$ fps. If the pressure at A is 6 psi, find the pressure at B.

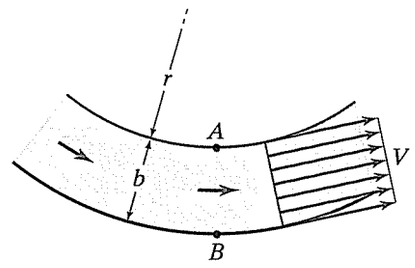


Figure X5.16.1

5.16.2 Refer to Fig X5.16.1. Flow occurs in a vertical plane. Data are as follows: $r = 7$ m, $b = 3$ m, $\gamma = 9.81$ kN/m^3, $V = 5$ m/s. Find the pressure at A if the pressure at B is 150 kPa.

5.17 FORCED OR ROTATIONAL VORTEX

In theory, we can make a fluid rotate as a solid body without relative motion between particles, either by rotating the containing vessel or by stirring the contained fluid. Thus, in one way or another, we apply an external torque. A common example is the rotation of liquid within a centrifugal pump or of gas in a centrifugal compressor.

Cylindrical Forced Vortex

If the entire body of fluid rotates as a solid then V varies directly with r; that is, $V = r\omega$, where ω (omega) is the imposed angular velocity. Substituting this for V in Eq. (5.47), for the case of rotation in any horizontal plane about a vertical axis, we have

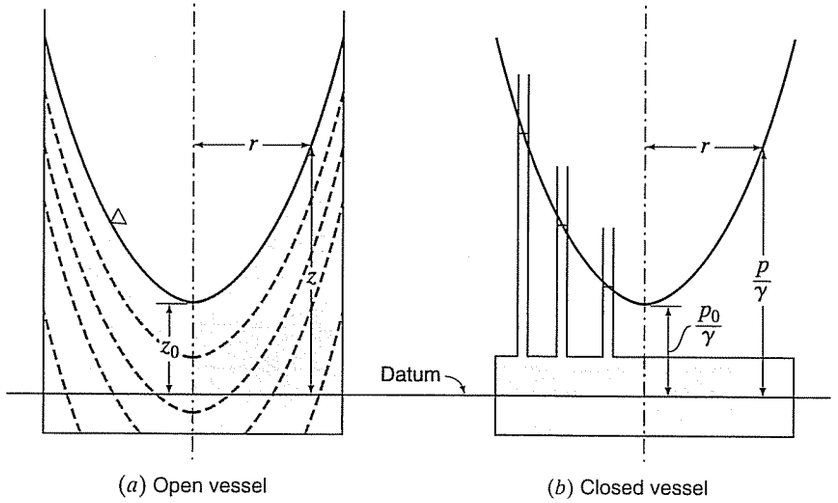
$$dp = \rho\omega^2 r\, dr = \frac{\gamma}{g}\omega^2 r\, dr$$

Between any two radii r_1 and r_2, we can integrate this to give

$$\frac{p_2}{\gamma} - \frac{p_1}{\gamma} = \frac{\omega^2}{2g}\left(r_2^2 - r_1^2\right) \tag{5.48}$$

which is the pressure head difference between two points on the same horizontal plane. If p_0 is the pressure when $r_1 = 0$, Eq. (5.48) becomes

$$\frac{p}{\gamma} = \frac{\omega^2}{2g} r^2 + \frac{p_0}{\gamma} \tag{5.49}$$



(*a*) Open vessel (*b*) Closed vessel

Figure 5.15
Forced vortex.

which we recognize as the equation of a parabola. In Fig. 5.15*a* we see that if the fluid is a liquid then the pressure head p/γ at any point is equal to z, the depth of the point below the free surface. Therefore we may also write the preceding equations as

$$z_2 - z_1 = \frac{\omega^2}{2g}(r_2^2 - r_1^2) \tag{5.50}$$

and

$$z = \frac{\omega^2}{2g}r^2 + z_0 \tag{5.51}$$

where z_0 is the depth when $r_1 = 0$. Equations (5.50) and (5.51) define the free surface, if one exists, and in general they define *any* surface of equal pressure in the liquid; these surfaces are a series of paraboloids of the same shape as the free surface, such as the dashed curves in Fig. 5.15*a*.

For the *open* vessel of Fig. 5.15*a*, the pressure head at any point is equal to its depth below the free surface. If the liquid is *confined* within a vessel, as in Fig. 5.15*b*, the pressure will vary along any radius in just the same way as if there were a free surface. Therefore the two are equivalent.

In this discussion we assumed the axis of the vessel was vertical; however, the axis might be inclined. Since pressure varies with elevation z as well as radius, a more general equation applicable to fluid in a *closed* tank with an inclined axis is

$$\frac{p_2}{\gamma} - \frac{p_1}{\gamma} + z_2 - z_1 = \frac{\omega^2}{2g}(r_2^2 - r_1^2) \tag{5.52}$$

Equation (5.48) is the special case where $z_1 = z_2$ (closed tank with vertical axis), and Eq. (5.50) is the special case where $p_1 = p_2$ (open tank with vertical axis).

Note that Eqs. (5.48)–(5.52) are *not* energy equations, since they represent conditions *across* streamlines rather than along a streamline.

Spiral Forced Vortex

So far we have confined this discussion to the rotation of all particles in concentric circles. Suppose that now we superimpose a flow with a velocity having radial components either outward or inward. If the height of the walls of the open vessel in Fig. 5.15*a* were less than that of the liquid surface, and if we supplied liquid to the center at the proper rate by some means, then it is obvious that liquid would flow outward. If, on the other hand, liquid flowed into the tank over the rim from some source at a higher elevation and flowed out at the center, the flow would be inward. The combination of this approximately radial flow with the circular flow would result in path lines that were some form of spirals.

If the closed vessel in Fig. 5.15*b* is arranged with suitable openings near the center and also around the periphery, and if it is provided with vanes, as shown in Fig. 5.16, it becomes either a centrifugal pump impeller or a turbine runner, as the case may be. The vanes constrain the flow of the liquid and determine both its relative magnitude and its direction. If the area of the passages normal to the direction of flow is A, the equation of continuity fixes the relative velocities, since

$$Q = A_1 v_1 = A_2 v_2 = \text{constant}$$

This relative flow is the flow as it would appear to an observer or a camera revolving with the rotor. Neglecting friction losses and assuming a vertical axis of rotation, we can use the energy equation to find that the pressure difference due to this superimposed flow alone is $p_2/\gamma - p_1/\gamma = (v_1^2 - v_2^2)/2g$.

For the case of rotation with flow (i.e., spiral forced vortex), we can find the total pressure difference between two points by adding together the pressure differences due to the two flows considered separately. That is, for the case of a vertical axis,

$$\frac{p_2}{\gamma} - \frac{p_1}{\gamma} = \frac{\omega^2}{2g}(r_2^2 - r_1^2) + \frac{v_1^2 - v_2^2}{2g} \tag{5.53}$$

Of course, friction losses will modify this result to some extent. If the axis is inclined, we must add z terms to the equation. We see that Eq. (5.48) is a special case of Eq. (5.53) when $v_1 = v_2$, either when both are finite or when both are zero.

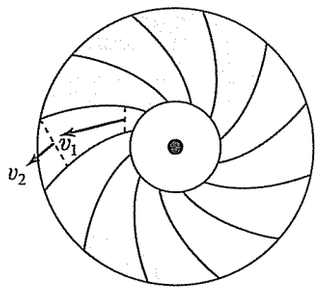


Figure 5.16
Flow through a rotor.

For a forced vortex with spiral flow, a pump adds energy to the fluid and a turbine extracts energy from it. In the limiting case of zero flow, when all path lines become concentric circles (i.e., a cylindrical forced vortex), a real fluid still needs energy input from some external source to maintain the rotation.

EXERCISES

5.17.1 A 16-in-diameter closed vessel completely filled with fluid rotates at 1800 rpm. What will be the pressure difference between the circumference and the axis of rotation in feet of the fluid and in pounds per square inch if the fluid is (*a*) air with a specific weight of 0.076 lb/ft^3; (*b*) water a 70°F; (*c*) oil with a specific weight of 46 lb/ft^3?

5.17.2 A 1.2-m-diameter closed vessel completely filled with oil ($\gamma = 8.3$ kN/m^3) rotates at 400 rpm. What will be the pressure difference between the circumference and the axis of rotation? Express the answer in Pa.

5.17.3 A 2-ft-diameter open cylindrical vessel partially filled with water rotates about its vertical axis. How many revolutions per minute would cause the water surface at the periphery to be 5 ft higher than the water surface at the axis? What would be the necessary speed for the same conditions if the fluid were mercury?

5.18 FREE OR IRROTATIONAL VORTEX

In a free vortex there is no expenditure of energy whatever from an outside source, and the fluid rotates because of some rotation previously imparted to it or due to some internal action. Some examples are a whirlpool in a river, the rotary flow that often arises in a shallow vessel when liquid flows out through a hole in the bottom (as we often see when water empties from a bathtub), and the flow in a centrifugal-pump casing just outside the impeller or that in a turbine casing as the water approaches the guide vanes.

As the fluid receives no additional energy, it follows that, neglecting friction, H is constant throughout; that is, $p/\gamma + z + V^2/2g = \text{constant}$.

Cylindrical Free Vortex

The angular momentum with respect to the center of rotation of a particle of mass m moving along a circular path of radius r at a velocity V_u is mV_ur, where V_u is the velocity along the circular path (i.e., tangential velocity).[9] Newton's second law states that, for the case of rotation, the torque is equal to the time rate of change of angular momentum. Hence torque $= d(mV_ur)/dt$. For a free vortex (frictionless fluid) no torque is applied; therefore $mV_ur = \text{constant}$, and thus $V_ur = C$, where we can determine the value of C by knowing the value of V at some radius r. Assuming a vertical axis of rotation and substituting $V_u = C/r$ in

[9] In this chapter we use V_u to represent the tangential component of velocity. Other symbols commonly used to represent tangential velocity are V_t and V_θ.

Eq. (5.47), we obtain

$$dp = \rho \frac{C^2}{r^2}\frac{dr}{r} = \frac{\gamma}{g}\frac{C^2}{r^3}dr$$

We can integrate this between any two radii r_1 and r_2, to get

$$\frac{p_2}{\gamma} - \frac{p_1}{\gamma} = \frac{C^2}{2g}\left(\frac{1}{r_1^2} - \frac{1}{r_2^2}\right) = \frac{V_{u1}^2}{2g}\left[1 - \left(\frac{r_1}{r_2}\right)^2\right] \tag{5.54}$$

If there is a free surface, the pressure head p/γ at any point is equal to the depth below the surface. Also, at any radius the pressure varies in the vertical direction according to the hydrostatic law. So this equation is merely a special case in which $z_1 = z_2$.

As $H = p/\gamma + z + V_u^2/2g = \text{constant}$, it follows that at any radius r

$$\frac{p}{\gamma} + z = H - \frac{V_u^2}{2g} = H - \frac{C^2}{2gr^2} = H - \frac{V_{u1}^2}{2g}\left(\frac{r_1}{r}\right)^2 \tag{5.55}$$

Assuming a vertical axis, we can find the pressure along the radius from this equation by taking z constant; and for any constant pressure p, we can find values of z determining a surface of equal pressure. If p is zero, the values of z determine the free surface (Fig. 5.17*a*), if one exists.

Equation (5.55) indicates that H is the asymptote which $p/\gamma + z$ approaches as r approaches infinity and V_u approaches zero. On the other hand, as r approaches zero, V_u approaches infinity, and $p/\gamma + z$ approaches minus infinity. Since this is physically impossible, the free vortex cannot extend to the axis of rotation. In reality, since high velocities are attained as the axis is approached, the friction losses, which vary as the square of the velocity, become of increasing importance and are no longer negligible. In this region, then, the assumption that H is constant no longer holds; the core of the vortex tends to rotate as a solid body, as in the central part of Fig. 5.17*b*.

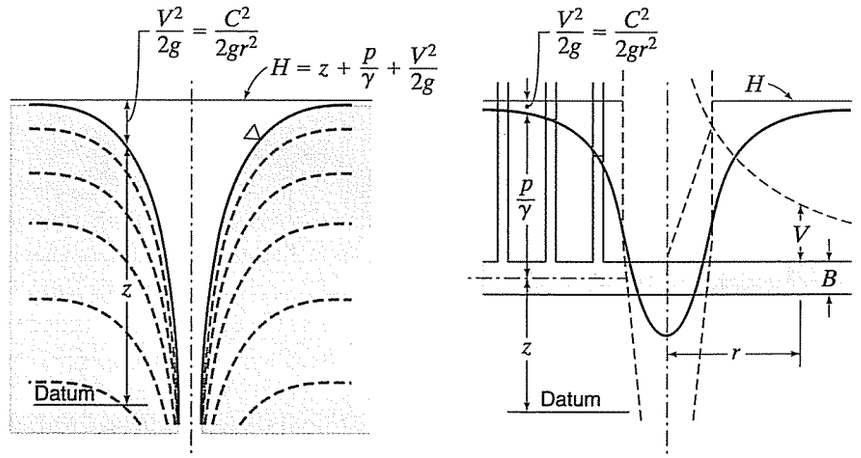


Figure 5.17 Free vortex.

(*a*) Free surface (*b*) Fluid enclosed

Spiral Free Vortex

If we superimpose a radial flow upon the concentric flow just discussed, the path lines will then be spirals. If the flow passes out through a circular hole in the bottom of a shallow vessel, the surface of the liquid takes the form shown in Fig. 5.17*a*, with an air ***core*** sucked down the hole. If there is an outlet symmetric with the axis as in Fig. 5.17*b*, we might have a flow component either radially inward or radially outward. If the two confining plates shown are a constant distance B apart, the radial flow component with velocity V_r is then across a series of concentric cylindrical surfaces whose area is $2\pi rB$. Thus

$$Q = 2\pi rBV_r = \text{constant}$$

from which we see that $rV_r = \text{constant}$. Therefore the radial velocity varies in the same way with r that the circumferential velocity did in the preceding discussion of the cylindrical free vortex. The pressure variation in a spiral free vortex (Fig. 5.17*b*) is given by

$$\frac{p_2}{\gamma} - \frac{p_1}{\gamma} = \frac{V_1^2}{2g} - \frac{V_2^2}{2g} \tag{5.56}$$

where $V = \sqrt{V_r^2 + V_u^2}$, the velocity of flow.

SAMPLE PROBLEM 5.14 A centrifugal pump with a 12-in-diameter impeller is inside a casing that has a constant height of 1.5 in between sections a and b and that then enlarges into a volute at c (Fig. S5.14). Water leaves the impeller with a velocity of 60 fps at an angle of 15° with the tangent. (*a*) At what rate is water flowing through the pump? (*b*) Neglecting friction, what will be the magnitude and direction of the velocity at b and what will be the gain in pressure head from a to b?

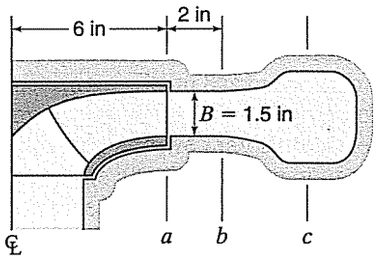


Figure S5.14

Solution

(*a*) Flow through the pump $Q = A_a(V_r)_a = 2\pi r_a B(V_r)_a$, where $(V_r)_a = 60\sin 15° = 15.53$ fps

$$Q = 2\pi(6/12)(1.5/12)15.53 = 6.10 \text{ cfs} \qquad \textbf{\textit{ANS}}$$

(*b*) From continuity: $Q = 2\pi r_a B(V_r)_a = 2\pi r_b B(V_r)_b$,

so
$$\frac{(V_r)_b}{(V_r)_a} = \frac{r_a}{r_b} = \frac{6}{8}$$

Because torque = 0 in the space between a and b, angular momentum must be conserved. Thus

$$m(V_u)_a r_a = m(V_u)_b r_b$$

so
$$\frac{(V_u)_b}{(V_u)_a} = \frac{r_a}{r_b} = \frac{6}{8}$$

The region between a and b is a spiral free vortex.

Because we have found that V_u and V_r both decrease in the same proportion as flow moves from a to b, the angle α does not change, and

$$V_b/V_a = \tfrac{6}{8}; \quad V_b = \left(\tfrac{6}{8}\right)60 = 45 \text{ fps} \quad \text{at } 15° \text{ with the tangent} \qquad \textbf{\textit{ANS}}$$

Finally, writing the energy equation along the flow lines gives

$$\frac{p_b}{\gamma} - \frac{p_a}{\gamma} = \frac{V_a^2}{2g} - \frac{V_b^2}{2g} = \frac{60^2 - 45^2}{2(32.2)} = 24.5 \text{ ft} \qquad \textbf{\textit{ANS}}$$

EXERCISES

5.18.1 Refer to Sample Prob. 5.14. If the impeller diameter is 220 mm, the casing height is 40 mm between a and b, and water leaves the impeller with a velocity of 18 m/s at an angle of 16° with the tangent, find the flow rate, the magnitude and direction of the velocity at b (where $r = 160$ mm), and the pressure increase from a to b. Neglect friction.

5.18.2 Refer to Sample Prob. 5.14. If the impeller diameter is 10 in, the casing height is 1.8 in between a and b, and water leaves the impeller with a velocity of 50 fps at an angle of 16° with the tangent, find the flow rate, the magnitude and direction of the velocity at b (where $r = 7$ in), and the pressure increase from a to b. Neglect friction.

PROBLEMS

5.1 Assume the seventh-root law (Eq. 8.49) for a turbulent-velocity distribution between two smooth flat plates. Find α.

5.2 Assume the seventh-root law (Eq. 8.49) for a turbulent-velocity distribution in smooth pipe flow. Find α.

5.3 Water flows through a long, horizontal, conical diffuser at the rate of 4.2 m³/s. The diameter of the diffuser changes from 1.0 m to 1.6 m. The pressure at the smaller end is 9.5 kPa. Find the pressure at the downstream end of the diffuser, assuming frictionless flow. Assume also, that the angle of the cone is small enough that the flow does not separate from the walls of the diffuser.

5.4 In Fig. P5.4, ABC is part of a piping system. Water at 50°F flows up AB (15 ft long, 1.5 in diameter), then along BC (10 ft long, 1.0 in diameter). The measured pressure and mean velocity at A are 36.3 psi and 4 ft/sec. (a) Find the pressure at C, neglecting pipe friction and energy losses. (b) Repeat for flow in the opposite direction.

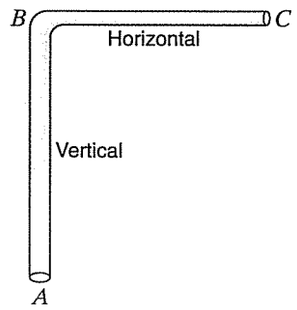


Figure P5.4

5.5 Refer to Fig. P5.4. Water at 10°C flows up pipe AB (5 m long, 40 mm diameter), then along BC (3 m long, 30 mm diameter). The measured pressure at A is 275 kPa. (a) Find the pressure at C if the flow rate is 2.0 L/s. Neglect pipe friction and energy losses. (b) Repeat for the same flow rate in the opposite direction.

5.6 Part of a vertical piping system consists of 8 ft of 4 in diameter, connected to 8 ft of 3 in diameter above, connected to 8 ft of 2 in diameter above that. Water at 60°F flows up the pipe (no down flow in permitted). (a) Neglecting friction, find the difference in water pressure (psi) between the two ends when the flow rate is 150 gpm. (b) What is the minimum possible value for this pressure difference, and under what circumstances does it occur?

5.7 In Fig. 4.12 the velocity of the undisturbed field is 22 fps and the velocities very near the surface at radii from the "source" making angles with the axis of 0, 60, 120, and 150° are 0, 17.5, 23.7, and 21.9 fps, respectively. What will be the elevation of the liquid surface relative to that of the free surface of the undisturbed field? (This problem illustrates the way in which the water surface drops alongside a bridge pier or past the side of a moving ship.)

5.8 In Fig. 4.12 the velocity of the undisturbed field is 6 m/s and the velocities very near the surface at radii from the "source" making angles with the axis of 0, 60, 120, and 150° are 0, 4.8, 6.5, and 6.0 m/s, respectively. What will be the elevation of the liquid surface relative to that of the free surface of the undisturbed field? (This problem illustrates the way in which the water surface drops alongside a bridge pier or past the side of a moving ship.)

5.9 If the body shown in Fig. 4.12 is not two-dimensional but is a solid of revolution about a horizontal axis, the flow will be three-dimensional and the streamlines will be differently spaced. Also, the distance between the stagnation point and the "source" will be $d/4$, where d is the diameter at a great distance from the stagnation point. At points very near the surface at radii from the source making angles with the axis of 0, 60, 120, and 150°, the velocities are 0, 14.0, 21.3, and 19.8 fps, respectively, when the velocity of the undisturbed field is 19 fps. If the body is a blimp and the atmospheric pressure in the undisturbed field is 14 psia, what will be the pressures at these points, for an air temperature of 53.9°F?

5.10 In Prob. 5.9 assume the body is a submarine with diameters at the four points of 0, 8.24, 14.28, and 15.90 ft, respectively. If the submarine is submerged in the ocean ($\gamma = 64.1$ lb/ft^3) with its axis 50 ft below the surface, find the pressures in pounds per square inch at these points along the top and along the bottom.

5.11 Refer to Fig. P5.4. ABC is part of a piping system. Water at 50°F flows up AB (15 ft long, 1.5 in diameter), then along BC (10 ft long, 1.0 in diameter). The measured pressure and mean velocity at A are 36.3 psi and 4 ft/sec; at C the pressure is 27.4 psi. Find the pipe friction head loss between A and C. Neglect energy losses caused by the diameter change and bend at B.

5.12 Refer to Fig. P5.4. Water at 10°C flows up pipe AB (5 m long, 40 mm diameter) and along BC (3 m long, 30 mm diameter) at 1.75 L/s. If the measured pressure at A is 250 kPa, and the pipe friction head loss between A and C is 1.45 m, find the pressure at C. Neglect energy losses caused by the diameter change and bend at B.

5.13 Water at 60°F flows at 3 cfs through a 150-ft-long duct of 6 in × 9 in cross section. The pressure at the inlet end is 15 psig, and at the outlet, 20 ft higher than the inlet, it is 4 psig. Find (a) the wall friction head loss, and (b) the friction force on the duct. Neglect energy losses caused by bends.

5.14 Water at 15°C flows up a 24-m-long conical pipe with its centerline sloping at 3° to the horizontal. At its lower end the diameter is 600 mm, the water pressure is 94.6 kPa, and the velocity is 1.3 m/s; at its upper, outlet end the diameter is 450 mm and the water pressure is 78.4 kPa. Find the shear stress at the wall, assuming it to be nonvarying. (*Hint:* You may use the mean diameter to find the pipe friction head loss.)

5.15 Water at 70°F flows up a 50-ft-long conical pipe with its centerline sloping at 5° to the horizontal. At its lower end the diameter is 24 in and the water pressure is 15 psi; at its upper, outlet end the diameter is 18 in and the water pressure is 12.5 psi. By the methods of Sec. 8.5, τ_0 has been calculated to be 0.25 lb/ft^2. Assuming this value to be nonvarying, calculate the flow rate. (*Hint:* You may use the mean diameter to find the pipe friction head loss.)

5.16 Find the stagnation pressure on a tree trunk at an elevation of 1000 m if the wind speed is 25 m/s.

5.17 Wind blows at a velocity of 20 m/s against the side of a pole at an elevation of 2000 m above sea level. What is the stagnation pressure assuming standard atmospheric conditions? Express your answer as a gage pressure and as an absolute pressure in kN/m^2, Pa, and mmHg.

5.18 In Fig. P5.18 water is admitted at the center at a rate of 2 cfs and is discharged into the air around the periphery. The upper circular plate in the figure is horizontal and is fixed in position, while the lower annular plate is free to move vertically and is not supported by the pipe in the center. The annular plate weighs 6 lb, and the weight of the water on it should be considered. (a) If the distance d between the two plates is to be maintained at 1.5 in, what is the total weight W that can be supported? (b) What is the pressure head where the radius is 4 in, and what is it at a radius of 8 in?

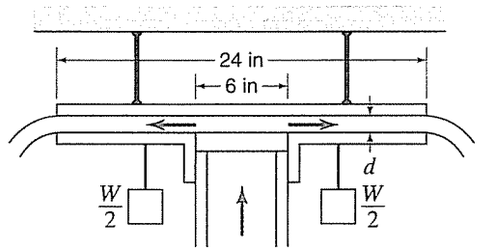


Figure P5.18

5.19 Plot the stagnation pressure (psia) on an object as it passes through air at sea level (standard atmosphere) as a function of velocity. Repeat for movement through air at 10,000 ft elevation. Let V vary from zero to c using 0, 25, 50, 75, and 100% of c.

5.20 Water is flowing at 12 m^3/s through a long pipe. The temperature of the water rises 0.18°C when heat is transferred to the water at the rate of 4500 kJ/s. Find the head loss in the pipe.

5.21 A pipeline supplies water to a hydroelectric plant from a reservoir in which the water temperature is 61.3°F. (a) Suppose that in the length of the pipe there is a total loss of heat to the surrounding air of 0.26 Btu/lb of water and the temperature of the water at the powerhouse is 61.2°F. What is the head loss per pound of water? (b) With the same flow as in (a), what will be the temperature of the water at the powerhouse if the water absorbs heat from hot sunshine at the rate of 2.9 Btu/lb of water?

5.22 A pump lifting water at 3.5 cfs adds 35 ft·lb/lb to the flow. The suction line diameter is 8 in, and at intake (elevation 350 ft) the water pressure is 5.2 psi. The discharge line diameter is 6 in, and at outlet (elevation 370 ft) the water pressure is 3.5 psi. Due to cold weather, 7 ft·lb/lb of thermal energy (heat) are lost to the environment. Find the change (rise or fall?) in water temperature between intake and outlet. Assume the specific weight of the water remains constant at 62.4 lb/ft^3.

5.23 A pump lifting water at 0.08 m^3/s adds 12 N·m/N to the flow. The suction line diameter is 200 mm, and at intake (elevation 50 m) the water pressure is 72 kPa. The discharge line diameter is 150 mm, and at outlet (elevation 56 m) the water pressure is 60 kPa. Sunshine striking the pipes adds 5 J/N of heat to the water. Find the change (rise or fall?) in water temperature between intake and outlet. Assume the specific weight of the water remains constant at 9.81 kN/m^3.

5.24 A pump, with an efficiency of 90%, circulates water at the rate of 2500 gpm in a closed circuit that holds 8500 gal. The net head developed by the pump is 360 ft. What is the change in water temperature after one hour, assuming that the bearing friction is negligible and that there is no heat loss from the system?

5.25 A pump, with an efficiency of 92%, circulates water at the rate of 130 L/s in a closed circuit that holds 45 m^3. The net head developed by the pump is 120 m. What is the change in water temperature after one hour, assuming that the bearing friction is negligible and that there is no heat loss from the system?

5.26 In Fig. P5.26, assume water is flowing and neglect all head losses except at discharge. Find the flow rate if $h = 8$ ft. Assuming that $d = 10$ ft, the throat diameter is two-thirds the pipe diameter where it joins the downstream tank, and the atmospheric pressure is equal to the standard atmospheric pressure at 10,000 ft elevation, calculate the gage pressure and the absolute pressure in the constriction. The throat diameter is 14 in.

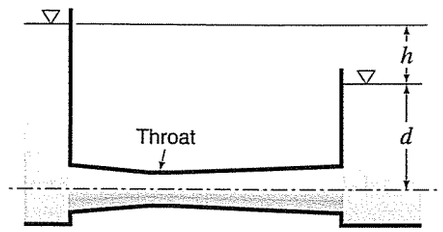


Figure P5.26

5.27 Referring to Fig. P5.26, assume water is flowing and neglect all head losses except at discharge. Find the flow rate if $h = 1.6$ m. Assuming that $d = 4$ m, the throat diameter is two-thirds the pipe diameter where it joins the downstream tank, and the atmospheric pressure is equal to the standard atmospheric pressure at 2000 m elevation, calculate the gage pressure and the absolute pressure in the constriction. The throat diameter is 300 mm.

5.28 Repeat Prob. 5.26, assuming head losses are as follows: 6 inches in the converging section and 30 inches in the diverging section.

5.29 Repeat Prob. 5.27, assuming head losses are as follows: 0.12 m in the converging section and 0.65 m in the diverging section.

5.30 In Fig. P5.26 neglect all head losses except at discharge, and assume water is flowing. If $h = 15$ ft and $d = 12$ ft, find the highest permissible water temperature in order that there be no cavitation. The throat diameter is 80% of the pipe diameter where it joins the downstream tank. Atmospheric pressure is 13.6 psia.

5.31 In Fig. P5.26 neglect all head losses except at discharge, and assume water is flowing. If $h = 4$ m and $d = 5.5$ m, find the highest permissible water temperature in order that there be no cavitation. The throat diameter is 70% of the pipe diameter where it joins the downstream tank. Atmospheric pressure is 97 kPa abs.

5.32 Redo Prob. 5.30, but this time let the water temperature be 50°F. Find the minimum permissible throat diameter in order to not have cavitation. Express the answer as a fraction of the outlet diameter.

5.33 In Fig. P5.33 friction loss between A and B is negligible while between B and C it is $0.15(V_B^2/2g)$. Given $h = 750$ mm, $d_A = d_C = 250$ mm, $d_B = 100$ mm. Find the pressure heads at A and C if the liquid is flowing through the circular pipe from A to C at the rate of 280 L/s.

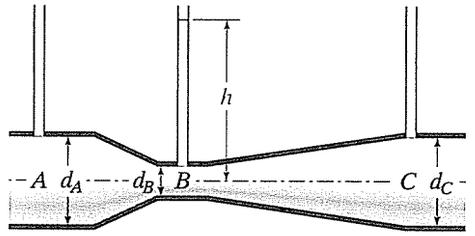


Figure P5.33

5.34 In Fig. P5.34 assume the tube flows full. At B, the diameter of the tube is 3 in and the diameter of the water jet discharging into the air at C is 4.5 in. (*a*) If all friction losses are negligible, what are the velocity and the pressure head at B if $h = 10$ ft. (*b*) What is the rate of discharge in cfs? And what would it be if the tube were cut off at B?

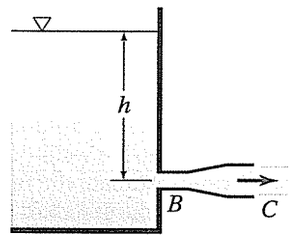


Figure P5.34

5.35 Referring to Fig. P5.34, assume the tube flows full and all friction losses are negligible. The diameter at B is 60 mm and the diameter of the jet discharging into the air is 80 mm. If $h = 5$ m, what is the flow rate? What is the pressure head at B? What would be the flow rate if the tube were cut off at B?

5.36 In Fig. P5.36 friction losses in the pipe below pump P are $1.8V^2/2g$ with the barometer pressure at 12.50 psia. The liquid in the suction pipe has a velocity of 7 fps. What would be the maximum allowable value of z if the liquid were (*a*) water at 70°F; (*b*) gasoline at a vapor pressure of 9 psia with a specific weight of 47 lb/ft^3?

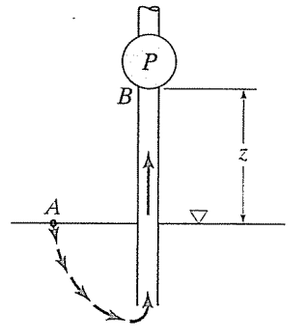


Figure P5.36

5.37 In Fig. P5.36 friction losses in the pipe below pump P are $1.6V^2/2g$ with the barometer pressure at 90 kPa. The liquid in the suction pipe has a velocity of 1.8 m/s. What would be the maximum allowable value of z if the liquid were (*a*) water at 20°C; (*b*) gasoline at a vapor pressure of 49 kPa abs, with a specific weight of 8 kN/m^3?

5.38 A discharge pressure gage reading, taken at a point of 6.5 ft above the centerline of a pump, is 25 psi. A suction pressure gage reading, taken 2.5 ft below the centerline, indicates a vacuum of 12 inHg when gasoline ($s = 0.75$) is pumped at the rate of 1.5 cfs. The diameters of the suction and discharge pipes of the pump are 8 and 6 in, respectively. What is the power delivered to the fluid? Sketch the energy line and the hydraulic grade line.

5.39 For this problem, use the same data as in Sample Prob. 5.11, except that, instead of the pump developing 80 ft of head, it delivers 110 hp to the water. Find the new flow rate. Plot the energy line and the hydraulic grade line. Calculate the pressure on the suction side of the pump.

5.40 Assume ideal fluid. The pressure at section 1 in the Fig. P5.40 is 10 psi, $V_1 = 15$ fps, $V_2 = 50$ fps, and $\gamma = 60$ lb/ft^3. (*a*) Determine the reading on the manometer. (*b*) If the downstream piezometer were replaced with a pitot tube, what would be the manometer reading? Comment on the practicality of these arrangements.

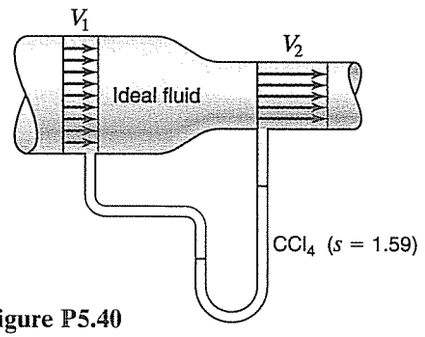


Figure P5.40

5.41 Refer to Fig. P5.40. Assume an ideal fluid with $\rho = 900$ kg/m^3. The pressure at section 1 is 100 kN/m^2, $V_1 = 10$ m/s, $V_2 = 20$ m/s. (*a*) Determine the reading on the manometer. (*b*) If the downstream piezometer were replaced with a pitot tube, what would be the manometer reading? Comment on the practicality of these arrangements.

5.42 By manipulation of Eq. (5.44), demonstrate that it represents a standard parabola of the form $z - z_0 = a(x - x_0)^2$, where a is a constant and x_0 and z_0 are the coordinates of the vertex.

5.43 Find the maximum ideal horizontal range of a jet having an initial velocity of 90 fps. At what angle of inclination is this obtained?

5.44 Repeat Exer. 5.16.1. Let $V = Q/A = 24$ fps, but assume a parabolic velocity profile.

5.45 Using Fig. X5.16.1, which depicts a two-dimensional flow in a vertical plane, find the pressure at B if the pressure at A is 32 kPa. Data are as follows: $r = 3$ m, $b = 1.2$ m, $\gamma = 9.81$ kN/m^3, $V = Q/A = 5$ m/s. Assume a parabolic velocity profile.

5.46 In Fig. P5.46 the rotor vanes are all straight and radial, $r_1 = 0.3$ ft, $r_2 = 0.9$ ft, and the height perpendicular to the plane of the figure is constant at $B = 0.25$ ft. Then $A = 2\pi rB$. If the rotation speed is 1000 rpm and the flow of liquid is 9.6 cfs, find the difference in the pressure head between the outer and the inner circumferences, neglecting friction losses. Does it make any difference whether the flow is outward or inward?

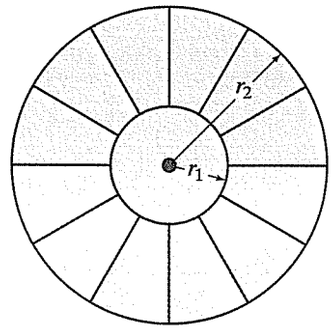


Figure P5.46

5.47 In Fig. P5.46 the vanes are all straight and radial, $r_1 = 10$ cm, $r_2 = 20$ cm, and the height perpendicular to the plane of the figure is constant at $B = 80$ mm. Then $A = 2\pi rB$. If the rotation speed is 1000 rpm and the flow of liquid is 0.3 m^3/s, find the difference in the pressure head between the outer and the inner circumferences, neglecting friction losses. Does it make any difference whether the flow is outward or inward?

5.48 An air duct of 2.5 ft by 2.5 ft square cross section turns a bend of radius 5 ft as measured to the centerline of the duct. If the measured pressure difference between the inside and outside walls of the bend is 1.5 in of water, estimate the rate of air flow in the duct. Assume standard sea-level conditions in the duct and assume ideal flow around the bend.

5.49 An air duct of 1.2 m by 1.2 m square cross section turns a bend of radius 2.4 m as measured to the centerline of the duct. If the measured pressure difference between the inside and outside walls of the bend is 50 mm of water, estimate the rate of air flow in the duct. Assume standard sea-level conditions in the duct and assume ideal flow around the bend.

CHAPTER 6
Momentum and Forces in Fluid Flow

Previously, we met two important fundamental concepts of fluid mechanics: the continuity equations and the energy equation. In this chapter we will develop a third basic concept, the ***momentum principle.*** This concept is particularly important in flow problems where we need to determine forces. Such forces occur whenever the velocity of a stream of fluid changes in either direction or magnitude. By the law of action and reaction, the fluid exerts an equal and opposite force on the body producing the change. After developing the momentum principle, we will discuss its application to a number of important engineering problems.

The last third of this chapter, Secs. 6.11–6.15, addresses the application of the momentum principle to rotating machines like fans, propellers, windmills, sprinklers, pumps, and turbines. Such rotating machines are common and important, and therefore the applications are also important. While these sections provide a basic introduction to Chaps. 15 and 16 on hydraulic machinery, they can, however, be omitted with little detrimental impact on Chaps. 7–14.

6.1 DEVELOPMENT OF THE MOMENTUM PRINCIPLE

We will derive the momentum principle from Newton's second law. The flow may be compressible or incompressible, real (with friction) or ideal (frictionless), steady or unsteady, and the equation is not limited to flow along a streamline. In Chap. 5 when applying the energy equation to real fluids we found that the energy loss must be computed. We do not encounter this difficulty in momentum analysis.

We can express Newton's second law as

$$\sum \mathbf{F} = \frac{d(m\mathbf{V})_S}{dt} \tag{6.1}$$

This states that the sum of the external forces $\mathbf{F}$ on a body of fluid or system S is equal to the rate of change of linear momentum $m\mathbf{V}$ of that body or system. The boldface symbols $\mathbf{F}$ and $\mathbf{V}$ represent vectors, and so the change in momentum must be in the same direction as the force. Because we can also express Eq. (6.1)

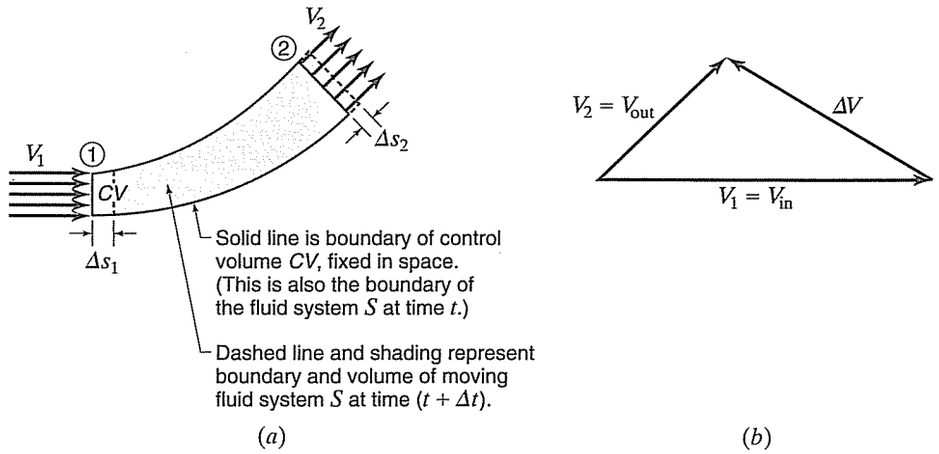


Figure 6.1
(*a*) Control volume for steady flow with control surface cutting a constant-velocity stream at right angles. (*b*) Velocity relations.

as $\sum(\mathbf{F})\,dt = d(m\mathbf{V})_S$, i.e., impulse equals change of momentum, we sometimes use the terminology ***impulse-momentum principle.***

Using the principles of Sec. 4.6, let us consider the linear momentum of the ***fluid system*** and ***control volume*** defined within the stream tube of Fig. 6.1*a*, just as we did for energy in Sec. 5.5. The fixed control volume (*CV*) lies between sections 1 and 2, and the moving fluid system (*S*) consists of the fluid mass contained at time t in the control volume. During a short time interval Δt, we shall assume that the fluid moves a short distance Δs_1 at section 1 and Δs_2 at section 2. Recalling the analysis of Sec. 4.6, and letting the general property X now be the momentum $m\mathbf{V}$, Eq. (4.9) becomes

$$\frac{d(m\mathbf{V})_S}{dt} = \frac{d(m\mathbf{V})_{CV}}{dt} + \frac{d(m\mathbf{V})^{\text{out}}_{CV}}{dt} - \frac{d(m\mathbf{V})^{\text{in}}_{CV}}{dt} \tag{6.2}$$

where, as before, subscript S denotes the moving fluid system and subscript CV denotes the fixed control volume. So, setting this equal to Eq. (6.1),

$$\text{Unsteady flow:} \qquad \sum\mathbf{F} = \frac{d(m\mathbf{V})_{CV}}{dt} + \frac{d(m\mathbf{V})^{\text{out}}_{CV}}{dt} - \frac{d(\text{m}\mathbf{V})^{\text{in}}_{CV}}{dt} \tag{6.3}$$

On the right side of this equation, the first term represents the rate of change or accumulation of momentum within the fixed control volume, whereas the second and third terms respectively represent the rates at which momentum enters and leaves the control volume. The entire Eq. (6.3) states that the resultant force acting on a fluid mass is equal to the rate of change of momentum of the fluid mass. It is perfectly general. It applies to compressible or incompressible, real or ideal, and steady or unsteady flow.

In the case of steady flow, conditions within the control volume do not change, so $d(m\mathbf{V})_{CV}/dt = 0$, and the equation becomes

$$\text{Steady flow:} \qquad \sum\mathbf{F} = \frac{d(m\mathbf{V})^{\text{out}}_{CV}}{dt} - \frac{d(m\mathbf{V})^{\text{in}}_{CV}}{dt} \tag{6.4}$$

Thus, for steady flow the net force on the fluid mass is equal to the net rate of *outflow* of momentum across the control surface.

Since Eqs. (6.1)–(6.4) are vectorial equations, we can also express them as scalar equations in terms of forces and velocities in the x, y, and z directions, respectively.

It helps if we select a control volume so that the control surface is normal to the velocity where it cuts the flow. Consider such a situation in Fig. 6.1*a*. Also, let us specify that the velocity is constant where it cuts across the control surface, and let us restrict ourselves to steady flow so that Eq. (6.4) is applicable. Since

$$\frac{d(m\mathbf{V})_1}{dt} = \frac{dm_1}{dt}\mathbf{V}_1 = \dot{m}_1\mathbf{V}_1 = \rho_1 Q_1\mathbf{V}_1$$

and the same relations hold for section 2, we can write Eq. (6.4) as

$$\text{Steady flow:} \qquad \sum\mathbf{F} = \dot{m}_2\mathbf{V}_2 - \dot{m}_1\mathbf{V}_1 = \rho_2 Q_2\mathbf{V}_2 - \rho_1 Q_1\mathbf{V}_1 \tag{6.5}$$

But since the flow we are considering is steady, from continuity, $\dot{m}_1 = \dot{m}_2 = \dot{m} = \rho_1 Q_1 = \rho_2 Q_2 = \rho Q$. Also, using the vector relations of Fig. 6.1*b*, let us for convenience write $\Delta\mathbf{V} = \mathbf{V}_2 - \mathbf{V}_1 = \mathbf{V}_{\text{out}} - \mathbf{V}_{\text{in}}$. Using these, Eq. (6.5) becomes

$$\text{Steady flow:} \qquad \sum\mathbf{F} = \dot{m}(\Delta\mathbf{V}) = \rho Q(\Delta\mathbf{V}) = \rho Q(\mathbf{V}_2 - \mathbf{V}_1) \tag{6.6}$$

The direction of $\sum\mathbf{F}$ must be the same as that of the velocity change, $\Delta\mathbf{V}$. Note that the $\sum\mathbf{F}$ represents the vectorial summation of *all* forces acting *on the fluid mass* in the control volume, including gravity forces, shear forces, and pressure forces including those exerted by fluid surrounding the fluid mass under consideration as well as the pressure forces exerted by the solid boundaries in contact with the fluid mass. Often the force sought is just *one* of these many forces. Frequently it is not even one of them, but instead it is *opposite* to one of them, being the force of the liquid acting on a boundary. The right side of Eq. (6.6) represents the change in momentum per unit time.

Since Eq. (6.6) is vectorial, we can express it by the following scalar (component) equations:

$$\text{Steady flow:} \qquad \sum F_x = \dot{m}(\Delta V_x) = \rho Q(\Delta V_x) = \rho Q(V_{2x} - V_{1x}) \tag{6.7a}$$

$$\sum F_y = \dot{m}(\Delta V_y) = \rho Q(\Delta V_y) = \rho Q(V_{2y} - V_{1y}) \tag{6.7b}$$

$$\sum F_z = \dot{m}(\Delta V_z) = \rho Q(\Delta V_z) = \rho Q(V_{2z} - V_{1z}) \tag{6.7c}$$

In Sec. 6.4 and succeeding sections we will apply these equations to several situations that are commonly encountered in engineering practice. If the flow in a

single stream tube splits up into several stream tubes, we just compute the ρQV values of each stream tube separately and then substitute them into Eqs. (6.5)–(6.7) (see Sample Prob. 6.2). The great advantage of the momentum principle is that we need not know the details of what is occurring within the flow; only the conditions at the end sections of the control volume govern the analysis.

6.2 NAVIER-STOKES EQUATIONS

We can derive a set of differential equations that describe the motion of a real fluid for the general case by considering the forces acting on a small element or control volume of fluid like Fig. 3.2. The forces include gravitational, viscous (frictional), and pressure forces. Before we can incorporate Newton's equation of viscosity (Eq. 2.9) for one-dimensional flow we must generalize it to three-dimensional flow.

The full derivation of these equations is lengthy and involved, and beyond the scope of this text. However, for an incompressible fluid with constant viscosity, in rectangular coordinates with z increasing vertically upwards, the result is

$$-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)=\rho\left[\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right] \quad (6.8a)$$

$$-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)=\rho\left[\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right] \quad (6.8b)$$

$$-\rho g-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)=\rho\left[\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right] \quad (6.8c)$$

These fundamental general equations of motion are known as the ***Navier-Stokes equations.*** We name them after the French scientist, C. L. M. H. Navier (1785–1836), who today we would describe as a civil engineer, and the English physicist, Sir George Stokes (1819–1903), both of whom first derived them. They are second-order nonlinear partial differential equations (Appendix B) that no one has analytically solved in general, although they have obtained analytical and numerical solutions for certain specific situations. Their complete derivations, for rectangular, cylindrical, and spherical coordinates, are given in advanced fluid mechanics texts.

The Navier-Stokes equations are in fact just a differential form of the linear momentum principle. Thus, on the left sides of Eqs. (6.8), we have the body force per unit volume (the term in g) and the surface force per unit volume (pressure force represented by terms in p, and viscous force represented by terms with μ and parentheses). These are equal to the time rate of change of momentum on the right side, consisting in the square brackets of the local acceleration (the derivatives with respect to time) and the convective acceleration (the other terms).

When we write Eqs. (6.8) in terms of the normal stresses σ and the shear stresses τ, we call them the ***Cauchy equations.*** For an ideal fluid ($\mu = 0$), they reduce to a set of three-dimensional equations known as the ***Euler equations*** of motion, which are the same as Eqs. (6.8) but with the terms containing μ and second-order derivatives eliminated.

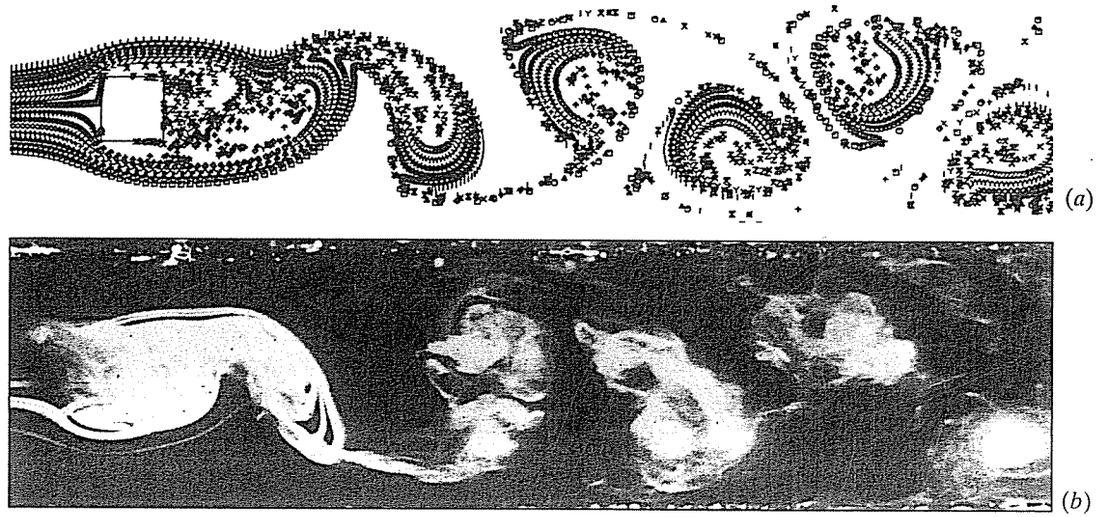


Figure 6.2
Comparison of flow around a square block. (*a*) Computed at $\mathbf{R} = 500$. (*b*) Visualization at $\mathbf{R} = 550$. (Courtesy Ronald W. Davis, Chemical Science & Technology Laboratory, NIST)

A major area of endeavor has used the Navier-Stokes equations, together with various numerical computation methods, in efforts to solve flow fields with challenging features. The entire flow region is divided into many small elements, and the equations applied to each element. We call such procedures ***computational fluid dynamics*** (CFD). In Fig. 6.2 we can compare the excellent results of such a numerical computation with a photograph of an actual flow field.

6.3 MOMENTUM CORRECTION FACTOR

If the velocity is not uniform over a section (e.g., Secs. 2.11, 4.1, 5.11), we shall find that the momentum per unit time crossing that section is greater than that computed by using the mean velocity. The rate of momentum transfer (***momentum flux***) across an elementary area dA, where the local velocity is u, is $\dot{m}u = (\rho u\,dA)u = \rho u^2\,dA$, and the rate of momentum transfer across the entire section is $\rho\int_A u^2\,dA$, while that computed by using the mean velocity is $\rho QV = \rho AV^2$. Thus the ***momentum correction factor*** β (beta), which we should multiply ρQV by to obtain the true momentum per unit time, is

$$\beta=\frac{1}{AV^2}\int_A u^2\,dA \quad (6.9)$$

For laminar flow in a circular pipe, $\beta = \frac{4}{3}$, but for turbulent flow in circular pipes, it usually ranges from 1.005 to 1.05, as we can see from Eq. (8.45*b*). For

open-channel flow, it may be greater. Mathematically, it cannot be less than 1.0, and, unless otherwise specified, we will take the value of β in the following discussion as 1.0.

EXERCISES

6.3.1 For laminar flow as in Sample Prob. 5.1, find β.

6.3.2 For the turbulent-flow case as approximated in Exer. 5.1.1, find β.

6.4 APPLICATIONS OF THE MOMENTUM PRINCIPLE

In a common application of the momentum principle, we use it to find forces that flowing fluid exert on structures open to the atmosphere, like gates (e.g., Fig. 11.34*a*) and overflow spillways (e.g., Fig. 11.32).

It is very important to remember that the momentum principle deals only with forces that act *on* the fluid mass in a *designated control volume* (*CV*); fluid forces acting on a structure are equal and opposite to boundary pressure forces acting on the fluid. To avoid confusion over signs, we strongly advise students to first solve for the magnitude and direction of the (reaction) force of the structure on the fluid, and only in the last step to find the equal and opposite force of the fluid on the structure. (We will use additional subscripts to designate forces *not* acting on the control volume.)

The force of a fluid on a structure is usually distributed as varying pressure forces over the surface. We are normally interested in the resultant of this distribution, and we usually only consider pressures that differ from atmospheric. If we need the total resultant force, we usually obtain this by writing equations like (6.7) in order to first find the perpendicular components of the required (opposite) force.

First, we must establish a control volume (*CV*), and, as noted in Sec. 6.1, we should do this by cutting the flow normal to the velocity, along a boundary where the velocity is constant. We can best further discuss applications to this type of problem with a sample problem.

SAMPLE PROBLEM 6.1 The water passage in Fig. S6.1 is 10 ft (3 m) wide normal to the plane of the figure. Determine the horizontal force acting on the shaded structure. Assume ideal flow.

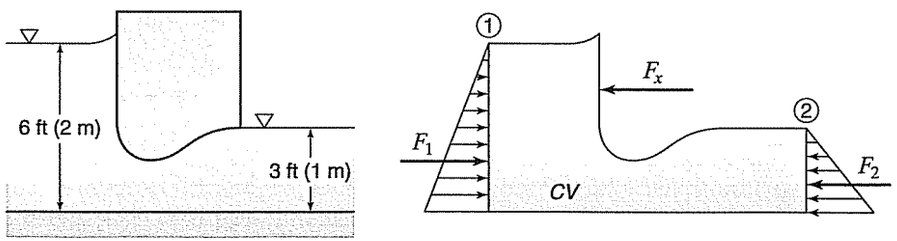


Figure S6.1

Solution (BG units)

In free-surface flow such as this where the streamlines are parallel, the water surface is coincident with the hydraulic grade line. Writing an energy equation from the upstream section to the downstream section, where $V = V_x$,

$$6 + \frac{V_1^2}{2g} = 3 + \frac{V_2^2}{2g} \tag{1}$$

From continuity, $$6(10)V_1 = 3(10)V_2 \tag{2}$$

Substituting Eq. (2) into Eq. (1) yields

$$V_1 = 8.02 \text{ fps}, \quad V_2 = 16.05 \text{ fps}$$

$$Q = A_1V_1 = A_2V_2 = 481 \text{ cfs}$$

Next take a free-body diagram of the control volume (*CV*) of water shown in the figure and apply the momentum equation (6.7*a*),

$$F_1 - F_2 - F_x = \rho Q(V_2 - V_1)$$

where F_x represents the force of the structure on the water (*CV*) in the horizontal direction, and the F's and V's are understood to have no y components.

From Eq. (3.16), we have $F_1 = \gamma h_{c1}A_1$ and $F_2 = \gamma h_{c2}A_2$. Hence

$$62.4(3)(10 \times 6) - 62.4(1.5)(10 \times 3) - F_x = 1.94(481)(16.05 - 8.02)$$

and $$F_x = +936 \text{ lb} = 936 \text{ lb} \leftarrow$$

The positive sign means that the assumed direction is correct. The force of the water on the structure is equal and opposite, namely,

$$(F_{W/S})_x = 936 \text{ lb} \rightarrow \qquad \textit{ANS}$$

Note that the momentum principle will not permit us to obtain the vertical component of the force of the water on the shaded structure, because the pressure distribution along the bottom of the channel is unknown. We can estimate the pressure distribution along the boundary of the structure and along the bottom of the channel by sketching a flow net and applying Bernoulli's principle. Then we can find the horizontal and vertical components of the force by computing the integrated effect of the pressure-distribution diagram.

Solution (SI units)

Energy: $$2 + \frac{V_1^2}{2(9.81)} = 1 + \frac{V_2^2}{2(9.81)} \tag{3}$$

Continuity: $$2(3)V_1 = 1(3)V_2 \tag{4}$$

Substituting Eq. (4) into Eq. (3) yields

$$V_1 = 2.56 \text{ m/s}, \quad V_2 = 5.11 \text{ m/s}$$

$$Q = A_1V_1 = A_2V_2 = 15.34 \text{ m}^2\text{/s}$$

Applying momentum equation (6.7*a*) to the free-body diagram,

$$F_1 - F_2 - F_x = \rho Q(V_2 - V_1)$$

$$9.81(1)(2)(3) - 9.81(0.5)(1)(3) - F_x = 1.0(15.34)(5.11 - 2.56)$$

$$F_x = +4.91 \text{ kN} = 4.91 \text{ kN} \leftarrow$$

So $(F_{W/S})_x = 4.91 \text{ kN} \rightarrow$ ***ANS***

There are numerous other fluid-flow situations where the momentum principle is useful. In the Secs. 6.5–6.15 we shall apply it to find forces exerted on pressure conduits such as at bends and nozzles, to forces exerted by jets on stationary and moving vanes or blades, and subsequently to rotating machines like pumps, turbines, propellers, and windmills. We will also use the momentum principle to develop an expression for the head loss in a pipe expansion (Sec. 8.24), for the conjugate depths of a hydraulic jump (Sec. 10.18), and in the development of the relationships in a shock wave (Sec. 13.10).

EXERCISES

6.4.1 A cylindrical drum of radius 2.2 ft is securely held in position in an open channel of rectangular section. The channel is 10 ft wide, and the flow rate is 200 cfs. Water flows beneath the drum as shown in Fig. X6.4.1. Determine the horizontal thrust on the cylinder using the momentum principle. Neglect fluid friction.

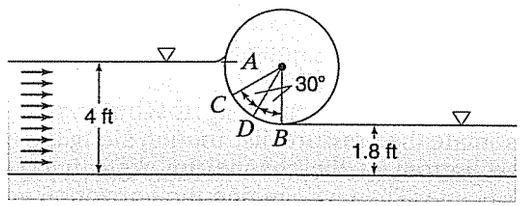


Figure X6.4.1

6.4.2 Find the horizontal thrust of the water on each meter of width of the sluice gate shown in Fig. X6.4.2, given $y_1 = 2.2$ m, $y_2 = 0.4$ m, and $y_3 = 0.5$ m. Neglect friction.

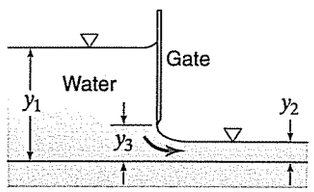


Figure X6.4.2

6.4.3 Refer to Fig. X6.4.2. Find the horizontal thrust of the water on each foot of width of the sluice gate, given $y_1 = 7$ ft, $y_2 = 1.2$ ft, and $y_3 = 1.4$ ft. Neglect friction.

6.4.4 Flow occurs over a spillway of constant section as shown in Fig. X6.4.4. Assuming ideal flow, determine the resultant horizontal force on the spillway per foot of spillway width (perpendicular to the spillway section), given that $y_1 = 4.2$ ft and $y_2 = 0.7$ ft.

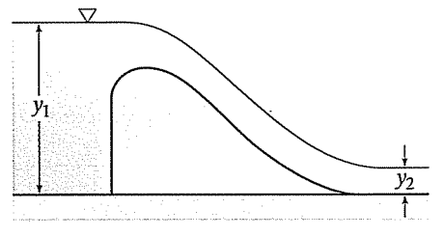


Figure X6.4.4

6.4.5 Flow occurs over the spillway of constant section as shown in Fig. X6.4.4. Given that $y_1 = 4.2$ m and $y_2 = 0.7$ m, determine the resultant horizontal force on the spillway per meter of spillway width (perpendicular to the spillway section). Assume ideal flow.

6.5 FORCE ON PRESSURE CONDUITS

Consider the case of horizontal flow to the right through the reducer of Fig. 6.3*a*. A free-body diagram of the forces acting on the fluid mass contained in the reducer (the control volume, *CV*) is shown in Fig. 6.3*b*. We shall apply Eq. (6.7*a*) to this fluid mass to examine the forces that are acting in the x direction. The forces p_1A_1 and p_2A_2 represent pressure forces that fluid located just upstream and just downstream exerts on the control volume. The force F_x represents the force exerted *by the reducer on the fluid* (*CV*) in the x direction. Neglecting shear forces at the boundary of the reducer, the force F_x is the resultant (integrated) effect of the normal pressure forces that the wall of the reducer exerts on the fluid. The intensity of pressure at the wall will decrease as the diameter decreases because of the increase in velocity head, in accordance with Bernoulli's theorem (Eq. 5.30). Figure 6.4 is a typical pressure diagram.

The effect that atmospheric pressure has on such analyses can be confusing. The pressures shown in Fig. 6.4 are gage pressures (Sec. 3.4). If we used absolute pressures, we would have to increase all the pressures shown by a constant amount, p_{atm} (about 14.70 psi or 101.3 kPa at sea level). This would

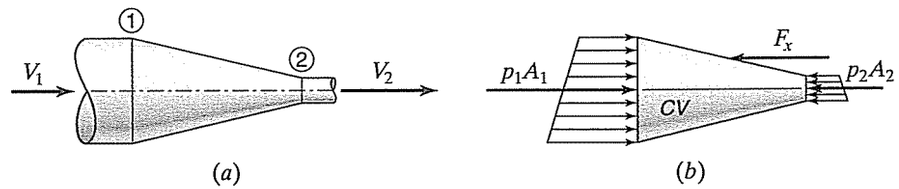


Figure 6.3

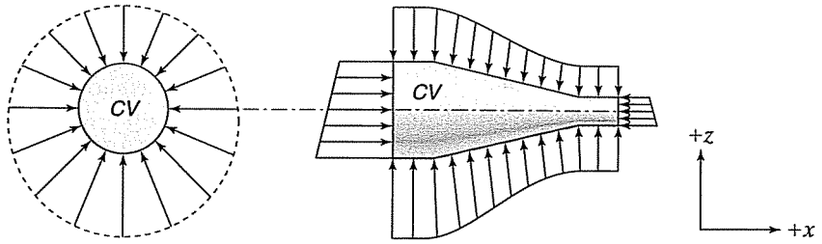


Figure 6.4
Gage pressure distribution on the fluid in a reducer.

increase $p_1A_1 - p_2A_2$, F, and the equal and opposite force exerted *by the fluid on the reducer,* $F_{F/R}$. However, this increased force on the inside of the reducer would exactly balance the force of the atmosphere on the outside. Therefore the atmospheric pressure does not affect the *net* force on the reducer, which results from the fluid flow and which tends to move the reducer. It is this *net* force which interests us, and we can most easily obtain it by excluding atmospheric pressure, i.e., by using gage pressures. Therefore, customarily we use *gage* pressures for p_1 and p_2.

Applying Eq. (6.7*a*) and assuming the fluid is ideal with F_x directed as shown, since the entry and exit velocities are parallel to the x direction, we get

$$\sum F_x = p_1A_1 - p_2A_2 - F_x = \rho Q(V_2 - V_1) \tag{6.10}$$

In Eq. (6.10) each term can be evaluated independently from the given flow data, except F_x, which is the quantity we wish to find. Rewriting Eq. (6.10), the result is

$$F_x = p_1A_1 - p_2A_2 - \rho Q(V_2 - V_1) \tag{6.11}$$

This gives the net force of the *reducer on the fluid* (the *CV*) in the x direction. This force acts to the left as assumed in Fig. 6.3*b* and as applied in Eq. (6.10). The force of the *fluid on the reducer* ($F_{F/R}$) is, of course, equal and opposite to that of the reducer on the fluid. If the flow were to the left in Fig. 6.3, a similar analysis would apply, but we need to be consistent in regard to plus and minus signs. Conventionally we usually take the flow direction as the positive direction.

By considering the weight of fluid between sections 1 and 2 in Fig. 6.3*a* we must conclude that pressures are larger on the bottom half of the pipe than on the upper half. Recall (Sec. 6.1) that it is the conditions at the end sections of the control volume that govern the analysis. How flow moves between sections 1 and 2 is unimportant to the determination of forces. Figure 6.4 gives a schematic representation of the gage pressure distribution on the fluid within the reducer. The integrated effect of the pressures exerted by the reducer itself is equivalent in the x direction to F_x and in the z direction to the weight of fluid between sections 1 and 2.

If the fluid undergoes a change in both direction and velocity, as in the reducing pipe bend in Fig. 6.5, the procedure is similar to that of the preceding

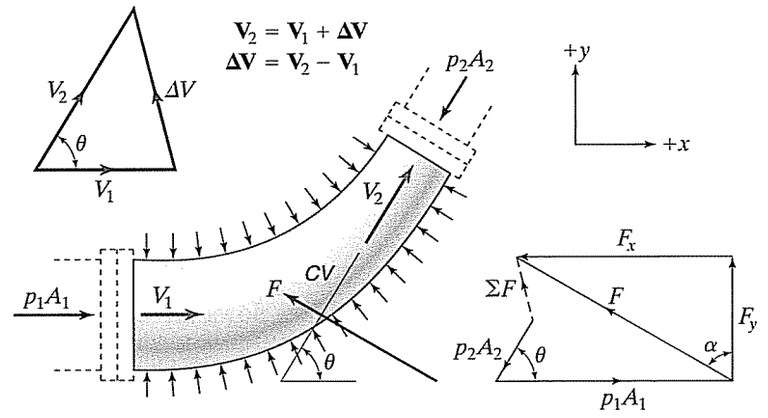


Figure 6.5
Forces on the fluid in a reducing bend. ($\Sigma\mathbf{F}$ is parallel to $\Delta\mathbf{V}$.)

case, except that we find it convenient to deal with components. Assuming the flow is in a horizontal plane so that we can neglect the weight, applying Eq. (6.7*a*) by summing up x-forces acting on the fluid in the *CV*, and equating them to the change in fluid momentum in the x direction, gives

$$\sum F_x = p_1A_1 - p_2A_2\cos\theta - F_x = \rho Q(V_{2x} - V_{1x}) \tag{6.12}$$

which, after noting that $V_{2x} = V_2\cos\theta$ and $V_{1x} = V_1$, when rewritten for the force we wish to find, becomes

$$F_x = p_1A_1 - p_2A_2\cos\theta - \rho Q(V_2\cos\theta - V_1) \tag{6.13}$$

Similarly, in the y direction,

$$\sum F_y = 0 - p_2A_2\sin\theta + F_y = \rho Q(V_{2y} - V_{1y}) \tag{6.14}$$

which, after noting that $V_{2y} = V_2\sin\theta$ and $V_{1y} = 0$, when rewritten, becomes

$$F_y = p_2A_2\sin\theta + \rho QV_2\sin\theta \tag{6.15}$$

In a specific case, if the numerical values of F_x and F_y determined from these equations are positive then the assumed directions are correct. A negative value for either one merely indicates that that component is in the direction opposite to that assumed.

Note that $\sum\mathbf{F} = \rho Q\,\Delta\mathbf{V}$ is the resultant of *all* the forces acting on the fluid in the control volume, which *includes* the pressure forces on the two ends and the force $\mathbf{F}$ exerted by the bend on the fluid. The directions of $\sum\mathbf{F}$ and $\Delta\mathbf{V}$ must be the same (see Fig. 6.5). The value of F is $\sqrt{F_x^2 + F_y^2}$, and we can obtain its direction α from the force diagram shown in Fig. 6.5.

The total force $\mathbf{F}_{F/B}$ exerted by the fluid on the bend is equal in magnitude but opposite in direction to the force $\mathbf{F}$ of the bend on the fluid. The force of the fluid on the bend tends to move the portion of the pipe under consideration. Hence, to prevent damage where such changes in velocity or alignment occur, a large pipe will usually be "anchored" by attaching it to a concrete block of sufficient size and/or weight to provide the necessary resistance.

If the flow in Fig. 6.5 had been in a vertical plane, i.e., y was vertical, we would have to calculate the weight of the fluid between sections 1 and 2 and include it in Eqs. (6.14) and (6.15). Also, we could include the effects of shear stresses due to fluid friction in the problem; however, these effects are usually small. If there are multiple inlets or exits, the principle remains the same: $\Sigma\mathbf{F} = \Sigma(\rho Q\mathbf{V})_{out} - \Sigma(\rho Q\mathbf{V})_{in}$; this is illustrated in Sample Prob. 6.2.

SAMPLE PROBLEM 6.2 Water flows through the double nozzle as shown in Fig. S6.2. Determine the magnitude and direction of the resultant force the water exerts on the nozzle. The velocity of both nozzle jets is 12 m/s. The axes of the pipe and both nozzles lie in a horizontal plane. $\gamma = 9.81$ kN/m^3. Neglect friction.

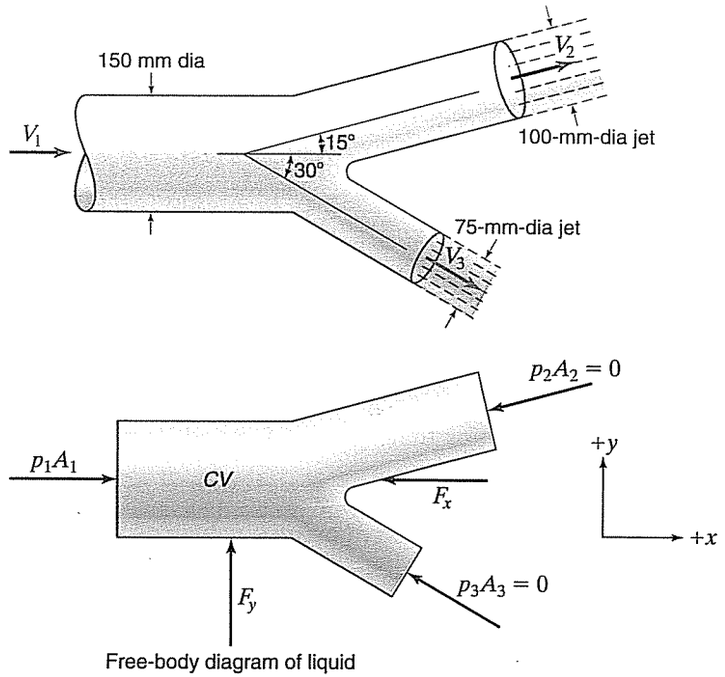


Figure S6.2

Solution

Continuity: $$A_1V_1 = A_2V_2 + A_3V_3$$

$$15^2V_1 = 10^2(12) + 7.5^2(12), \quad V_1 = 8.33 \text{ m/s}$$

$$Q_1 = \frac{\pi}{4}(0.15)^2 8.33 = 0.1473 \text{ m}^3\text{/s}, \quad Q_2 = 0.0942 \text{ m}^3\text{/s}, \quad Q_3 = 0.0530 \text{ m}^3\text{/s}$$

Jets 2 and 3 are "free," i.e., in the atmosphere, so $p_2 = p_3 = 0$. Writing energy equation (5.29) along a streamline:

$$\frac{p_1}{\gamma} + z + \frac{8.33^2}{2(9.81)} = 0 + z + \frac{12^2}{2(9.81)}$$

So $$\frac{p_1}{\gamma} = 3.80 \text{ m}, \quad p_1 = 37.3 \text{ kN/m}^2, \quad p_1A_1 = 0.659 \text{ kN}$$

Eq. (6.7*a*): $$\Sigma F_x = p_1A_1 - 0 - F_x = (\rho Q_2V_{2x} + \rho Q_3V_{3x}) - \rho Q_1V_{1x}$$

$$\rho = \frac{\gamma}{g} = \frac{9.81 \text{ kN/m}^3}{9.81 \text{ m/s}^2} = 1.0 \frac{\text{kN}\cdot\text{s}^2}{\text{m}^4} = 10^3 \frac{\text{kg}}{\text{m}^3}$$

$$V_{2x} = V_2 \cos 15° = 12(0.966) = 11.59 \text{ m/s}$$

$$V_{3x} = V_3 \cos 30° = 12(0.866) = 10.39 \text{ m/s}, \quad V_{1x} = V_1 = 8.33 \text{ m/s}$$

$$0.659 - F_x = 10^3(0.0942)11.59 + 10^3(0.0530)10.39 - 10^3(0.1473)8.33$$
$$= 0.417 \text{ kN}$$
$$F_x = 0.659 - 0.417 = 0.242 \text{ kN} \leftarrow$$

Eq. (6.7*b*): $$\Sigma F_y = 0 - 0 + F_y = (\rho Q_2V_{2y} + \rho Q_3V_{3y}) - \rho Q_1V_{1y}$$

$$V_{2y} = V_2 \sin 15° = 12(0.259) = 3.11 \text{ m/s}$$

$$V_{3y} = -V_3 \sin 30° = -12(0.50) = -6.00 \text{ m/s}, \quad V_{1y} = 0$$

So $$F_y = 10^3(0.0942)3.11 + 10^3(0.0530)(-6.00) - 10^3(0.1473)(0)$$
$$= 0.291 - 0.318 - 0 = -0.027 \text{ kN}\uparrow = 0.027 \text{ kN}\downarrow$$

The minus sign indicates that the direction we assumed for F_y was wrong. Therefore F_y acts in the negative y direction. $F_{L/N}$ is equal and opposite to F.

$$(F_{L/N})_x = 0.242 \text{ kN}\rightarrow \quad \text{(in the positive } x \text{ direction)}$$
$$(F_{L/N})_y = 0.027 \text{ kN}\uparrow \quad \text{(in the positive } y \text{ direction)}$$
$$F_{L/N} = 0.243 \text{ kN at } 5.90° \quad \textit{ANS}$$

EXERCISES

6.5.1 A nozzle that discharges a 4-in-diameter water jet into the air is on the right end of a horizontal 10-in-diameter pipe. In the pipe the water has a velocity of 12 fps and a gage pressure of 60 psi. Find the magnitude and direction of the resultant axial force the water exerts on the nozzle, and the head loss in the nozzle.

6.5.2 A nozzle that discharges a 60-mm-diameter water jet into the air is on the right end of a horizontal 120-mm-diameter pipe. In the pipe the water has a velocity of 4 m/s and a gage pressure of 400 kPa. Find the magnitude and direction of the resultant axial force the water exerts on the nozzle, and the head loss in the nozzle.

6.5.3 In Fig. X6.5.3, both nozzles discharge water horizontally into the atmosphere at 30 fps. Find θ so that the resultant force on the unit is along the axis of the 8-in-diameter pipe.

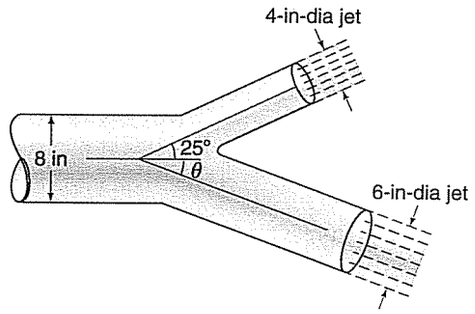


Figure X6.5.3

6.5.4 Water under a gage pressure of 65 psi flows with a velocity of 12.5 fps through a right-angled bend that has a uniform diameter of 10 in. The bend lies in a horizontal plane, and water enters from the west and leaves toward the north. Assuming no drop in pressure, what is the magnitude and direction of the resultant force acting on the bend?

6.5.5 Water under a gage pressure of 350 kPa flows with a velocity of 5 m/s through a right-angled bend that has a uniform diameter of 250 mm. The bend lies in a horizontal plane, and water enters from the west and leaves toward the north. Assuming no drop in pressure, what is the magnitude and direction of the resultant force acting on the bend?

6.5.6 A diverging nozzle that discharges an 8-in-diameter water jet into the air is on the right end of a horizontal 6-in-diameter pipe. If the velocity in the pipe is 12 fps, find the magnitude and direction of the resultant axial force the water exerts on the nozzle. Neglect fluid friction.

6.6 FORCE OF A FREE JET ON A STATIONARY VANE OR BLADE

We can use a procedure similar to that of Sec. 6.5 to find the force a jet exerts on a stationary vane or blade. The main difference is that with a jet in the open atmosphere the gage pressures p in the jet are zero and the pA forces disappear. We call a jet in contact with the atmosphere in this way a ***free jet.*** Another difference is that in many types of fluid machinery that use vanes or blades the velocities are often so high that neglecting friction may introduce a sizeable error. In such cases, for accurate results, we should consider friction. We usually handle this by prescribing a reduction in the velocity of the flow between its arrival and departure points on the blade. The following sample problem illustrates these points.

POLITECNICA SALESIANA BIBLIOTECA

SAMPLE PROBLEM 6.3 A free jet of water with an initial diameter of 2 in strikes the vane shown in Fig. S6.3. Given that $\theta = 30°$ and $V_1 = 100$ fps. Owing to friction losses assume that $V_2 = 95$ fps. Flow occurs in a horizontal plane. Find the resultant force on the blade.

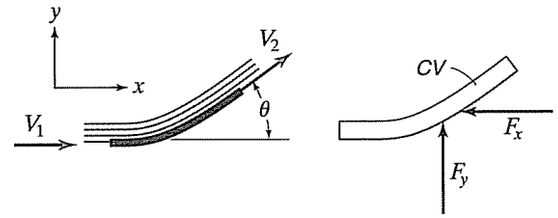


Figure S6.3

Solution

Take as a free body diagram the element of fluid (control volume, *CV*) in contact with the blade. Assume the forces acting on the element are as shown in the sketch. The forces F_x and F_y represent the components (directions assumed) of the net force of the blade on the water (*CV*) in the x and y directions. This net force includes shear stresses tangential to the blade and pressure forces normal to the blade.

Applying Eq. (6.7*a*) along the x axis and noting that $A = \pi(2/12)^2/4 = 0.0218$ ft^2,

$$-F_x = \rho Q(V_{2x} - V_{1x}) = 1.94(0.0218 \times 100)(95\cos 30° - 100)$$
$$= 4.23(-17.7) = -75.0 \text{ lb}$$

So
$$F_x = +75.0 \text{ lb} = 75.0 \text{ lb} \leftarrow$$

The plus sign indicates that the assumed direction of F_x was correct.

Applying Eq. (6.7*b*) along the y axis,

$$+F_y = \rho Q(V_{2y} - V_{1y}) = 4.23(95 \sin 30° - 0) = +201 \text{ lb} = 201 \text{ lb} \uparrow$$

The resultant force of the blade on the control volume is the sum of these two components. The force of the fluid on the blade is equal and opposite to this. The resultant force on the blade is 214 lb at an angle of 69.5° ↘ ***ANS***

Note that if we neglected friction (i.e., $V_2 = V_1 = 100$ fps), we would have calculated the forces as $F_x = 56.7$ lb and $F_y = 212$ lb. So, when the angle of deflection θ from the initial direction of the jet is less than 90°, we find that friction increases the value of F_x over the value it would have if there were no friction. But when θ is greater than 90°, friction decreases the value of F_x. On the other hand, friction decreases the value of F_y for *any* value of angle θ.

If the flow had been in a vertical plane, we would have to consider the effect on V_2 of the higher elevation at exit from the blade, and we would have to estimate the weight of the liquid on the blade and add it to $\rho Q(\Delta V_z)$ to get the total value of F_z.

EXERCISES

6.6.1 A 1-in-diameter jet has a velocity of 95 fps. Calculate the resultant force on a large flat plate if this jet were to strike it normally.

6.6.2 A 40-mm-diameter jet has a velocity of 25 m/s. If this jet were to strike a large flat plate normally, what would be the resultant force on the plate?

6.6.3 In Exer. 6.6.1 assume the center of the jet is coincident with the center of the circular plate. Find (*a*) the stagnation pressure and (*b*) the average pressure on the plate if the area of the plate is 22 times the area of the jet.

6.6.4 In Exer. 6.6.2 assume the center of the jet is coincident with the center of the circular plate. Find (*a*) the stagnation pressure and (*b*) the average pressure on the plate if the area of the plate is 25 times the area of the jet.

6.6.5 A jet containing any type of fluid of specific weight γ and with velocity V and area A is deflected through an angle θ without changing the velocity magnitude. Derive an equation for the dynamic force exerted.

6.6.6 In Fig. X6.6.6 assume that friction is negligible, that $\theta = 115°$, and that the water jet has a velocity of 95 fps and a diameter of 1 in. Find (*a*) the component of the force acting on the blade in the direction of the jet; (*b*) the force component normal to the jet; (*c*) the magnitude and direction of the resultant force exerted on the blade.

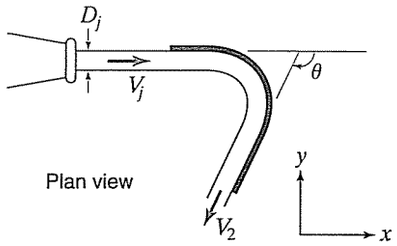


Figure X6.6.6

6.6.7 Refer to Fig. X6.6.6. Assume that friction is negligible, that $\theta = 115°$, and that the water jet has a velocity of 25 m/s and a diameter of 40 mm. Find (*a*) the component of the force acting on the blade in the direction of the jet; (*b*) the force component normal to the jet; (*c*) the magnitude and direction of the resultant force exerted on the blade.

6.7 MOVING VANES: RELATION BETWEEN ABSOLUTE AND RELATIVE VELOCITIES

In much of the work that follows we will need to deal with both absolute and relative velocities of the fluid. The absolute velocity $\mathbf{V}$ of a body (Fig. 6.6) is its velocity relative to the earth. The relative velocity $\boldsymbol{v}$ of a body is its velocity relative to a second body,[1] which may in turn be in motion with absolute velocity $\mathbf{u}$ relative to the earth.

The absolute velocity $\mathbf{V}$ of the first body is the vector sum of its velocity $\boldsymbol{v}$ relative to the second body and the absolute velocity $\mathbf{u}$ of the latter. The relation

[1] This text uses a rounded lower case v (vee) to help distinguish it from the capital V and from the Greek ν (nu) used for kinematic viscosity.

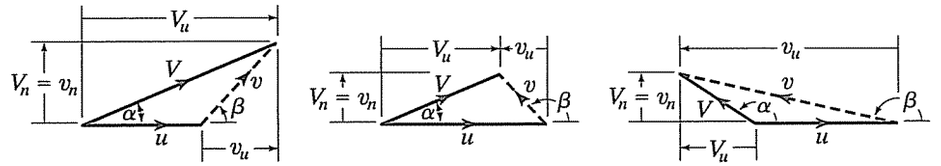


Figure 6.6
Relative (dashed) and absolute velocity relations, with their components.

of the three is thus

$$\mathbf{V} = \mathbf{u} + \boldsymbol{v} \tag{6.16}$$

Because the directions of these three velocities may vary, we represent them by vectors and by the boldface variables $\mathbf{V}$, $\mathbf{u}$, and $\boldsymbol{v}$ in equations. The relative velocities $\boldsymbol{v}$ are shown dashed in Fig. 6.6, which presents three different possible forms of the relations.

Let us define α and β as the angles made by the absolute and relative velocities of a fluid, respectively, with the positive direction of the linear velocity $\mathbf{u}$ of some point on a solid body. We see in Fig. 6.6 that, whatever the shape of the velocity vector triangle, the velocity components parallel and normal to $\mathbf{u}$ are always given by

$$\| \text{ to } \mathbf{u}: \qquad V_u = u + v_u = V\cos\alpha = u + v\cos\beta \tag{6.17}$$

$$\perp \text{ to } \mathbf{u}: \qquad V_n = v_n = V\sin\alpha = v\sin\beta \tag{6.18}$$

Here the subscript u indicates components parallel to $\mathbf{u}$, and the subscript n indicates components normal to $\mathbf{u}$. For *rotating* blades (Secs. 6.11–6.14), we shall see that u corresponds to the tangential direction and n corresponds to the radial direction.

6.8 FORCE OF A JET ON ONE OR MORE MOVING VANES OR BLADES

Single Blade, Moving Parallel to Jet

We can determine the force exerted by a stream on a single moving object by an equation very similar to Eq. (6.6), *provided the flow is steady and the body has a motion of translation along the same line as the initial stream,* i.e., provided $\mathbf{u}$ is parallel to $\mathbf{V}_1$. If the latter condition is not fulfilled, the case becomes a complex one of unsteady flow.

There are two major differences between jet action on a stationary object and on a moving object. The first is that the amount of fluid that strikes a single moving object in any time interval Δt must be different from that which strikes a stationary object, and so this changes the rate of momentum transfer. The second difference is that for a moving object we must consider both relative and absolute velocities, which makes the determination of the required $\Delta\mathbf{V}$ more involved.

Let us consider the first issue, regarding the rate of momentum transfer. If the cross-sectional area of a jet is A_1 and its velocity is V_1 then the rate at which fluid issues from the nozzle in terms of volume is $Q = A_1V_1$ and the rate in terms of mass in $\dot{m} = \rho Q = \rho A_1V_1$. This of course is the same as the amount of fluid that would strike a stationary body. But the amount of fluid that strikes a single moving body per unit time will be less than this if the body is moving away from the nozzle and more than this if it is moving toward the nozzle. As an extreme case, suppose the body is moving away from the jet (along the same axis) and with the same or higher velocity, magnitude u. It is clear that none of the fluid will then act upon the body. But if it is moving with a velocity u less than that of the jet, the amount of fluid that strikes the body per unit time will be proportional to the difference between the two velocities, i.e., to $V_1 - u = v_1 =$ the relative velocity. Accordingly, the rate at which fluid strikes the moving body will be, on a volume basis,

$$Q' = A_1(V_1 - u) = A_1v_1 \tag{6.19}$$

and, on a mass basis,

$$\dot{m}' = \rho Q' = \rho A_1(V_1 - u) = \rho A_1v_1 \tag{6.20}$$

In effect, then, Q' and $\dot{m}'$ are *relative* flow rates. The difference between Q and Q' is also apparent if we consider Fig. 6.7, where the fluid issues from a nozzle at the rate $Q = A_1V_1$ per unit time. But the object has moved away from the nozzle a distance u in this unit of time, and the volume of fluid between the two has increased by the amount A_1u. Because V_1 is greater than u, the difference, equal to $A_1V_1 - A_1u = A_1(V_1 - u) = A_1v_1$, must be the amount that struck the object within unit time.

Let us now consider the second issue, regarding the relative velocities. The $\Delta\mathbf{V}$ in momentum equation (6.6) is the difference of the absolute velocities, $\mathbf{V}_{out} - \mathbf{V}_{in}$. In Fig. 6.7 this is the same as $\mathbf{V}_2 - \mathbf{V}_1$, and $\Delta\mathbf{V}$ is shown on a velocity triangle. Usually $\mathbf{V}_2$ is unknown, and therefore so is the required $\Delta\mathbf{V}$. We can find these by first solving the relative velocity triangle (Fig. 6.6) for exit, and by next solving the absolute velocity triangle in Fig. 6.7. However, we can often avoid much of this work, because for the special case we are considering here in which u is constant for the entire vane (blade) or body, we have

$$\begin{aligned}\Delta\mathbf{V} &= \mathbf{V}_2 - \mathbf{V}_1\\ &= (\mathbf{u} + \boldsymbol{v}_2) - (\mathbf{u} + \boldsymbol{v}_1)\\ &= \boldsymbol{v}_2 - \boldsymbol{v}_1\\ &= \Delta\boldsymbol{v}\end{aligned}$$

Thus we can use either $\Delta\mathbf{V}$ or $\Delta\boldsymbol{v}$ in this case, and $\Delta\boldsymbol{v}$ is often easier to obtain.

Noting the outcomes of the two issues just discussed, for the present case of a single moving vane, Eq. (6.6) therefore becomes

$$\sum\mathbf{F} = \dot{m}'(\Delta\mathbf{V}) = \dot{m}'(\Delta\boldsymbol{v}) = \rho Q'(\Delta\mathbf{V}) = \rho Q'(\Delta\boldsymbol{v}) \tag{6.21}$$

Usually we are most interested in the force exerted in the direction of the jet, which we previously named the u or the x direction. From Fig. 6.7 and

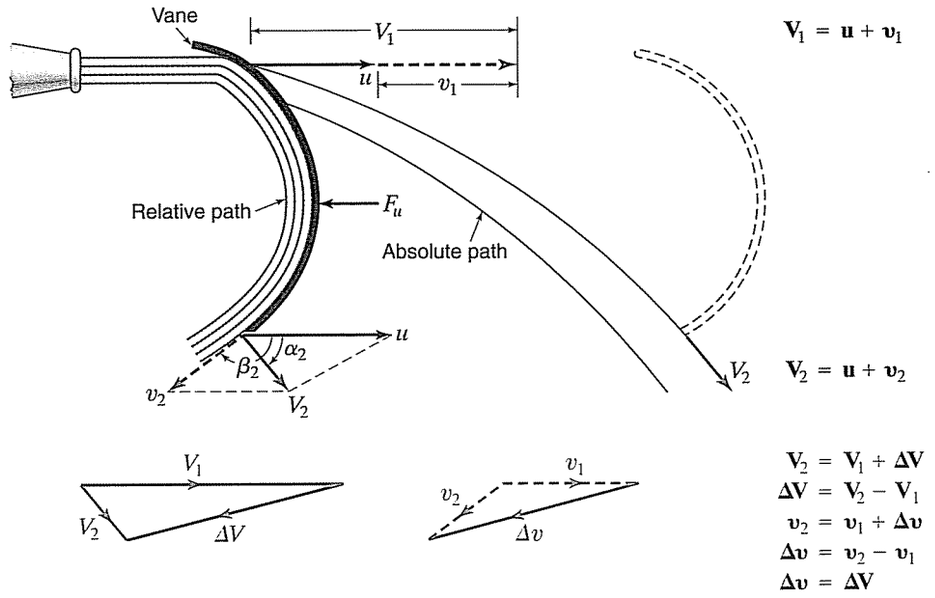


Figure 6.7
Jet acting on a vane in translation.

Eq. (6.21), noting that $p_1 = p_2 = 0$ for a free jet, we obtain

$$\sum F_u = 0 - F_u = \rho Q'(\Delta V_u) = \rho Q'(\Delta v_u) \tag{6.22}$$

where the minus sign results from assuming that the u component of the force of the blade (vane) on the water, F_u, acts to the left in Fig. 6.7. Because the blade slows down the jet, $\Delta v_u = \Delta V_u$ is also negative, and so F_u *does* act to the left. If we remember that F_u acts in the same direction as Δv_u, we can drop the minus sign in Eq. (6.22). Equal and opposite to F_u is the u component of the force of the water jet on the blade, $(F_{W/B})_u$, which therefore acts to the right in Fig. 6.7, i.e., in the direction of the blade movement.

Recalling that $\rho = \gamma/g$ and $Q' = A_1v_1$, another convenient form of these last two equations, which we can use for either force, is

Jet on single blade: $$F_u = \frac{\gamma A_1 v_1}{g}\Delta v_u \tag{6.23}$$

In Fig. 6.7, a particle of fluid that strikes the moving vane (blade) at the instant it is in the position shown by the solid line takes sometime to reach the point of outflow from the vane; during this time the vane will have moved to the position shown by the dashed outline. Thus we can trace two paths for the fluid: one, known as the ***relative path,*** is relative to the moving vane, as it would appear to an observer (or a camera) moving with the vane, and the other, known as the ***absolute path,*** is relative to the earth, as it would appear to an observer (or a camera) stationary with respect to the earth.

From Fig. 6.7 we can see that the direction of the relative velocity at outflow from the vane results from the shape of the latter, but the relative velocity at entrance, just *before* the fluid strikes the vane, depends only on the relation between V_1 and u. Just *after* the fluid strikes the vane, its relative velocity must be tangent to the vane surface. To avoid excess energy loss, these two directions should agree; otherwise there will be an abrupt change in velocity and direction of flow at this point, known as ***shock.***

Series of Rotating Blades

A stream of fluid impinging on a single moving vane, as we just discussed, occurs rarely. More commonly, the jet is directed at a *series* of closely spaced rotating vanes (blades), as with a Pelton wheel (Figs. 16.1 and 16.3). In such cases, whatever flow does not impinge on the first vane will strike the second one, and so on around the circle. As a result, for a series of vanes we use the full momentum transfer rate $\dot{m} = \rho Q = \rho A_1 V_1$ instead of the rate given by Eq. (6.20). Thus we can express the u component of the force exerted by the fluid on a series of vanes as

Jet on series of rotating blades:

$$F_u = \dot{m}\Delta V_u = \rho Q \Delta V_u = \frac{\gamma A_1 V_1}{g}\Delta V_u = \frac{\gamma A_1 V_1}{g}\Delta v_u \qquad (6.24)$$

The force of the series of vanes on the fluid is again in the direction of ΔV_u, that is, to the left in Fig. 6.7.

SAMPLE PROBLEM 6.4 The 2-in-diameter water jet with a velocity of 100 fps shown in Fig. S6.4 impinges on a single vane moving in the same direction (thus $F_x = F_u$) at a velocity of 60 fps. (*a*) If $\beta_2 = 150°$ and friction losses over the vane make $v_2 = 0.9v_1$, compute the net force the water exerts on the vane. If this vane were one of a series of vanes, find (*b*) the horsepower transferred to the vanes, (*c*) the horsepower of the water leaving the system, and (*d*) the horsepower loss due to friction.

Solution

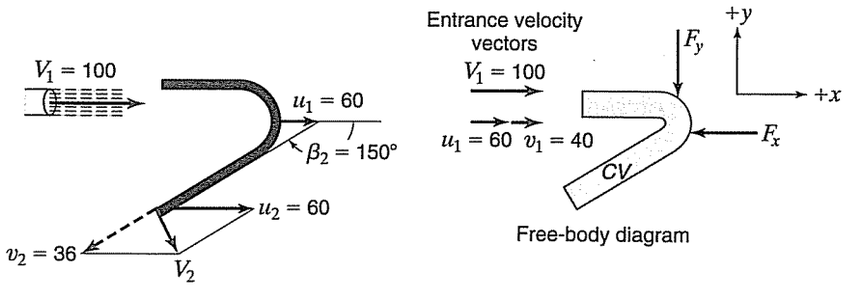


Figure S6.4

(*a*) The velocity vector diagrams at entrance and exit to the vane are given in Fig. S6.4. Since $v_2 = 0.9(40) = 36$ fps,

Eq. (6.17):

$$V_{2u} = V_2\cos\alpha_2 = u + v_2\cos\beta_2 = 60 + 36\cos 150° = 28.8 \text{ fps} \qquad (1)$$

Eq. (6.18): $V_{2n} = V_2\sin\alpha_2 = v_2\sin\beta_2 = -36\sin 150° = -18$ fps (2)

$$Q' = A_1 v_i = A_i(V_1 - u) = \frac{\pi}{4}\left(\frac{2}{12}\right)^2(100 - 60) = 0.873 \text{ cfs}$$

Eq. (6.22):

$$-F_x = \rho Q'(V_{2u} - V_{1u}) = 1.938(0.873)(28.8 - 100) = -120.4 \text{ lb}$$

So $F_x = 120.4$ lb. The force of the vane on the water is to the left, as assumed; hence the force of water on the vane is 120.4 lb to the right.

$$-F_y = \rho Q'(V_{2n} - V_{1n}) = 1.938(0.873)(-18 - 0) = -30.4 \text{ lb}$$

Thus $F_y = 30.4$ lb in the direction shown. The force of water on the vane is equal and opposite and thus 30.4 lb upward.

Therefore the net

$$F_{W/B} = 124.2 \text{ lb at } 14.19° \nearrow \qquad \textbf{\textit{ANS}}$$

If needed, we may be solve (1) and (2) simultaneously to yield $V_2 = 34.0$ fps, $\alpha_2 = 32.0°$.

(*b*) If the blade were one of a *series* of blades,

$$Q = A_1 V_1 = \frac{\pi}{4}\left(\frac{2}{12}\right)^2(100) = 2.18 \text{ cfs}$$

Eq. (6.24):

$$-F_x = \rho Q(V_2\cos\alpha_2 - V_1) = 1.938(2.18)(28.8 - 100) = -301 \text{ lb}$$

The horsepower transferred (delivered) to the blades (i.e., out of the fluid) is

Sec. 5.9: $$P_{\text{transfer}} = \frac{Fu}{550} = \frac{(301)(60)}{550} = 32.8 \text{ hp} \qquad \textbf{\textit{ANS}}$$

Also, for a series of blades, we can use energy considerations for the solution. The horsepower of the original jet is

Sec. 5.9: $$P_{\text{in}} = \frac{\gamma Q(V_1^2/2g)}{550} = \frac{62.4(2.18)(100)^2}{550(2)32.2} = 38.4 \text{ hp}$$

(*c*) The horsepower of the water as it leaves the system is

$$P_{\text{out}} = \frac{\gamma Q(V_2^2/2g)}{550} = \frac{62.4(2.18)(34.0)^2}{550(2)32.2} = 4.44 \text{ hp} \qquad \textbf{\textit{ANS}}$$

(*d*) An equation for conservation of energy expressed in terms of power is

$$P_{\text{in}} - P_{\text{out}} - P_{\text{transfer}} - P_{\text{friction loss}} = 0$$

Thus $38.4 - 4.44 - 32.8 = P_{\text{friction loss}}$

Therefore $P_{\text{friction loss}} = 1.168$ hp ***ANS***

We may verify this by computing

$$\frac{\gamma Q(v_1^2/2g) - \gamma Q(v_2^2/2g)}{550} = \frac{62.4(2.18)[(40)^2 - (36)^2]}{550(2)32.2} = 1.168 \text{ hp}$$

Note: The horsepower loss due to friction is a small percentage of the power of the original jet. Therefore, in problems of this type with free jets, the common assumption that $v_1 = v_2$ in magnitude gives reasonably good results.

EXERCISES

6.8.1 If a jet of fluid strikes a single body moving in the same direction with a velocity u, flows over it without friction loss, and leaves with a relative velocity in the direction of β_2, prove that $F_u = (\gamma A_1/g)(1 - \cos\beta_2)(V_1 - u)^2$.

6.8.2 A jet of water strikes a single vane, which reverses it through 180° without friction loss. If the jet has an area of 3.5 in^2 and a velocity of 175 fps, find the force exerted if the vane moves (*a*) in the same direction as the jet with a velocity of 75 fps; (*b*) in a direction opposite to that of the jet with a velocity of 75 fps.

6.8.3 A jet of water strikes a single vane, which reverses it through 180° without friction loss. If the jet has an area of 2500 mm^2 and a velocity of 55 m/s, find the force exerted if the vane moves (*a*) in the same direction as the jet with a velocity of 20 m/s; (*b*) in a direction opposite to that of the jet with a velocity of 20 m/s.

6.8.4 A 4-in-diameter water jet with a velocity of 105 fps acts on a series of vanes with $\alpha_1 = \beta_1 = 0$. Neglect friction and find the required blade angle β_2 in order that the resultant force acting on the vane in the direction of the jet is 200 lb. Solve using vane velocities of 0, 15, 45, and 75 fps. Also find the maximum possible vane velocity.

6.8.5 A 100-mm-diameter water jet with a velocity of 35 m/s acts on a series of vanes with $\alpha_1 = \beta_1 = 0$. Neglect friction and find the required blade angle β_2 in order that the resultant force acting on the vane in the direction of the jet is 950 N. Solve using vane velocities of 0, 5, 15, and 25 m/s. Also find the maximum possible vane velocity.

6.8.6 What would be the resultant force components on the single vane of Sample Prob. 6.4 if it were traveling to the left toward the nozzle at 15 fps?

6.9 REACTION OF A JET

Consider a jet issuing steadily from a tank (Fig. 6.8). The tank is large enough so that we may neglect the velocities within it. Let the area of the jet be A_2 and its velocity V_2, and assume an ideal fluid, so that $V_2 = \sqrt{2gh}$. In this case, with the jet flowing to the right, the momentum principle indicates that a force equal to $\rho Q_2 V_2$ is exerted to the left on the tank. We may confirm that this is so by

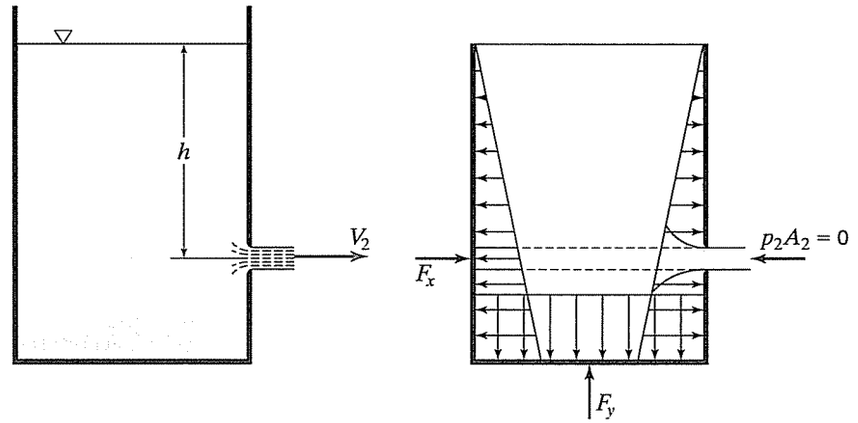


Figure 6.8

applying momentum Eq. (6.7*a*) to the free-body diagram (Fig. 6.8*b*) of the liquid in the tank. In Fig. 6.8*b* two heavy vectors represent the resultant horizontal and vertical forces of the tank on the liquid, while the distributed load represents the force of the liquid on the tank. The distributed load shown is in the vertical plane through the centerline of the jet.

Applying Eq. (6.7*a*) to the liquid (*CV*), we get

$$\sum F_x = F_x = \rho Q_2(V_2 - 0) = \rho A_2 V_2^2 = \rho A_2(2gh) = 2\gamma h A_2 \qquad (6.25)$$

This is the net force of the tank on the ideal liquid in the x direction; it acts to the right, and causes the increase in velocity of the flowing liquid, from zero to V_2. Equal and opposite to this force is the force of the liquid on the tank, often referred to as the ***jet reaction.*** If the tank were supported on frictionless rollers, this force would move it to the left. This net force $\rho Q_2 V_2$ is equal to the difference in the magnitude of the pressure forces on the two sides of the tank. On the left side a typical hydrostatic pressure distribution exists, while on the right side the pressure is lower near the orifice because of the increase in velocity in that region (see Fig. 6.8*b*). From the last term of Eq. (6.25) we see that this net jet reaction force is equal to *twice* the hydrostatic force on A_2. Thus the net force (shaded area at the right side of the tank) is equal to twice the hydrostatic force on A_2 (shaded area at the left side of the tank).

Let us now refer to Fig. 6.9, where a jet of the same liquid of cross-sectional area A_1 discharges *into* the tank with a velocity V_1. In this case the jet exerts a force $\mathbf{F} = \rho Q_1 \mathbf{V}_1$ on the liquid, which, in turn, transmits the force to the tank. We call this ***jet action.***

Two jets act on the tank of Fig. 6.9, one entering the tank at section 1 and the other leaving the tank at section 2. The resultant force on the tank is the vector sum of $\rho Q_1 \mathbf{V}_1$ and $\rho Q_2 \mathbf{V}_2$, where the first vector (jet action) acts in the direction of V_1 (downward to the right in Fig. 6.9) and the second vector (jet reaction) acts in the direction opposite to that of V_2. Thus a jet *entering* a system acts

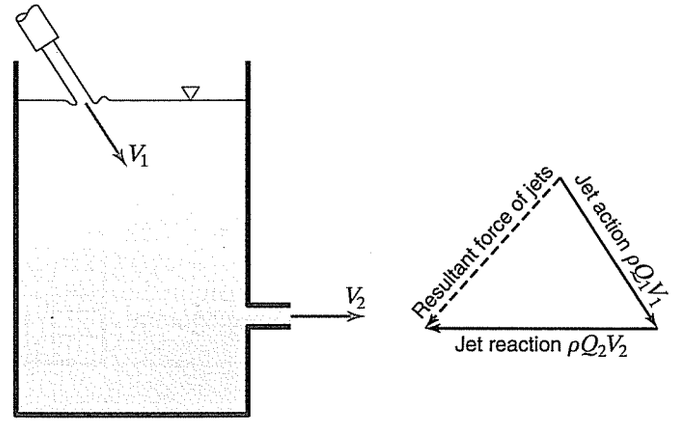
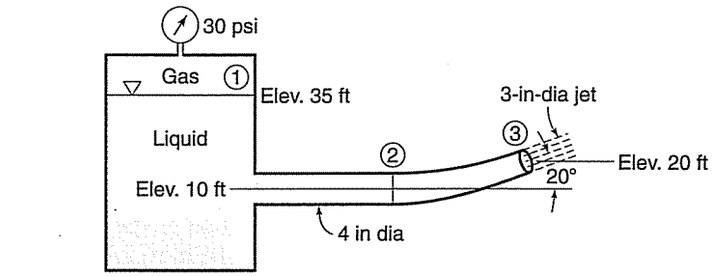


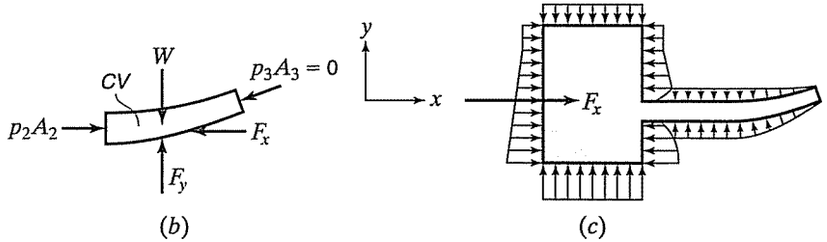
Figure 6.9 (*a*) (*b*)

on the system in the *same* direction in which the jet is traveling, while a jet *leaving* a system acts on the system in the direction *opposite* to that in which the jet is traveling.

SAMPLE PROBLEM 6.5 In Fig. S6.5*a* a curved pipe section 40 ft long is attached to a section of straight pipe. (*a*) Determine the resultant force on the curved pipe, and (*b*) find the horizontal component of the jet reaction. All significant data are given in the figure. Assume an ideal liquid with $\gamma = 55$ lb/ft^3.



(*a*)



(*b*) (*c*)

Figure S6.5

Solution

Energy Eq. (5.29) between points 1 and 3 in Fig. S6.5*a* gives

$$\frac{30(144)}{55} + 35 + 0 = 0 + 20 + \frac{V_3^2}{2(32.2)}$$

from which the jet velocity $V_3 = 77.6$ fps.

(*a*) For the curved pipe:

$$A_3 = \frac{\pi}{4}\left(\frac{3}{12}\right)^2 = 0.0491 \text{ ft}^2, \quad \text{so} \quad Q = A_3V_3 = 3.81 \text{ cfs}$$

$$A_2 = \frac{\pi}{4}\left(\frac{4}{12}\right)^2 = 0.0873 \text{ ft}^2, \quad \text{so} \quad V_2 = \frac{Q}{A_2} = 43.6 \text{ fps}$$

Energy Eq. (5.29) between points 2 and 3 gives

$$\frac{p_2(144)}{55} + 10 + \frac{(43.6)^2}{2(32.2)} = 0 + 20 + \frac{(77.6)^2}{2(32.2)}$$

from which $p_2 = 28.3$ psi

The free-body diagram of the forces acting on the liquid contained in the curved pipe between points 2 and 3 (the control volume) is shown in Fig. S6.5*b*. Applying Eq. (6.7*a*),

$$\sum F_x = p_2A_2 - p_3A_3\cos 20° - F_x = \rho Q(V_3\cos 20° - V_2)$$

where F_x represents the force of the curved pipe on the liquid (*CV*) in the x direction. Since section 3 is a jet in contact with the atmosphere, $p_3 = 0$. The liquid density $\rho = 55/32.2 = 1.708$ slug/ft^3. Thus

$$28.3\left(\frac{\pi}{4} \times 4^2\right) - 0 - F_x = (1.708)3.81(77.6\cos 20° - 43.6)$$

$$356 - F_x = 191$$

$$F_x = +165 \text{ lb} = 165 \text{ lb} \leftarrow$$

The plus sign indicates that the assumed direction is correct. In the y direction the p_2A_2 force has no component. Estimating the weight of liquid W as 150 lb,

$$\sum F_y = 0 - 0 + F_y - 150 = (1.708)3.81(77.6\sin 20° - 0) = 173 \text{ lb}$$

$$F_y = 173 + 150 = +323 \text{ lb} = 323 \text{ lb} \uparrow$$

The resultant force of liquid on the curved pipe is equal and opposite to the force of the curved pipe on the liquid. The resultant force of liquid on the curved

pipe is $[(165)^2 + (323)^2]^{1/2} = 363$ lb downward and to the right at an angle of 62.9° with the horizontal. ***ANS***

(*b*) For the entire system:

The horizontal jet reaction is best found by taking a free-body diagram of the liquid (*CV*) in the entire system as shown in Fig. S6.5*c*. From Eq. (6.7*a*),

$$F_x = \rho Q(V_3 \cos 20° - 0) = +475 \text{ lb} = 475 \text{ lb} \rightarrow$$

where F_x represents the force of the system on the liquid in the x direction. F_x is equivalent to the integrated effect of the x components of the pressure vectors shown in Fig. S6.5*c*. Equal and opposite to F_x is the force of the liquid on the system, i.e., the jet reaction. So the horizontal jet reaction is a 475-lb force to the left. ***ANS***

In summary, therefore, there is a 165-lb force to the right tending to separate the curved pipe section from the straight pipe section, while at the same time there is a 475-lb force tending to move the entire system to the left.

EXERCISES

6.9.1 Find the thrust developed when water is pumped in through a 9-in-diameter pipe in the bow of a boat at $v = 6$ fps and emitted through a 5-in-diameter pipe in the stern of the boat.

6.9.2 Find the thrust developed when water is pumped in through a 225-mm-diameter pipe in the bow of a boat at $v = 2.5$ m/s and emitted through a 125-mm-diameter pipe in the stern of the boat.

6.10 JET PROPULSION

In Sec. 6.9 we derived an expression for the reaction of a jet from a stationary tank. Assume now that the tank in Fig. 6.8 is moving to the left with a velocity u. If the orifice is small compared with the size of the tank, we may disregard the relative velocity within the tank, and we may also disregard any change in h for a short interval of time. Thus the absolute velocity of the fluid within the tank is $V_1 = u$ to the left. If the jet issues from the orifice with a relative velocity v_2, taking velocities to the right as positive, the absolute velocity of the jet will be $V_2 = v_2 - u$. Thus

$$\Delta V = V_2 - V_1 = (v_2 - u) - (-u) = v_2$$

We obtain the same result if the tank is stationary (i.e., $u = 0$). Then $\Delta V = V_2 - 0 = v_2$. Therefore the force of reaction is independent of the velocity of the tank, and Eq. (6.25) applies for either rest or motion.

Rocket

Both the fuel and the oxygen for combustion are contained within a rocket, which is analogous to the tank of Fig. 6.8. The only difference is that the exit pressure p_n of the gases leaving the nozzle or orifice at section 2 may exceed the atmospheric pressure p_a. If A_2 equals the area of the jet, the rocket thrust is

$$F = \rho A_2 v_2^2 + (p_n - p_a)A_2 \tag{6.26}$$

where v_2 is the velocity at which the jet issues from the rocket. The thrust F is independent of the speed of the rocket.

Jet Engine

A ***jet engine*** is a device that carries only its fuel, and takes in the air for combustion from the atmosphere. It is analogous to the tank of Fig. 6.9, including the intake of fluid at section 1, except that the velocity of the air received is usually in the same straight line as the velocity of the exit jet at section 2. There are three forms of jet engines, but the equation is the same for all three. The ***ramjet*** must achieve a sufficiently high speed by some other means so that it can scoop in air from in front that has been compressed by the stagnation pressure due to its speed (Sec. 5.4). The ***turbojet*** can take off from the ground, since in it a compressor driven by a gas turbine compresses the air, and the exhaust from the gas turbine supplies the jet propulsion. Then there is a ***pulsating machine,*** which scoops in air in cycles; after scooping the inlet is closed, the fuel–air mixture explodes, and a jet gives the device a spurt; then the process repeats itself.

The thrust of a jet engine is

$$F = (\dot{m}_a + \dot{m}_f)v_2 - \dot{m}_a u \tag{6.27}$$

where $\dot{m}_a$ = mass of air entering per second
$\dot{m}_f$ = mass of fuel consumed per second
v_2 = velocity of exhaust with respect to the engine
u = velocity of flight
= velocity of air entry with respect to the engine

The thrust varies with the speed of flight. Usually $p_n = p_a$, and so we do not include the last term of Eq. (6.26) in Eq. (6.27).

EXERCISES

6.10.1 Find the thrust of a turbojet whose speed is 750 fps and whose air intake rate is 50 lb/sec. The air/fuel ratio is 30:1 and the exhaust velocity is 1800 fps.

6.10.2 Find the thrust of a turbojet whose speed is 280 m/s and whose air intake rate is 12 kg/s. The air/fuel ratio is 25:1 and the exhaust velocity is 550 m/s.

6.11 ROTATING MACHINES: CONTINUITY, RELATIVE VELOCITIES, TORQUE[2]

In this section we shall consider only rotating *hydraulic* machines, designed for incompressible liquids like water. Further, we shall consider here only those machines that flow full of liquid, so that pressures may vary within them. Such machines fall into two categories, ***pumps*** and ***turbines,*** which are discussed further in Chaps. 15 and 16.

These machines work when liquid flows through a rotating element known as a ***rotor.*** The rotor has a number of blades or vanes, which guide the flow through and which make possible the energy exchange between liquid and machine. In pumps the rotor is called an ***impeller,*** and in turbines it is called a ***runner.***

Different machines have different flow patterns. Figures 6.10–6.12 show two-dimensional flow in planes normal to the axis of rotation. This is known as ***radial flow.*** The streamlines and velocities lie in the plane of the paper and so are readily represented. Do not misinterpret the term "radial flow" to mean that the flow is confined along radial lines like the spokes of a wheel, it is not. With radial flow, although the flow paths remain in the normal plane, they are curved, as indicated in Figs. 6.10–6.12. Radial flow is usually inward in turbines for

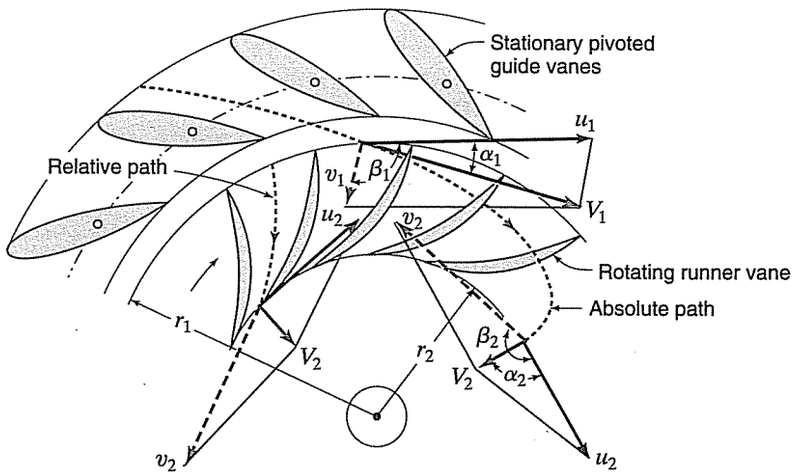


Figure 6.10
Radial-flow hydraulic turbine. (Flow is inward.)

[2] This and Secs. 6.12–6.15 will be of greater interest to those concerned with applications of the momentum principle to rotating machines for liquids and gases, and they provide an important introduction to Chaps. 15 and 16 on hydraulic machinery. They may be omitted with little detrimental impact on Chaps. 7–14.

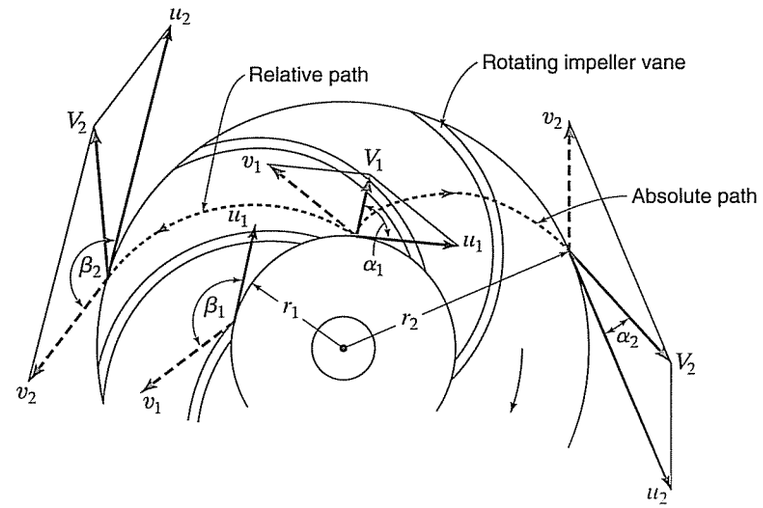
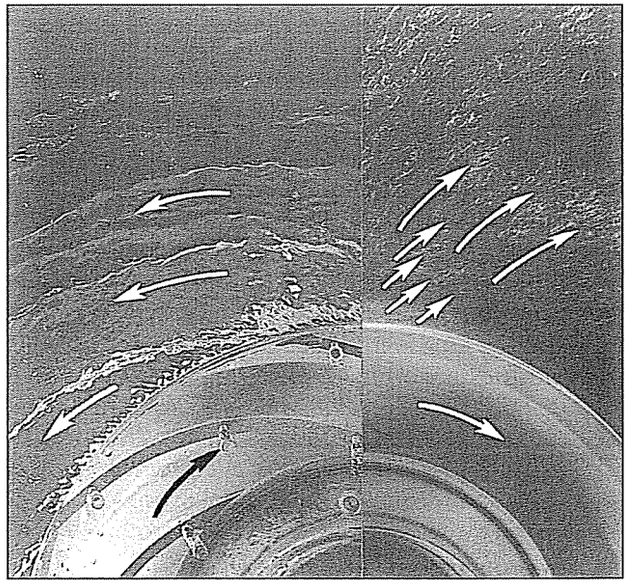
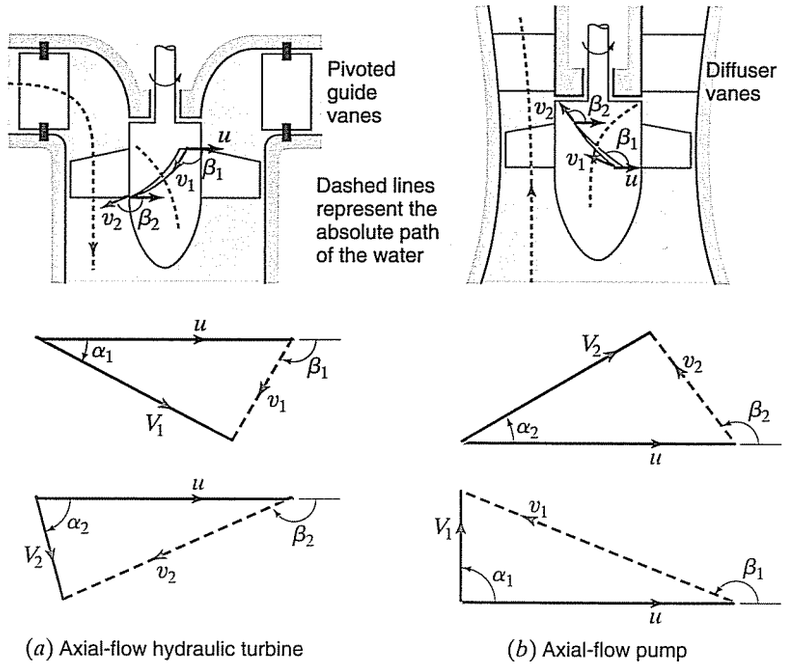


Figure 6.11
Centrifugal-pump impeller with radial flow. (Flow is outward.)



(*a*) (*b*)

Figure 6.12
Radial-flow-pump impeller rotating at 200 rpm. (*a*) Instantaneous photo showing relative flow. (*b*) Time exposure showing absolute flow. (Courtesy of the Archives, California Institute of Technology)



(*a*) Axial-flow hydraulic turbine (*b*) Axial-flow pump

Figure 6.13
Axial flow machines.

practical reasons of construction; in pumps it is outward due to centrifugal action, and we call such pumps ***centrifugal pumps.***

In Fig. 6.13 we see a different flow pattern, both for a turbine and a pump, in which fluid particles remain at constant distances from the rotation axis. This we call ***axial flow,*** which occurs for example through fans and ship's propellers. The streamlines are spirals on coaxial cylinders.

Mixed flow is intermediate between the two extremes just described, and its velocities have radial, axial, and tangential components. A streamline is a conical spiral with a varying radius from the axis of rotation. Needless to say, this is a complicated three-dimensional flow situation.

Although we will develop correct and accurate-looking equations for velocities and torque, etc., in the following paragraphs and sections, we must remain aware that it is difficult to determine the numerical values to be used in them. Thus fluid particles in different streamlines may flow with different velocities, and we must estimate what the average velocity may be. Also, investigators have determined that the average direction of a stream is often different from that of the vane that it is supposed to follow, but as yet we do not know the exact amount of deviation in every case. Thus we cannot even obtain the average velocity precisely by dividing the flow by the cross-sectional area of a rotor passage. Furthermore, the entrance or exit edges of vanes are not always parallel to the axis of rotation, and so the radii will be different for different streamlines.

Despite these defects, the idealized theory is useful. It yields the shape or nature of the performance curves of a given machine; it indicates the influence of each separate factor; and it suggests the direction in which to make design changes in order to alter the performance characteristics of an existing machine.

Continuity

For radial-flow machines, we can make an approximate analysis of the flow behavior by assuming that all elements of the vanes are parallel to the rotation axis and that water enters and leaves the vanes smoothly. Sample Prob. 6.6 provides an example of such an analysis. A key feature of the analysis is its use of the principle of flow continuity in the radial direction. Namely,

$$Q = A_{c1}V_{r1} = A_{c2}V_{r2} \tag{6.28}$$

where A_{c1} and A_{c2} represent circumferential flow areas, and V_{r1} and V_{r2} represent the radial components of the velocity at radii r_1 and r_2. Because the vanes occupy some of the space, we should find the circumferential flow area A_c from the total circumferential area multiplied by a reduction factor m, which we can calculate from the blade thickness, the number of blades, and the circumference. Thus

$$A_c = m2\pi rB \tag{6.29}$$

where B is the depth of the flow passage between the sides of the turbine, and

$$m = 1 - \frac{nt}{2\pi r} \tag{6.30}$$

where n is the number of blades and t is the blade thickness. Often we assume m_1 and m_2 have values of 1.0, although in fact m must be less than 1.0, perhaps about 0.8–0.9. Usually the passage depths B_1 and B_2 are equal. With such an approach, therefore, given Q and the machine dimensions, we can calculate an average value of the radial velocity component V_r at any radius in a radial-flow machine.

For axial-flow machines, it might appear that we could apply a similar analysis to the flow area, now annular in shape. However, because the blade speed varies with radius, velocity variations are more complex, making analyses less reliable.

Velocity Triangles for Radial Flow

We can continue our theoretical analysis, and solve velocity triangles like those on Figs. 6.10 and 6.11, if, in addition to the flow rate and machine dimensions discussed just above, we know the blade angles β and the angular speed of the rotor ω. Before doing so, however, it is important that we clearly understand the various motions, absolute and relative flow paths, and velocity triangles represented in Figs. 6.10 and 6.11.

First, we note that the tangential blade speed u is equal to $r\omega$. The radial component of the flow velocity, V_r, which we can obtain as just described, is perpendicular to u. We see that we can use the relations of Sec. 6.7 and Fig. 6.6 to solve the velocity triangles, noting that the subscript n (for normal) in Sec. 6.7 has now become the subscript r, for radial. Also, we note from Sec. 6.7 that $v_r = V_r$.

For each radius of interest, we can proceed as follows. Find $v_u = v\cos\beta$ from $V_r/\tan\beta$. Next find v from $v_u/\cos\beta$ or from $v_r/\sin\beta = V_r/\sin\beta$ or from $\sqrt{v_u^2 + v_r^2}$. Then find $V_u = V\cos\alpha$ from $u + v_u$ (Eq. 6.17). Next find α from $\tan^{-1}(V_r/V_u)$, and last find V from $V_u/\cos\alpha$ or from $\sqrt{V_u^2 + V_r^2}$. Sample Prob. 6.6 includes an example of such calculations. When making repetitive calculations of this type, it is very helpful to tabulate (or do so by using a spreadsheet).

Torque

Because the radius usually varies as fluid flows through a rotor, we prefer to compute ***torque*** rather than force. The resultant torque is the sum of the torques produced by all the elementary forces, but investigators have shown that we may consider this sum as equivalent to two single forces, one concentrated at the entrance to and the other at the exit from any device. For steady flow, these equivalent forces are equal to ρQV_1 and ρQV_2 (see, for example, Eq. 6.5). Referring to Figs. 6.10 and 6.11 and taking moments, the resulting torque is

$$T = \rho Q(r_1V_1\cos\alpha_1 - r_2V_2\cos\alpha_2) \tag{6.31}$$

This equation states that torque equals the time rate of change of moment of momentum (angular momentum). As before in Sec. 6.7, we define α and β as the angles made by the absolute and relative velocities of the fluid, respectively, with the positive direction of the linear velocity u of a point on the moving body (tip or root of the blade).

The subscripts in Eq. (6.31), when compared with Figs. 6.10–6.11, indicate that it computes torque from inflow minus outflow, instead of the opposite that we previously used to find effects on the fluid (CV). Therefore this equation computes the torque acting on the *machine*. From the sizes of the radii and the tangential velocities, it follows that if T resulting from this equation is positive, it is the value of the torque that the fluid exerts on the runner of a turbine. The torque output from the shaft of the turbine is less than this because of mechanical friction. If the value of T is negative, it represents the torque that the impeller of a pump or compressor or fan exerts on the fluid. The torque input to the shaft of such a machine is greater than this because of mechanical friction.

When using Eq. (6.31) and subsequent equations it is immaterial whether the fluid flows radially inward, as in Fig. 6.10, or radially outward, as in Figs. 6.11 and 6.12, or remains at a constant distance from the axis, as in Fig. 6.13. In any case, r_1 is the radius at entrance and r_2 is that at exit.

SAMPLE PROBLEM 6.6 A radial-flow turbine runner has 18 blades each 0.2 in thick, with $r_1 = 10$ in, $\beta_1 = 65°$, $r_2 = 6$ in, and $\beta_2 = 122°$. The depth B of the flow passage between the two sides of the turbine is 4 in. When rotating at 180 rpm, the water flow rate is 7.5 cfs. (*a*) For both entrance and exit conditions, tabulate values of m, A_c, and V_r, showing the method of calculation. (*b*) For the same conditions, tabulate values of v_u, v, u, V_u, α, and V, and draw and label velocity triangles. (*c*) Find the torque exerted by the water, and the horsepower delivered to the shaft. Assume that water enters and leaves the blades without shock.

Solution

(a) Continuity

Eq. (6.30): $$m = 1 - \frac{nt}{2\pi r} = 1 - \frac{18(0.2)}{2\pi r}$$

Eq. (6.29): $$A_c = m2\pi rB = m2\pi r\left(\frac{4}{12}\right)$$

From Eq. (6.28): $$V_r = v_r = \frac{Q}{A} = \frac{7.5}{mA_c}$$

Substituting for r-values and evaluating,

Point	r, ft	m	A_c, ft^2	$V_r = v_r$, fps	
1	0.833	0.943	1.645	4.56	***ANS***
2	0.5	0.905	0.947	7.92	***ANS***

(b) Velocity triangles

$$\omega = \frac{2\pi n}{60} = 180\left(\frac{2\pi}{60}\right) = 18.85 \text{ rad/sec}$$

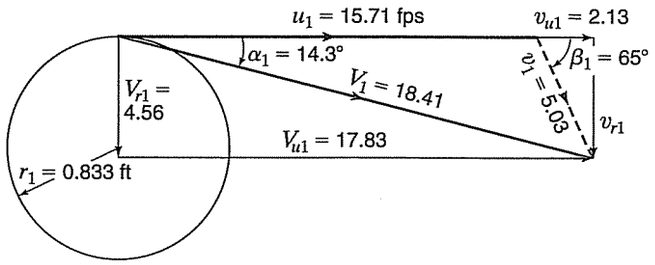
$$v_u = \frac{V_r}{\tan\beta};\quad v = \frac{v_u}{\cos\beta};\quad u = r\omega;\quad V_u = u + v_u;$$

$$\alpha = \tan^{-1}\left(\frac{V_r}{V_u}\right);\quad V = \frac{V_u}{\cos\alpha}$$

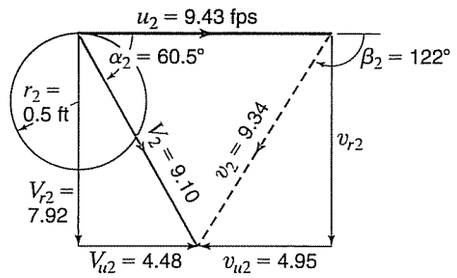
Point	r, ft	β	v_u, fps	v, fps	u, fps	
1	0.833	65°	2.13	5.03	15.71	***ANS***
2	0.5	122°	−4.95	9.34	9.43	***ANS***

Point	r, ft	V_u, fps	α	V, fps	
1	0.833	17.83	14.3°	18.41	***ANS***
2	0.5	4.48	60.5°	9.10	***ANS***

Entrance/outer periphery (Point 1):



Exit/inner periphery (Point 2):



(c) Torque

From Eq. (6.31): $T = \rho Q|\Delta(rV_u)| = \rho Q(r_1V_{u1} - r_2V_{u2})$

$= (62.4/32.2)7.5(0.83 \times 17.83 - 0.5 \times 4.48)$

$= 183.4$ ft·lb ***ANS***

Eqs. (5.38) and (5.40):

$P = T\omega/550 = 183.4(18.85)/550 = 6.28$ hp ***ANS***

EXERCISES

6.11.1 The absolute velocity of a jet of steam impinging on the blades of a steam turbine is 3800 fps, and that leaving is 2600 fps. $\alpha_1 = 20°$, $\alpha_2 = 150°$, $u_1 = u_2 =$ 500 fps, and $r_1 = r_2 = 0.5$ ft. Find the torque exerted on the rotor and the power delivered to it if the steam flows at 0.4 lb/sec.

6.11.2 The absolute velocity of a jet of steam impinging on the blades of a steam turbine is 1200 m/s, and that leaving is 950 m/s. $\alpha_1 = 20°$, $\alpha_2 = 150°$, $u_1 = u_2 =$ 180 m/s, and $r_1 = r_2 = 120$ mm. Find the torque exerted on the rotor and the power delivered to it if the steam flows at 2 N/s.

6.12 HEAD EQUIVALENT OF MECHANICAL WORK

If we multiply Eq. (6.31) for torque by angular velocity ω and compare the result with Eq. (5.38), we will see that the product represents the rate at which a pump delivers mechanical energy to the fluid or at which a turbine removes mechanical energy from the fluid. From Eqs. (5.38) and (5.39), power $= T\omega = \gamma Qh$. Replacing h by a specific value h_M (for machine) and noting that, when we multiply Eq. (6.31) by ω, $r_1\omega = u_1$ and $r_2\omega = u_2$, we have

$$T\omega = \gamma Q h_M = \rho Q(u_1V_1\cos\alpha_1 - u_2V_2\cos\alpha_2)$$

or

$$h_M = \frac{u_1V_1\cos\alpha_1 - u_2V_2\cos\alpha_2}{g} \tag{6.32}$$

which is the ***head utilized*** by a turbine (h_t) or, when h_M is negative, the ***head imparted*** to the fluid by the impeller of a pump (h_p).

If the value of h_M given by Eq. (6.32) is positive, it is the mechanical work done *by the fluid* on the vanes of a turbine runner per unit weight of fluid. If the value is negative, it is the mechanical work done on the fluid *by the impeller* of a pump or similar device per unit weight of fluid. Obviously, the work done by or on the fluid is equal to the loss or gain of energy, respectively, of the fluid.

SAMPLE PROBLEM 6.7 For the same turbine as Sample Prob. 6.6, find the head converted into mechanical work.

Solution

Eq. (6.32): $h_M = \left|\frac{\Delta(uV_u)}{g}\right| = \frac{u_1V_{u1} - u_2V_{u2}}{g}$

$= \frac{15.708 \times 17.833 - 9.425 \times 4.477}{32.2} = 7.39$ ft ***ANS***

Note: $P = \frac{\gamma Q h_M}{550} = \frac{62.4(7.5)7.39}{550} = 6.29$ hp

This agrees very well with Part (*c*) of Sample Prob. 6.6.

6.13 FLOW THROUGH A ROTATING CHANNEL

The passageway between the vanes of a turbine or pump is a channel that rotates as the flow passes through. We can write the usual energy equation (5.28) between entrance to and exit from such a passage that is itself rotating about an axis, but in addition to the friction loss h_L, in a turbine there is an additional loss h_M, due to the fact that the fluid is delivering mechanical work and losing energy

thereby. (If the passage is that of a pump, the numerical value of h_M will be negative, per Sec. 6.12.) Thus

$$H_1 - H_2 = h_L + h_M$$

or

$$\left(\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g}\right) - \left(\frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}\right) = h_L + \frac{u_1 V_1 \cos\alpha_1 - u_2 V_2 \cos\alpha_2}{g} \tag{6.33}$$

We recall that we derived h_M from Eq. (6.31) for torque, which we derived in turn from the momentum principle.

Wishing to eliminate angles from this equation, we note from Eq. (6.17) that

$$V\cos\alpha = u + v\cos\beta$$

Also, by trigonometry on the relative velocity triangle (Figs. 6.10 and 6.11), noting that β is an external angle of the u-v-V triangle and that $\cos(180° - \beta) = -\cos\beta$, we obtain

$$V^2 = v^2 + u^2 + 2vu\cos\beta$$

Inserting these values into Eq. (6.33) to eliminate α and β, it reduces to

$$\left(\frac{p_1}{\gamma} + z_1 + \frac{v_1^2 - u_1^2}{2g}\right) - \left(\frac{p_2}{\gamma} + z_2 + \frac{v_2^2 - u_2^2}{2g}\right) = h_L \tag{6.34}$$

Sometimes we call this equation the ***equation of relative velocities,*** because it replaces the absolute velocities of the energy equation by relative velocities.

If there is no flow, both v_1 and v_2 become zero and Eq. (6.34) reduces to that of a forced vortex (Eq. 5.52), since $u = \omega r$. If there is no rotation, both u_1 and u_2 become zero, the relative velocities become absolute velocities, and the equation becomes the usual energy equation. Because the frame of reference has changed, the mechanical work done does not appear as a separate term in Eq. (6.34).

EXERCISE

6.13.1 Develop Eq. (6.34) by making the substitutions indicated in the text.

6.14 REACTION WITH ROTATION

We found the force of reaction of a jet from a stationary body in Sec. 6.9 and that from a body in translation in Sec. 6.10. Since Sec. 6.13 developed the equation for the flow through a channel in rotation, we are now ready to consider the force of reaction of a fluid discharged from a rotating body.

A familiar object to illustrate this subject is the rotating lawn sprinkler. In Fig. 6.14 assume that the cross-sectional area of the arms is so large relative to

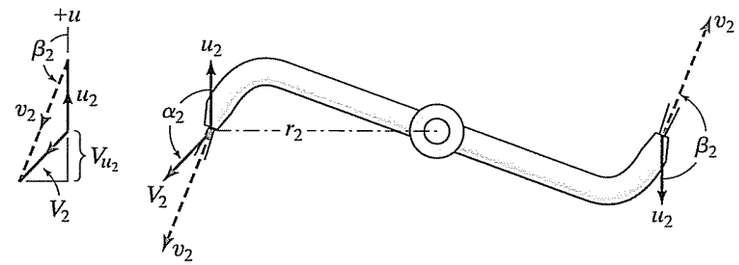


Figure 6.14

the area of the jets that we can neglect fluid-friction loss in the arms. Water enters at the center, where $r_1 = 0$, so that $u_1 = 0$ in Eq. (6.34). With the sprinkler arms lying in a horizontal plane, $z_1 - z_2 = 0$, and for the jets discharging into the air, p_2 is atmospheric pressure and we will regard it as zero. Since we neglect friction, $h_L = 0$, and if we let $h = p_1/\gamma + v_1^2/2g$ then Eq. (6.34) applied to Fig. 6.14 becomes

$$h - \left(\frac{v_2^2 - u_2^2}{2g}\right) = 0$$

or

$$v_2 = \sqrt{2gh + u_2^2} \tag{6.35}$$

where h is the sum of the pressure head and velocity head at entry to the sprinkler.

If A_2 denotes the sum of the areas of all the jets (two in Fig. 6.14), then $Q = A_2 v_2$. This, with Eq. (6.35), shows that the discharge increases with the rotative speed, since $u_2 = r_2\omega$.

From Eq. (6.17) for the relative velocity triangle in Fig. 6.14, the tangential component of the absolute velocity of discharge is $V_{u_2} = u_2 + v_2\cos\beta_2$, and hence the tangential component of the reaction force is

$$F_u = \frac{\gamma Q}{g}\Delta V_u = \frac{\gamma A_2 v_2}{g}(0 - V_{u_2}) = -\frac{\gamma A_2 v_2}{g}(u_2 + v_2\cos\beta_2)$$

As the radius is a factor in any rotating body, we usually prefer to compute torque rather than a force. In this case the torque is

$$T = F_u r_2 = -\frac{\gamma A_2 v_2}{g} r_2(u_2 + v_2\cos\beta_2) \tag{6.36}$$

The ideal maximum speed, or ***runaway speed,*** is when $T = 0$, and this will be the case when $u_2 = -v_2\cos\beta_2$ and when $V_2\cos\alpha_2 = 0$ or $\alpha_2 = 90°$. Because of mechanical friction, this condition will never occur. Of the total power supplied to the sprinkler, the greater part is lost in the kinetic energy of the jets. The total power *developed* by the sprinkler is used in overcoming friction in the bearings and air resistance. If there were more arms, with larger orifices, discharging more water, there could be a surplus of power, which would be useful power delivered. A primitive turbine constructed in this manner was called Barker's mill.

EXERCISES

6.14.1 The flow from a lawn sprinkler such as Fig. 6.14 is 140 L/min, $\beta_2 = 180°$, and the total area of the jets is 120 mm^2. The jets are located 200 mm from the center of rotation. Determine the speed of rotation if there is no friction.

6.14.2 A lawn sprinkler like that of Fig. 6.14 with $\beta_2 = 156°$ has a total jet area of 0.00086 ft^2 at a radius of 16 in. Compute the discharge rate, the torque exerted by the water, and the power developed, when $h = 120$ ft and the sprinkler is prevented from rotating.

6.14.3 Repeat Exer. 6.14.2 with the following changes: total jet area = 75 mm^2, radius = 360 mm, $h = 45$ m.

6.14.4 How fast would the sprinkler of Exer. 6.14.2 rotate if there were no mechanical friction or air resistance (i.e., consider the case of runaway speed, where $T = 0$)?

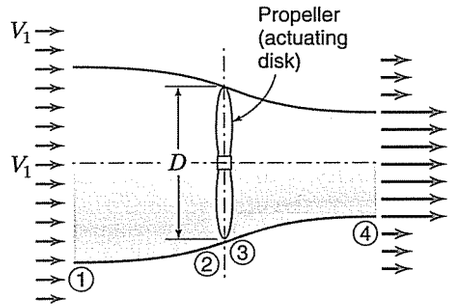
6.14.5 How fast would the sprinkler of Exer. 6.14.3 rotate if there were no mechanical friction or air resistance (i.e., consider the case of runaway speed, where $T = 0$)?

6.15 MOMENTUM PRINCIPLE APPLIED TO PROPELLERS AND WINDMILLS

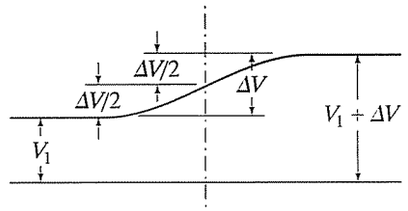
In the case of a fan in a duct, the cross section of the fluid passing through the fan is the same upstream as it is downstream, and the principal effect of a fan is to increase the pressure in the duct. In the case of a propeller revolving in free air, however, this is not so. The pressure must necessarily be the same at a distance either upstream or downstream from the propeller. How, then, may we consider the revolving blades to do work on the air? We may analyze this situation by considering the ***slipstream,*** or ***propeller race,*** which is nothing more than the body of air affected by the propeller (Fig. 6.15). The flow is undisturbed at section 1 upstream from the propeller, and is accelerated as it approaches the propeller. Additional increase in velocity occurs downstream of the propeller until it reaches a value of V_4 at section 4. Customarily we replace the propeller in simple slipstream theory with a stationary ***actuating disk*** across which the pressure rises, as we see in the pressure profile below the slipstream of Fig. 6.15 and also in Fig. 6.16. We thereby neglect the rotational effect of the propeller, together with the helical path of vortices shed from the blade tips (Sec. 9.8). The thrust force F_T is given by the pressure change at the disk times the area of the disk,

$$F_T = (p_3 - p_2)\frac{\pi D^2}{4} = \Delta p A \tag{6.37}$$

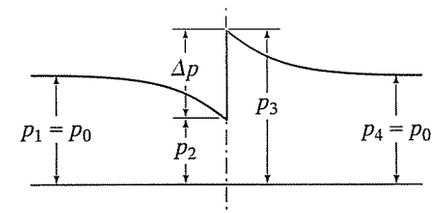
where D and A represent the diameter and area of the actuating disk, and p_2 and p_3 represent the pressures just upstream and downstream of the propeller, as shown in Figs. 6.15 and 6.16. Note that the pressures exerted on the boundary of the slipstream between sections 1 and 4 balance one another out and so we need not consider them.



(*a*) Slipstream



(*b*) Velocity profile



(*c*) Pressure profile

Figure 6.15
Effects of a propeller in free fluid. V_1 represents the velocity of the undisturbed fluid relative to the propeller; p_0 represents the undisturbed pressure in the fluid.

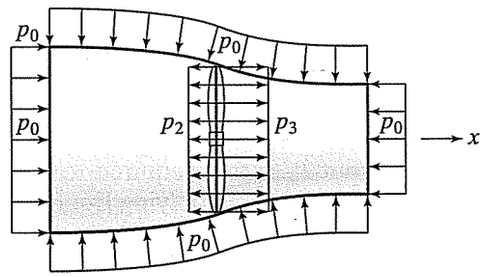


Figure 6.16
Forces acting on the fluid within the slipstream of Fig. 6.15.

By Newton's second law, the force F_T must equal the rate of change of momentum of the fluid that it acts on, in the control volume. If we let Q be the rate of flow through the slipstream,

$$F_T = \rho Q(\Delta V) = \rho A V(V_4 - V_1) \tag{6.38}$$

where V represents the mean velocity through the actuating disk, and V_1 and V_4 are the velocities in the slipstream at sections 1 and 4 of Fig. 6.15, where the pressures correspond to the normal undisturbed pressure p_0 in the flow field.

The propeller we are considering could be a stationary one like a fan or a moving one such as the propeller of a moving aircraft or ship. If it were a fan, V_1 would generally be equal to zero, and the slipstream upstream of the fan would have a much larger diameter than that shown in Fig. 6.15. If we were dealing with the propeller of a moving aircraft or ship, the craft would be moving to the left with a velocity V_1 through a stationary fluid, in which case Fig. 6.15 shows velocities relative to the craft.

Writing the Bernoulli equation (5.29) from a point upstream where the velocity is V_1 to a point downstream where it is $V_4 = V_1 + \Delta V$, recognizing that the pressure terms at these points cancel and (assuming an ideal fluid) that the disk adds $\Delta p/\gamma$ units of energy to the fluid per unit weight of fluid, we get

$$\frac{V_1^2}{2g} + \frac{\Delta p}{\gamma} = \frac{(V_1 + \Delta V)^2}{2g} \tag{6.39}$$

Equating Eqs. (6.37) and (6.38) and solving for Q in terms of Δp, and then eliminating Δp by using Eq. (6.39), gives

$$Q = A\left(V_1 + \frac{\Delta V}{2}\right) \tag{6.40}$$

As $\Delta V = V_4 - V_1$, we may express this as

$$Q = A\left(V_1 + \frac{V_4 - V_1}{2}\right) = A\left(\frac{V_1 + V_4}{2}\right) = AV$$

This shows that the velocity V at the disk is the average of the upstream and downstream velocities. It also shows that one-half of ΔV occurs upstream of the propeller, while the other half of ΔV occurs downstream.

Solving Eq. (6.39) for ΔV and substituting F_T/A for Δp from Eq. (6.37) gives

$$\Delta V = -V_1 + \sqrt{V_1^2 + \frac{2F_T}{A\rho}} \tag{6.41}$$

We may use the slipstream analysis to determine the maximum possible efficiency of a propeller. The power output P_{out} is given by

$$P_{out} = F_T V_1 = (\rho Q \Delta V) V_1$$

The power input P_{in} is that required to increase the velocity of the fluid in the slipstream from V_1 to V_4. Applying Eq. (5.39), we get

$$P_{in} = \gamma Q\left(\frac{V_4^2}{2g} - \frac{V_1^2}{2g}\right) = \frac{1}{2}\rho Q(V_4^2 - V_1^2)$$

$$= \rho Q\left(\frac{V_4 + V_1}{2}\right)(V_4 - V_1) = (\rho QV)(\Delta V) = F_T V$$

The overall efficiency η (eta) of the propeller is given by the ratio of the power output to the power input (Sec. 5.9). Thus

$$\eta = \frac{P_{out}}{P_{in}} = \frac{(\rho Q \Delta V)V_1}{(\rho QV)\Delta V} = \frac{V_1}{V} = \frac{V_1}{V_1 + \frac{1}{2}\Delta V} = \frac{1}{1 + \frac{1}{2}(\Delta V/V_1)} \tag{6.42}$$

We see that the efficiency is a function of the ratio $\Delta V/V_1$. The efficiency approaches 100% as ΔV approaches zero, but if $\Delta V = 0$, the propeller produces no force. The actual maximum efficiency of aircraft propellers is about 85%. However, the efficiency of an airplane propeller drops rapidly at speeds in excess of 400 mph (640 km/h) because of compressibility effects. For ships, maximum propeller efficiencies are only 60 or 70%.

Windmills are essentially the opposite of propellers because windmills *extract* energy from the wind. The slipstream for a windmill *expands* as it passes through the actuated disk, and the pressure drops, as does the velocity. By a procedure similar to that for a propeller, we can show that the maximum theoretical efficiency of a windmill is 59.3%. Because of friction and other losses, the actual efficiency of windmills rarely exceeds 40%.

SAMPLE PROBLEM 6.8 A total of 20,000 cfs of air (0.072 lb/ft^3) flows through two 6.5-ft-diameter propellers that are attached to an airplane moving at 150 mph through still air. Find (*a*) the total thrust and (*b*) the efficiency of the propellers. Also find (*c*) the pressure rise across the propellers and (*d*) the horsepower input to each propeller. Neglect eddy losses.

Solution

(*a*) The velocity of air relative to the airplane is

$$V_1 = 150 \text{ mph} = 150\left(\frac{44}{30}\right) = 220 \text{ fps}$$

The velocity of air through the actuating disk is

$$V = V_1 + \frac{\Delta V}{2} = \frac{Q}{A} = \frac{20{,}000/2}{(\pi/4)(6.5)^2} = 301 \text{ fps}$$

Thus $$\Delta V = 2(301 - 220) = 162.7 \text{ fps}$$

Eq. (6.38): $$F_T = \rho Q \Delta V = \frac{0.072}{32.2}(20{,}000)162.7$$

$$= 7280 \text{ lb (total thrust of both propellers)} \quad \textbf{\textit{ANS}}$$

(*b*) Eq. (6.42): $$\eta = \frac{1}{1 + \Delta V/2V_1} = \frac{1}{1 + 162/440} = 0.730 = 73\% \quad \textbf{\textit{ANS}}$$

(*c*) F_T on one propeller = 7280/2 = 3640 lb. But from Eq. (6.37) $F_T = \Delta p A$, thus $3640 = \Delta p(\pi/4)(6.5)^2$,

$$\Delta p = 109.6 \text{ psf} = 0.761 \text{ psi} \quad \textbf{\textit{ANS}}$$

(*d*) $$P_{in}/\text{propeller} = \frac{\gamma Q(\Delta p/\gamma)}{550} = \frac{Q\Delta p}{550} = \frac{10{,}000(109.6)}{550} = 1994 \text{ hp} \quad \textbf{\textit{ANS}}$$

Check: $$P_{in}/\text{propeller} = \frac{F_T V}{550} = \frac{3640(301)}{550} = 1994 \text{ hp}$$

EXERCISES

6.15.1 An 18-in-diameter household fan drives air ($\gamma = 0.076$ lb/ft^3) at a rate of 1.80 lb/sec. (*a*) Find the thrust exerted by the fan. (*b*) What is the pressure difference on the two sides of the fan? (*c*) Find the required horsepower to drive the fan. Neglect losses.

6.15.2 A 1.8-m-diameter fan drives air ($\gamma = 12$ N/m^3) at a rate of 50 N/s. (*a*) Find the thrust exerted by the fan. (*b*) What is the pressure difference on the two sides of the fan? (*c*) Find the required power to drive the fan. Neglect losses.

6.15.3 A fan sucks air from outside to inside a building through a 20-in-diameter duct. The density of the air is 0.0022 slug/ft^3 and the pressure difference across the two sides of the fan is 4.0 in of water. (*a*) What thrust must the fan support be designed to withstand? (*b*) Determine the flow rate of the air in cubic feet per second.

PROBLEMS

6.1 For two-dimensional laminar flow between two stationary parallel plates, find (*a*) the ratio of mean velocity to maximum velocity; (*b*) α; (*c*) β. The velocity profile is parabolic as in Sample Prob. 5.1.

6.2 In Sample Prob. 6.1 suppose the passage narrows down to a width of 9 ft at the second section. With the same depths, find the flow rate and the horizontal force on the structure.

6.3 In Sample Prob. 6.1 suppose the passage narrows down to a width of 2.5 m at the second section. With the same depths, find the flow rate and the horizontal force on the structure.

6.4 A hydraulic jump (Sec. 10.18) occurs in a transparent closed conduit with the diamond-shaped cross section shown in Fig. P6.4. The conduit is horizontal, and the water depth just upstream of the jump is 2.0 ft. The conduit is completely full of water downstream of the jump. Pressure-gage readings are as shown in the figure. (*a*) Compute the flow rate. Note that, because of turbulence in the jump, there is a substantial energy loss. So we cannot assume ideal flow. However, we may neglect shear forces along the boundary. (*b*) Determine the horsepower loss in the jump.

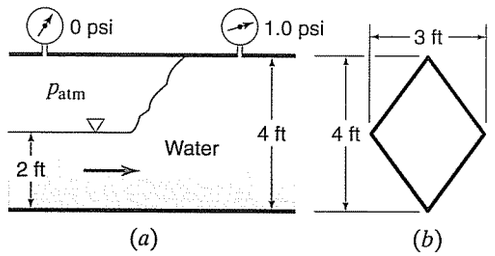


Figure P6.4

6.5 The diameters in Fig. 6.3 are 42 in and 30 in. At the larger end the pressure is 90 psi and the velocity is 12 fps. Neglecting friction, find the resultant force on the conical reducer if water flows (*a*) to the right; (*b*) to the left. (*c*) What would happen to the two previous answers if we did not neglect friction?

6.6 The diameters in Fig. 6.3 are 750 mm and 500 mm. At the larger end the pressure is 650 kPa and the velocity is 2.8 m/s. Neglecting friction, find the resultant force on the horizontal conical reducer if water flows (*a*) to the right; (*b*) to the left. (*c*) What would happen to the two previous answers if we did not neglect friction?

6.7 A reducing right-angled bend lies in a horizontal plane. Water enters from the west with a velocity of 6 fps and a pressure of 4 psi, and it leaves toward the north. The diameter at the entrance is 22 in and that at the exit is 20 in. Neglecting any friction loss, find the magnitude and direction of the resultant force on the bend.

6.8 A reducing right-angled bend lies in a horizontal plane. Water enters from the west with a velocity of 3 m/s and a pressure of 30 kPa, and it leaves toward the north. The diameter at the entrance is 500 mm and at the exit it is 400 mm. Neglecting any friction loss, find the magnitude and direction of the resultant force on the bend.

6.9 Both nozzle jets in Fig. P6.9 discharge horizontally into the atmosphere with a velocity of 40 fps. The liquid has a specific weight of 62.4 lb/ft^3. The axes of the pipe and both nozzles all lie in a horizontal plane. Find the magnitude and direction of the resultant force on this double nozzle while neglecting friction.

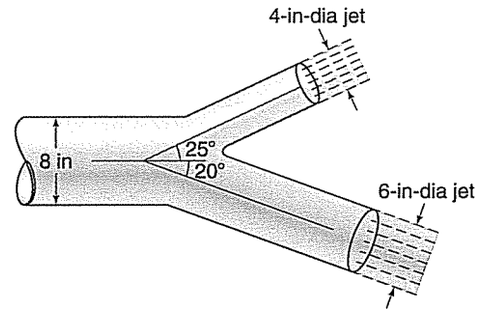


Figure P6.9

6.10 Assuming ideal flow, determine the total pull on the bolts in Fig. P6.10, where $y =$ 6 ft, $d_1 = 2$ in, $d_2 = 4$ in, $d_3 = 1$ in and γ_M of the manometer fluid (oil) is 52 pcf.

6.11 Repeat Prob. 6.10 for the case where $y =$ 1.80 m, $d_1 = 50$ mm, $d_2 = 100$ mm, $d_3 =$ 25 mm, and the manometer liquid has a specific gravity of 0.80.

6.12 If friction reduces the velocity V_2 of a jet of any fluid to $0.8V_1$ while deflecting through an angle θ, derive an equation for the dynamic force exerted in terms of $\dot{m}$, V_1, and θ.

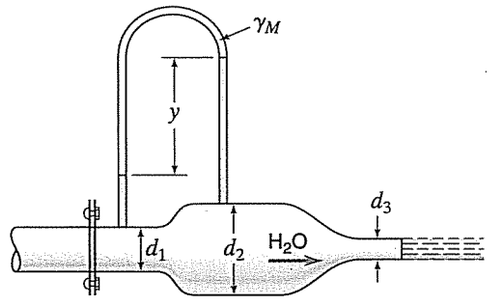


Figure P6.10

6.13 Solve Exer. 6.6.6, assuming that friction reduces V_2 to 80 fps.

6.14 Solve Exer. 6.6.7, assuming that friction reduces V_2 to 22 m/s.

6.15 Assuming ideal flow in a horizontal plane, calculate the magnitude and direction of the resultant force on the stationary blade in Fig. P6.15, knowing that $V_j = 50$ fps and $D_j = 6$ in. Note that the jet is divided by the splitter so that one-third of the water is diverted toward A.

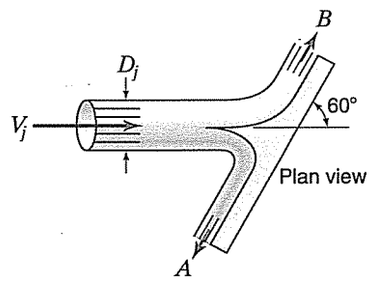


Figure P6.15

6.16 Refer to Fig. P6.15. Assuming ideal flow in the horizontal plane, calculate the magnitude and direction of the resultant force on the stationary blade. Note that the jet ($V_j = 12$ m/s, $D_j = 150$ mm) is divided by the splitter so that one-third of the water is diverted toward *A*.

6.17 A horizontal jet of water issues from an orifice in the side of a tank under a head h_1 and strikes a large plate a short distance away that covers the end of a horizontal tube in the side of a second tank (Fig. P6.17). The second tank contains oil of specific weight 50 lb/ft^3 at rest. The height of the oil above the tube is h_2. The jet diameter is three-fourths of the inside diameter of the tube. The jet and the tube are at the same elevation. (*a*) If the impact of the water is just sufficient to hold the plate in place, find the relation between h_1 and h_2. Neglect the weight of the plate and assume ideal flow. (*b*) Consider the effect of the weight of the plate. Find h_2 if $h_1 = 9$ ft, the weight of the plate = 8 lb, the jet diameter = 1.4 in, and the coefficient of friction between the plate and tube = 0.4. (*c*) Repeat part (*b*) with the plate weighing only 4 lb.

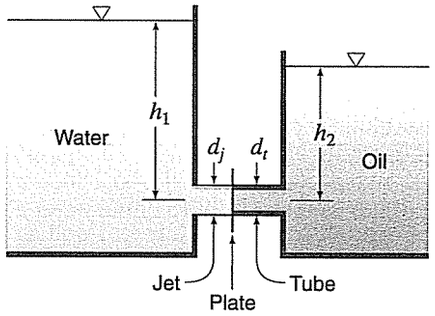


Figure P6.17

6.18 Repeat Prob. 6.17 with the following data: (*a*) oil specific weight = 7500 N/m^3, (*b*) h_1 = 2.8 m, W = 25 N, jet diameter = 35 mm, $\mu = 0.4$; (*c*) W = 18 N.

6.19 At section P a 5-in-diameter water jet with a velocity of 28 fps is directed vertically upward against the cone shown in Fig. P6.19. Neglecting friction and assuming the streamlines at Q are parallel, find the weight of the cone if $a = 1.5$ ft, $b = 0.6$ ft, and $c = 4$ ft.

6.20 Repeat Prob. 6.19 if the jet diameter is 0.6 m, the jet velocity is 22 m/s at P, and $a = 1.6$ m, $b = 0.6$ m, and $c = 3.5$ m.

6.21 Repeat Prob. 6.19 with a jet velocity of 19 fps, and $a = 0$, $b = 2$ ft, and $c = 4$ ft.

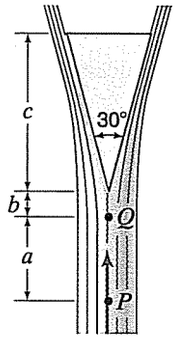


Figure P6.19

6.22 In Fig. P6.22 streamlines are plotted to scale in the plane of the center of a free jet impinging vertically on a horizontal circular plate. The jet diameter is 280 mm and the stagnation pressure at point O is 5.5 kPa. By scaling off the pertinent dimensions, determine as accurately as possible the velocity of the water as it leaves the plate and the total resultant force exerted by the water on the plate.

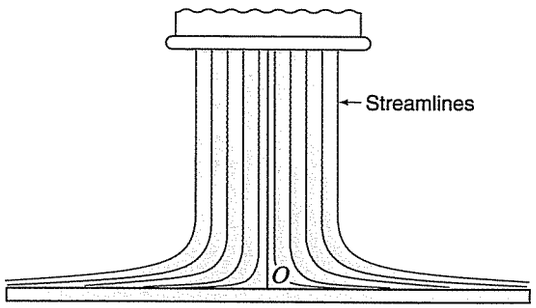


Figure P6.22

6.23 Repeat Prob. 6.22 with a jet diameter of 12 in and a jet velocity of 10 fps.

6.24 A locomotive tender running at 20 mph scoops up water from a trough between the rails, as shown in Fig. P6.24. The scoop delivers water at a height $h = 8$ ft above its original level and in the direction of motion. The area of the stream of water at entrance is 40 in^2. The water is everywhere under atmospheric pressure. Neglecting all losses, what is the absolute velocity of the water as it leaves the scoop? What force on the tender does the water cause? At what minimum speed will water rise to the height h above the original level?

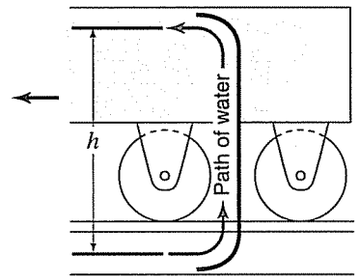


Figure P6.24

6.25 Solve Prob. 6.24 for the following data: locomotive speed = 12 m/s, h = 2.3 m, stream area = 0.03 m^2.

6.26 A 4-in-diameter water jet has a velocity of 130 fps. It strikes a single vane, which has an angle $\beta_2 = 90°$ and which is moving in the same direction as the jet with a velocity u. When u has values of 0, 45, 65, 85, 105, and 130 fps, find the values of (*a*) $\dot{m}'g$; (*b*) $V_2 \cos\alpha_2$; (*c*) ΔV_u; (*d*) Δv_u; (*e*) F_u. Assume $v_2 = 0.9v_1$. Present answers in a neat tabular form or on a spreadsheet.

6.27 Assume all the data in Prob. 6.26 are the same except that $\beta_2 = 180°$. Find the values of (*a*) $\dot{m}'g$; (*b*) v_2; (*c*) V_2; (*d*) ΔV; (*e*) Δv; (*f*) F_u. Assume $v_2 = 0.8v_1$. Present answers in a neat tabular form or on a spreadsheet.

6.28 Solve Prob. 6.27, for the case when $v_2 = 0.7v_1$.

6.29 Assume that all the data are the same as in Prob. 6.26, except that $\beta_2 = 145°$ and $v_2 = 0.7v_1$. Find the values of (*a*) $v_2 \cos\beta_2$; (*b*) $V_2 \cos\alpha_2$; (*c*) ΔV_u; (*d*) Δv_u; (*e*) F_u. Present answers in a neat tabular form or on a spreadsheet.

6.30 For the conditions of Prob. 6.15, compute the magnitude and direction of the resultant force on the single blade if it is moving to the right at a velocity of 20 fps.

6.31 For the conditions of Prob. 6.16, compute the magnitude and direction of the resultant force on the single blade if it is moving to the right at a velocity of 5 m/s.

6.32 Suppose the blade of Prob. 6.15 is one of a series of blades that are moving to the right at 15 fps. (*a*) Determine the resultant horizontal force on the blade system, and (*b*) compute the power transferred to the blades. (*c*) Compute the power of the jet and of the water leaving the blade system to verify an energy balance (see Sample Prob. 6.4).

6.33 Suppose the blade of Prob. 6.15 is one of a series of blades that are moving to the right at 4 m/s. (*a*) Determine the resultant horizontal force on the blade system, and (*b*) compute the power transferred to the blades. (*c*) Compute the power of the jet and of the water leaving the blade system to verify an energy balance (see Sample Prob. 6.4).

6.34 A 3-in-diameter air jet impinges on a series of blades, entering smoothly, and having absolute velocities $V_1 = 200$ fps and $V_2 = 150$ fps as shown in Fig. P6.34. Assume $\gamma = 0.075$ lb/ft^3, that the pressure is the same on both sides, and neglect friction. (*a*) What is the velocity of the blades and the power being transmitted to them? (*b*) Determine the blade angles necessary at entrance and exit.

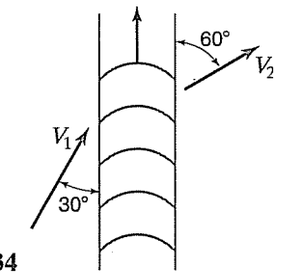


Figure P6.34

6.35 A 60-mm-diameter air jet impinges on a series of blades, entering smoothly, and having absolute velocities $V_1 = 60$ m/s and $V_2 = 45$ m/s as shown in Fig. P6.34. Assume $\gamma = 11$ N/m^3, that the pressure is the same on both sides, and neglect friction. (*a*) What is the velocity of the blades and the power being transmitted to them? (*b*) Determine the blade angles necessary at entrance and exit.

6.36 An ideal liquid ($\gamma = 52$ lb/ft^3) flows from a 2-ft-diameter tank as shown in Fig. P6.36. The jet diameter is 3 in and $a = 1$ ft. If the static coefficient of friction between the tank and floor is 0.52, determine the minimum value of h at which the tank will start to move to the left. The tank itself weighs 80 lb.

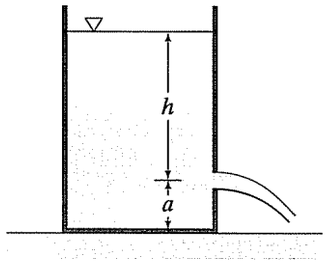


Figure P6.36

6.37 An ideal liquid ($\gamma = 9810$ N/m^3) flows from a 400-mm-diameter tank as shown in Fig. P6.36. The jet diameter is 80 mm and $a = 250$ mm. If the static coefficient of friction between the tank and the floor is 0.6, determine the minimum value of h at which the tank will start to move to the left. The tank itself weighs 500 N.

6.38 Find the magnitude and direction of the resultant force of the fluid on the compressor shown in Fig. P6.38. Air ($\gamma = 0.075$ lb/ft^3) enters at A through a 4-ft^2 area at a velocity of 15 fps. Air discharges at B through a 3-ft^2 area with a velocity of 17 fps.

6.39 Solve Prob. 6.38 for the case where a gas ($\gamma = 12.1$ N/m^3) enters at A through a 600-mm-diameter pipe at 5 m/s and leaves at B through a 500-mm-diameter pipe at 7 m/s.

6.40 A rocket has a propellant flow rate of 21.6 lb/sec through a nozzle with a throat area of 9.3 in^2. The nozzle expands the gases down to 14.7 psia at exit. The exit area of the nozzle is 48.5 in^2 and the exhaust velocity is 6370 fps. Find the rocket's thrust

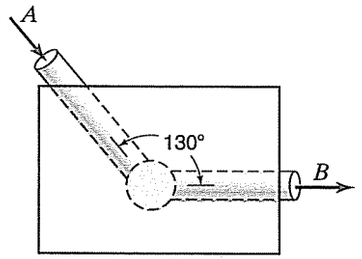


Figure P6.38

(a) at sea level and (b) at an elevation of 20,000 ft where the barometer pressure is 6.75 psia. (c) Find the specific weight of the exhaust gas.

6.41 A rocket has a propellant flow rate of 10 kg/s through a nozzle with a 90-mm-diameter throat. The nozzle is designed to expand the gases down to 101.3 kPa abs at exit. The exit diameter of the nozzle is 200 mm, and the exhaust velocity is 2100 m/s. Find the rocket's thrust (a) at sea level and (b) at an elevation of 6 km where the barometer pressure is 47.2 kPa abs; (c) find the density of the exhaust gas.

6.42 A radial-flow turbine has the following dimensions: $r_1 = 0.5$ m, $r_2 = 0.3$ m, $\beta_1 = 74°$, and $\beta_2 = 126°$. The width B of the flow passage between the two sides of the turbine is 0.2 m. When operating at 160 rpm, the flow rate through the turbine is 1.5 m^3/s. Find (a) the torque exerted by the water; (b) the power delivered to the shaft; (c) the head converted into mechanical work.

6.43 A radial-flow turbine has the following dimensions: $r_1 = 3.2$ ft, $r_2 = 1.6$ ft, $\beta_1 = 76°$, and $\beta_2 = 135°$. The width B of the flow passage between the two sides of the turbine is 0.65 ft. When operating at 150 rpm, the flow rate through the turbine is 60 cfs. Find (a) the torque exerted by the water; (b) the horsepower delivered to the shaft; (c) the head converted into mechanical work.

6.44 A paddle wheel with vanes that are all straight and radial is to be used as a crude centrifugal pump for water (Fig. P6.44): $r_1 = 3$ in, $r_2 = 9$ in, and the dimension B perpendicular to the plane of the figure is 0.2 ft. If the speed is 1200 rpm and the flow is 3380 gpm, (a) at the centerline elevation find the difference in pressure (psi) between the inner and outer circumferences, neglecting friction losses, (b) find which of these two points has the higher pressure, (c) compute the torque required to drive the pump, (d) calculate the horsepower requirement, and (e) verify that the horsepower requirement is equal to the difference between the horsepower of the outflow minus the horsepower of the inflow.

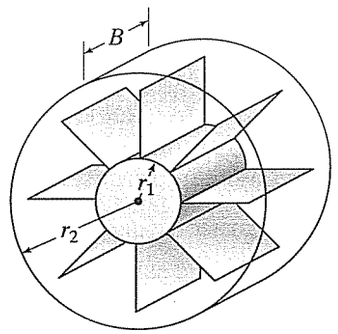


Figure P6.44

6.45 Repeat Prob. 6.44, where the data in SI units are as follows: $r_1 = 65$ mm, $r_2 = 215$ mm, and the dimension B perpendicular to the plane of figure = 48 mm. The speed is 1200 rpm and the flow is 150 L/s. Express pressure difference in kPa and power in kW.

6.46 Consider a lawn sprinkler such as that in Fig. 6.14 with $\beta_2 = 160°$, in which the total area of the jets at a radius of 15 in is 0.0008 ft^2. When $h = 144$ ft, compute the rate of discharge, the torque exerted by the water, and the power developed if the rotative speed of the sprinkler is 400 rpm. Neglect fluid friction, but note that the calculated torque must overcome mechanical friction and air resistance.

6.47 Repeat Prob. 6.46 with the following changes: radius = 400 mm, total jet area = 70 mm^2, $h = 42$ m.

6.48 For a lawn sprinkler like that of Fig. 6.14, develop an expression for the runaway speed ω in terms of h, r, and β_2. This would occur if there were no mechanical friction or air resistance, i.e., zero torque.

6.49 At what approximate speed will the sprinkler of Prob. 6.46 develop maximum horsepower?

6.50 By placing a 12-in electric fan on a frictionless mount it is observed to exert a thrust of 0.8 lb. (a) Find the approximate velocity of the slipstream of standard air (sea level) that it produces. (b) If 45% of the power supplied to the blades is lost in eddies and friction and if the driving motor has an efficiency of 60%, find the required electrical input in watts.

6.51 By placing a 300-mm electric fan on a frictionless mount it is observed to exert a thrust of 2.5 N. (a) Find the approximate velocity of the slipstream of standard air (sea level) that it produces. (b) If 45% of the power supplied to the blades is lost in eddies and friction and if the driving motor has an efficiency of 60%, find the required electrical input in watts.

6.52 Apply the momentum and energy principles to the case of a windmill (essentially the opposite of a propeller), to determine the maximum theoretical efficiency based on an input energy available from the wind velocity in a stream tube having a cross section equal to that of the windmill blade circle.

CHAPTER 7
Similitude and Dimensional Analysis

7.1 DEFINITION AND USES OF SIMILITUDE

Usually we find it impossible to determine all the essential facts for a given fluid flow by pure theory alone, and so we must often depend on experimental investigations. However, we can greatly reduce the number of tests needed by systematically using dimensional analysis and the laws of similitude or similarity. For these enable us to apply test data to other cases than those observed.

The similarity laws enable us to experiment with a convenient fluid such as water or air, for example, and then apply the results to a fluid that is less convenient to work with, such as hydrogen, steam, or oil. Also, in both hydraulics and aeronautics, we can obtain valuable results at a minimum cost from tests made with small-scale models of the full-size apparatus. The laws of similitude enable us to predict the performance of the ***prototype,*** which means the full-size device, from tests made with the model. We need not use the same fluid for the model and its prototype. Neither must the model necessarily be smaller than its prototype. So, for example, we might study the flow in a carburetor in a very large model. And we might investigate the flow of water at the entrance to a small centrifugal-pump runner by studying the flow of air at the entrance to a large model of the runner.

A few examples of where we have used models are ships in towing basins, airplanes in wind tunnels, hydraulic turbines, centrifugal pumps, spillways of dams, river channels, and the study of such phenomena as the action of waves and tides on beaches, soil erosion and the transport of sediment.

We should emphasize that the model size need not necessarily be different from the prototype. In fact, it may be the same device, the variables in this case being the velocity and the physical properties of the fluid.

7.2 GEOMETRIC SIMILARITY

One of the desirable features in model studies is that we have ***geometric similarity,*** which means that the model and its prototype have identical shapes but differ only in *size*. The important consideration is that the flow patterns must be geometrically similar. If subscripts p and m denote prototype and model,

respectively, we define the ***length scale ratio***[1] as

$$L_r = \frac{L_p}{L_m} \tag{7.1}$$

the ratio of the linear dimensions of the prototype to, or divided by, the corresponding dimensions in the model. It follows that areas vary as L_r^2 and volumes as L_r^3. Complete geometric similarity is not always easy to attain. For example, we may not be able to reduce the surface roughness of a small model in proportion unless we can make its surface very much smoother than that of the prototype. Similarly, in the study of sediment transport, we may not be able to scale down the bed materials without having material so fine as to be impractical. Fine powder, because of cohesive forces between the particles, does not simulate the behavior of sand. Again, in the case of a river the horizontal scale is usually limited by the available floor space, and this same scale used for the vertical dimensions may produce a stream so shallow that capillarity has an appreciable effect and also the bed slope may be so small that the flow is laminar. In such cases we need to use a distorted model, which means that the vertical scale is larger than the horizontal scale. Then, if the horizontal scale ratio is denoted by L_r and the vertical scale ratio by $L_{r'}$, the cross section area ratio is $L_r L_{r'}$.

7.3 KINEMATIC SIMILARITY

Kinematic similarity implies that, *in addition* to geometric similarity, the ratio of the *velocities* at all corresponding points in the flows are the same. The ***velocity scale ratio*** is

$$V_r = \frac{V_p}{V_m} \tag{7.2}$$

and this is a constant for kinematic similarity. Its value in terms of L_r is determined by dynamic considerations, as explained in the following section.

As time T is dimensionally L/V, the ***time scale ratio*** is

$$T_r = \frac{L_r}{V_r} \tag{7.3}$$

and in a similar manner the ***acceleration scale ratio*** is

$$a_r = \frac{L_r}{T_r^2} = \frac{V_r^2}{L_r} \tag{7.4}$$

[1] The reciprocal of the length scale ratio we will refer to here as the ***model ratio,*** or model scale, λ (lambda) $= L_m/L_p$. Thus a model ratio of 1:20 or $\lambda = 0.05$ corresponds to a length scale ratio of 20:1 or $L_r = 20$ (the prototype is 20 times larger than the model).

7.4 DYNAMIC SIMILARITY

Two systems have ***dynamic similarity*** if, *in addition to* kinematic similarity, corresponding forces are in the same ratio in both. The ***force scale ratio*** is

$$F_r = \frac{F_p}{F_m} \tag{7.5}$$

which must be constant for dynamic similarity.

Forces that may act on a fluid element include those due to gravity (F_G), pressure (F_P), viscosity (F_V), and elasticity (F_E). Also, if the element of fluid is at a liquid-gas interface, there are forces due to surface tension (F_T). If the sum of forces on a fluid element does not add up to zero, the element will accelerate in accordance with Newton's law. We can transform such an unbalanced force system into a balanced system by adding an inertia force F_I that is equal and opposite to the resultant of the acting forces. Thus, generally,

$$\sum \mathbf{F} = \mathbf{F}_G + \mathbf{F}_P + \mathbf{F}_V + \mathbf{F}_E + \mathbf{F}_T = \text{Resultant}$$

and

$$\mathbf{F}_I = -\text{Resultant}$$

Thus

$$\mathbf{F}_G + \mathbf{F}_P + \mathbf{F}_V + \mathbf{F}_E + \mathbf{F}_T + \mathbf{F}_I = 0$$

These forces can be expressed in the simplest terms as:

Gravity: $F_G = mg = \rho L^3 g$

Pressure: $F_P = (\Delta p)A = (\Delta p)L^2$

Viscosity: $F_V = \mu\left(\frac{du}{dy}\right)A = \mu\left(\frac{V}{L}\right)L^2 = \mu VL$

Elasticity: $F_E = E_v A = E_v L^2$

Surface tension: $F_T = \sigma L$

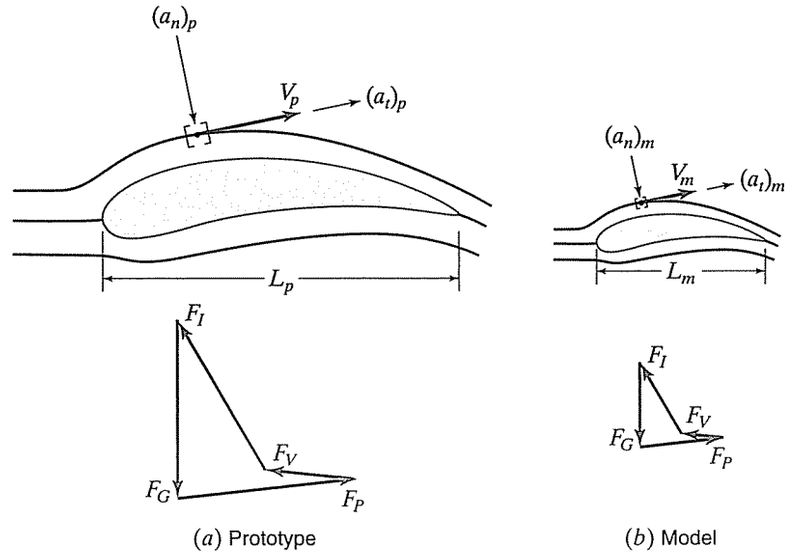
Inertia: $F_I = ma = \rho L^3 \frac{L}{T^2} = \rho L^4 T^{-2} = \rho V^2 L^2$

In many flow problems some of these forces are either absent or insignificant. In Fig. 7.1 we see two geometrically similar flow systems. Let us assume that they also possess kinematic similarity, and that the forces acting on any fluid element are F_G, F_P, F_V, and F_I. Then we will have dynamic similarity if

$$\frac{F_{G_p}}{F_{G_m}} = \frac{F_{P_p}}{F_{P_m}} = \frac{F_{V_p}}{F_{V_m}} = \frac{F_{I_p}}{F_{I_m}} = F_r$$

where subscripts p and m refer to prototype and model as before. We can also express these relations as

$$\left(\frac{F_I}{F_G}\right)_p = \left(\frac{F_I}{F_G}\right)_m, \quad \left(\frac{F_I}{F_P}\right)_p = \left(\frac{F_I}{F_P}\right)_m, \quad \left(\frac{F_I}{F_V}\right)_p = \left(\frac{F_I}{F_V}\right)_m$$



(*a*) Prototype (*b*) Model

Figure 7.1
Two flow systems, with geometric similarity ($L_r = L_p/L_m$), kinematic similarity ($V_r = V_p/V_m$), and dynamic similarity ($F_r = F_p/F_m$).

Each of the quantities is dimensionless. With four forces acting, we must satisfy three independent expressions; with three forces, we must satisfy two independent expressions; and so on. We shall discuss the significance of the various dimensionless ratios in the following subsections.

Reynolds Number[2]

In the flow of a fluid through a completely filled conduit, gravity does not affect the flow pattern. Also, since there are no free liquid surfaces, capillarity is obviously of no practical importance. Therefore the significant forces are inertia and fluid friction due to viscosity. The same is true of an airplane traveling at speeds below that at which much air compression occurs. Also, for a submarine submerged far enough that it does not produce waves on the surface, the only forces involved are those of friction and inertia.

For the ratio of *inertia forces to viscous forces,* we call the resulting parameter the ***Reynolds number,*** or **R**, in honor of Osborne Reynolds (1842–1912), the English physicist and professor who presented this in a publication of his

[2] It is now becoming standard practice to represent Reynolds number, Froude number, etc., by bold face **R**, **F**, etc., as this simplifies their use with subscripts. For handwriting on the blackboard and overhead projector, we suggest using $\mathbb{R}$ and $\mathbb{F}$, etc., with double lines on the left. The symbols *Re* and N_R are also sometimes used for Reynolds number, and *Fr* and N_F for Froude number, etc.

experimental work in 1882. But it was Lord Rayleigh (1842–1919), another English physicist and a Nobel Laureate, who 10 years later developed the theory of dynamic similarity. The ratio of these two forces is

$$\mathbf{R} = \frac{F_I}{F_V} = \frac{L^2V^2\rho}{LV\mu} = \frac{LV\rho}{\mu} = \frac{LV}{\nu} \tag{7.6}$$

For any consistent system of units, **R** is a dimensionless number, which turns out to be useful for comparing different flows. The linear dimension L may be any length that is significant in the flow pattern. Thus, for a pipe completely filled, it might be either the diameter or the radius, and the numerical value of **R** will vary accordingly. Most commonly we use the pipe diameter for L. Thus, for a pipe flowing full, $\mathbf{R} = DV\rho/\mu = DV/\nu$, where D is the diameter of the pipe.

If two systems, such as a model and its prototype, or two pipelines with different fluids, are dynamically equivalent so far as inertia and viscous forces are concerned, they must both have the same value of **R**. Thus, for such cases, we will have dynamic similarity when

$$\left(\frac{LV}{\nu}\right)_m = \mathbf{R}_m = \mathbf{R}_p = \left(\frac{LV}{\nu}\right)_p \tag{7.7}$$

For the same fluid in the model and the prototype, Eq. (7.7) shows that for dynamic similarity, we must have a high velocity with a model of small linear dimensions. The fluid used in the model need not be the same as that in the prototype, provided L and V are chosen so that they give the same value of **R** in model and prototype.

SAMPLE PROBLEM 7.1 If the Reynolds numbers of a model and its prototypes are the same, find expressions for the scale ratios V_r, T_r, and a_r.

Solution

$$\mathbf{R} = \frac{L_mV_m}{\nu_m} = \frac{L_pV_p}{\nu_p}$$

$$V_r = \frac{V_p}{V_m} = \frac{L_m\nu_p}{L_p\nu_m} = \frac{\nu_r}{L_r} = \left(\frac{\nu}{L}\right)_r \qquad \textit{ANS}$$

$$T_r = \frac{L_r}{V_r} = L_r\left(\frac{L}{\nu}\right)_r = \left(\frac{L^2}{\nu}\right)_r \qquad \textit{ANS}$$

$$a_r = \frac{V_r}{T_r} = \left(\frac{\nu}{L}\right)_r\left(\frac{\nu}{L^2}\right)_r = \left(\frac{\nu^2}{L^3}\right)_r \qquad \textit{ANS}$$

Froude Number[3]

When we consider *inertia and gravity forces* alone, we obtain a ratio called a ***Froude number,*** or **F**. This dimensionless number was named to honor William Froude (1810–1879), a British naval architect who experimented with flat plates towed lengthwise through water in order to estimate the resistance of ships due to wave action. The ratio of inertia forces to gravity forces is

$$\frac{\rho L^2V^2}{\rho gL^3} = \frac{V^2}{gL}$$

Although this is sometimes defined as a Froude number, it is more common to use the square root so that V is in the first power, as in the Reynolds number. Thus a Froude number is

$$\mathbf{F} = \frac{V}{\sqrt{gL}} \tag{7.8}$$

Systems involving gravity and inertia forces include the wave action set up by a ship, the flow of water in open channels, the forces of a stream on a bridge pier, the flow over a spillway, the flow of a jet from an orifice, and other cases where gravity is the dominant factor.

To compute **F**, the length L must be some linear dimension that is significant in the flow pattern. For a ship, we commonly take this as the length at the waterline. For an open channel, we take it as the depth of flow. For situations where inertia and gravity forces predominate, dynamic similarity occurs when

$$\left(\frac{V}{\sqrt{gL}}\right)_m = \mathbf{F}_m = \mathbf{F}_p = \left(\frac{V}{\sqrt{gL}}\right)_p \tag{7.9}$$

In some flow situations fluid friction is a factor as well as gravity and inertia. In such cases, to achieve dynamic similarity, we must satisfy both the Reynolds number and the Froude number criteria simultaneously. By comparing Eqs. (7.7) and (7.9) it is clear that we cannot satisfy them both at the same time with fluids of the same viscosity. The only way to satisfy Eqs. (7.7) and (7.9) for both model and prototype is to use fluids of different viscosities. Simultaneous solution of Eqs. (7.7) and (7.9) yields $(\nu_m/\nu_p) = (L_m/L_p)^{3/2}$. Sometimes a fluid with the proper viscosity can be found, but usually this is either impractical or impossible. As a result, the usual technique is to operate the model so as to satisfy one of the dimensionless numbers and to then correct the results with experimental data dependent on the other dimensionless number. In the case of a ship, towing a model will give the total resistance, from which we must subtract the empirically-computed skin friction to determine the wave-making resistance of the model. Then by using the Froude number criterion we can determine the wave-making resistance of the full-size ship. Last we add a computed

[3] Froude is pronounced (frood), to rhyme with brood.

skin friction for the ship to obtain the total ship resistance. The details of such calculations are deferred to Chap. 9.

When water flows in an open channel, fluid friction as well as gravity and inertia may be a factor. However, in such flows turbulence is often fully developed, in which case the hydraulic friction is independent of the Reynolds number (Sec. 8.12). So, for this case, identical Froude numbers will give dynamic similarity. When a high-viscosity liquid flows in an open channel or when water flows at a relatively low Reynolds number, the effect of Reynolds number cannot be neglected.

Let us now discuss the scale ratios for Froude number similarity. From Eq. (7.8), V varies as $\sqrt{gL}$, and if we consider g to be the same in prototype and model, as is usually the case, then, from Eq. (7.2),

$$V_r = \frac{V_p}{V_m} = \sqrt{\frac{L_r}{1}} \quad \text{(for the same } \mathbf{F} \text{ and same } g\text{)}$$

and, from Eq. (7.3), the scale ratio of time for prototype to model is

$$T_r = \frac{T_p}{T_m} = \frac{L_r}{V_r} = \sqrt{\frac{L_r}{1}} \quad \text{(for the same } \mathbf{F} \text{ and same } g\text{)}$$

and

$$a_r = \frac{V_r}{T_r} = 1 \quad \text{(for the same } \mathbf{F} \text{ and same } g\text{)}$$

A knowledge of the time scale ratio helps us when studying cyclic phenomena such as waves and tides.

Since the velocity varies as $\sqrt{L_r}$ and the cross-sectional area as L_r^2, it follows that the discharge ratio

$$Q_r = \frac{Q_p}{Q_m} = \frac{L_r^{5/2}}{1} \quad \text{(for the same } \mathbf{F} \text{ and same } g\text{)}$$

As mentioned in Sec. 7.2, for river models we usually need to use an enlarged vertical scale to provide the model with adequate depth.[4] From the Froude number we find in this case that V varies as $\sqrt{L_{r'}}$, where $L_{r'}$ is the length scale ratio in the vertical direction. Thus

$$\frac{Q_p}{Q_m} = \frac{L_r L_{r'}^{3/2}}{1} \quad \text{(for the same } \mathbf{F} \text{ and same } g\text{)}$$

Mach Number

When compressibility is important, we need to consider the ratio of the *inertia to the elastic forces.* The ***Mach number*** **M** is defined as the square root of this ratio. It is named after Ernst Mach (1838–1916), an Austrian physicist and philosopher who investigated the shock waves of supersonic projectiles in the 1880s. Thus

$$\mathbf{M} = \left(\frac{\rho V^2 L^2}{E_v L^2}\right)^{1/2} = \frac{V}{\sqrt{E_v/\rho}} = \frac{V}{c} \tag{7.10}$$

where c is the sonic velocity (or celerity) in the medium in question (see Sec. 13.3). So the Mach number is the ratio of the fluid velocity (or the velocity of the body through a stationary fluid) to that of a sound wave in the same medium. If **M** is less than 1, the flow is ***subsonic;*** if it is equal to 1, it is ***sonic;*** if it is greater than 1, the flow is called ***supersonic;*** and for extremely high values of **M**, the flow is called ***hypersonic.*** The Mach number squared is equal to the Cauchy number.

Weber Number

Surface tension may be important in a few cases of flow, but usually it is negligible. The ratio of *inertia forces to surface tension* is $\rho V^2 L^2/\sigma L$, the square root of which is known as the ***Weber number:***

$$\mathbf{W} = \frac{V}{\sqrt{\sigma/\rho L}} \tag{7.11}$$

It is named in honor of Moritz Weber (1871–1951), who developed the modern laws of similitude. An example of an application of the Weber number is at the leading edge of a very thin sheet of liquid flowing over a surface.

Euler Number[5]

A dimensionless quantity related to the ratio of the *inertia forces to the pressure forces* is known as the ***Euler number.*** It is named after the Swiss mathematician Leonhard Euler (1707–1783), renowned for his prolific work in pure mathematics. It is expressed in a variety of ways, one form being

$$\mathbf{E} = \frac{V}{\sqrt{2(\Delta p/\rho)}} = \frac{V}{\sqrt{2g(\Delta p/\gamma)}} \tag{7.12}$$

If only pressure and inertia influence a flow, the Euler number for any boundary form will remain constant. If other parameters (viscosity, gravity, etc.) cause the flow pattern to change, however, **E** will also change. We can see that this expression for **E** is equivalent to the coefficient of velocity, discussed in Chap. 11.

Other Dimensionless Numbers

Rearranging Eq. (7.12) to evaluate $1/\mathbf{E}^2$, we get a dimensionless quantity known as the ***pressure coefficient:***

$$C_p = \frac{\Delta p}{\frac{1}{2}\rho V^2} = \frac{1}{\mathbf{E}^2} \tag{7.13}$$

[4] We also often use enlarged vertical scales in models of large water bodies such as lakes, reservoirs, estuaries, and bays.

[5] Euler is pronounced (oi'lər), to rhyme with boiler.

This, or half this quantity, is also called the Euler number by some authors; we noted in Sec. 5.4 that some call $\frac{1}{2}\rho V^2$ the dynamic pressure. When Δp is referred to the vapor pressure p_v, the pressure coefficient becomes a dimensionless quantity called the ***cavitation number:***

$$\mathbf{C} = \frac{p - p_v}{\frac{1}{2}\rho V^2} \tag{7.14}$$

Here both pressures must be absolute. Some authors omit the $\frac{1}{2}$ from this definition.

The seven dimensionless numbers just named are among the best known and most used. We may use some others frequently without thinking of them as dimensionless numbers; these include the specific heat ratio (Sec. 2.7), the friction factor (Sec. 8.5), and drag and lift coefficients (Secs. 9.7 and 9.12). There are a great many others, related to the many different areas of fluid mechanics; Ref. 49 (Appendix E) lists more than 150.

SAMPLE PROBLEM 7.2 A submerged body is to move horizontally through oil ($\gamma = 52$ lb/ft^3, $\mu = 0.0006$ lb·sec/ft^2) at a velocity of 45 fps. To study the characteristics of this motion, an enlarged model of the body is tested in 60°F water. The model ratio λ is 8:1. Determine the velocity at which this enlarged model should travel through the water to achieve dynamic similarity. If the drag force on the model is 0.80 lb, predict the drag force on the prototype.

Solution

The body is submerged; so there is no wave action. Reynolds' criterion must be satisfied:

Table A.1 for water at 60°F: $\nu_m = 12.17 \times 10^{-6}$ ft^2/sec.

Eq. (2.11): $$\nu_p = \left(\frac{\mu}{\rho}\right)_p = \left(\frac{\mu}{\gamma/g}\right)_p = \frac{0.0006}{52/32.2} = 0.000372 \text{ ft}^2/\text{sec}$$

From Eq. (7.7): $$\left(\frac{DV}{\nu}\right)_p = \left(\frac{DV}{\nu}\right)_m, \quad \text{where} \quad \frac{D_m}{D_p} = \frac{8}{1}$$

$$\frac{D_p(45)}{0.000372} = \frac{(8D_p)V_m}{12.17 \times 10^{-6}}$$

$$V_m = 0.1843 \text{ fps} \qquad \textbf{\textit{ANS}}$$

$$F \propto \rho V^2 L^2; \quad \text{therefore}$$

$$\frac{F_p}{F_m} = \frac{\rho_p V_p^2 L_p^2}{\rho_m V_m^2 L_m^2} = \frac{(52/32.2)(45)^2 1}{1.938(0.1843)^2(8)^2} = 777$$

$$F_p = 777 F_m = 777(0.8) = 621 \text{ lb} \qquad \textbf{\textit{ANS}}$$

EXERCISES

7.4.1 Water at 65°F in an 8-in-diameter pipe flows with a velocity of 5 fps. What is the Reynolds number? Note that in the Reynolds number the significant length $L = D$, and that Appendix A gives physical properties of water.

7.4.2 Oil ($s = 0.85$ and $\mu = 0.24$ N·s/m^2) in a 100-mm-diameter pipe flows with a velocity of 3.5 m/s. What is the Reynolds number? Note that in the Reynolds number the significant length $L = D$.

7.4.3 A vertical jet of water issuing upward from a nozzle at a velocity of 76 fps will rise to a height of approximately 90 ft on the earth. To get a water jet to rise to a height of 120 ft on the moon, where the gravity is one-sixth of that on earth, what must be the jet velocity? Neglect atmospheric resistance.

7.4.4 A wind tunnel test on a 1:40 scale model of a submarine is planned to find the drag on the submarine when it is moving at 10 knots through 50°F ocean water. What should the test velocity be in the wind tunnel if it contains 60°F air at atmospheric pressure? If the drag on the model is 75 lb, what will the drag on the prototype be?

7.4.5 A 500-ft-long ship is to operate at a speed of 20 mph. If a model of this ship is 10 ft long, what should its speed be in fps to give the same Froude number? What is the value of this Froude number?

7.4.6 The dimensions of a model airplane are $\frac{1}{30}$ those of its prototype. In planned tests in a pressure wind tunnel the model will operate at the same speed and air temperature as the prototype. What pressure in the wind tunnel, relative to the atmospheric pressure, will make the Reynolds number the same?

7.4.7 Water flows over the crest of a 1:35 model spillway, and the velocity measured at a particular point is 0.46 m/s. What velocity does this represent in the prototype? The force exerted on a certain area of the model is measured to be 0.12 N. What would be the force on the corresponding area in the prototype? Develop your own dimensionless ratios.

7.4.8 A certain missile travels at 2500 km/h through the atmosphere at an elevation of 5 km. Air at 20°C and standard atmospheric conditions will flow around a model of the missile in a wind tunnel. What air speed in the wind tunnel will achieve dynamic similarity?

7.5 SCALE RATIOS

The Reynolds number, the Froude number, and the Mach number are the dimensionless parameters most commonly encountered in fluid mechanics. In the preceding section we developed the scale ratios for velocity, time, and acceleration for the Reynolds and Froude numbers. We can develop scale ratios for other quantities in a similar fashion. Table 7.1 presents such relations. These enable us to quickly calculate the scale ratio (prototype divided by model) of any desired quantity whenever the given dimensionless number is the same in both prototype and model. The computed ratio, of course, gives a

TABLE 7.1 **Flow characteristics and similitude scale ratios (ratio of prototype quantity to model quantity)**

Characteristic	Dimension	Scale ratios for laws of		
		Reynolds	**Froude**	**Mach**
Geometric				
Length	L	L_r	L_r	L_r
Area	L^2	L_r^2	L_r^2	L_r^2
Volume	L^3	L_r^3	L_r^3	L_r^3
Kinematic				
Time	T	$\left(\frac{L^2\rho}{\mu}\right)_r$	$(L^{1/2}g^{-1/2})_r$	$\left(\frac{L\rho^{1/2}}{E_v^{1/2}}\right)_r$
Velocity	LT^{-1}	$\left(\frac{\mu}{L\rho}\right)_r$	$(L^{1/2}g^{1/2})_r$	$\left(\frac{E_v^{1/2}}{\rho^{1/2}}\right)_r$
Acceleration	LT^{-2}	$\left(\frac{\mu^2}{\rho^2L^3}\right)_r$	g_r	$\left(\frac{E_v}{L\rho}\right)_r$
Discharge	L^3T^{-1}	$\left(\frac{L\mu}{\rho}\right)_r$	$(L^{5/2}g^{1/2})_r$	$\left(\frac{L^2E_v^{1/2}}{\rho^{1/2}}\right)_r$
Dynamic				
Mass	M	$(L^3\rho)_r$	$(L^3\rho)_r$	$(L^3\rho)_r$
Force	MLT^{-2}	$\left(\frac{\mu^2}{\rho}\right)_r$	$(L^3\rho g)_r$	$(L^2E_v)_r$
Pressure	$ML^{-1}T^{-2}$	$\left(\frac{\mu^2}{L^2\rho}\right)_r$	$(L\rho g)_r$	$(E_v)_r$
Impulse and momentum	MLT^{-1}	$(L^2\mu)_r$	$(L^{7/2}\rho g^{1/2})_r$	$(L^3\rho^{1/2}E_v^{1/2})_r$
Energy and work	ML^2T^{-2}	$\left(\frac{L\mu^2}{\rho}\right)_r$	$(L^4\rho g)_r$	$(L^3E_v)_r$
Power	ML^2T^{-3}	$\left(\frac{\mu^3}{L\rho^2}\right)_r$	$(L^{7/2}\rho g^{3/2})_r$	$\left(\frac{L^2E_v^{3/2}}{\rho^{1/2}}\right)_r$

Note: Usually g is the same in model and prototype.

realistic result only if the flow is predominantly governed by that particular dimensionless number. Thus an important aspect of physical modeling of fluid phenomena is our ability to identify which dimensionless number is most important; this is discussed in Sec. 7.4 under the various dimensionless numbers.

7.6 COMMENTS ON MODELS

When using models the fluid velocity should never be so low that flow is laminar when the flow in the prototype is turbulent. Also, conditions in the model should not allow surface tension to be important if it is not in the prototype. For example, the depth of water flowing over the crest of a model spillway should not be too small.

While model studies are very important and valuable, we must exercise proper judgment when transferring results from the model to the prototype. It is not always necessary or desirable to adhere to these various dimensionless ratios in every case. Thus, in tests of model centrifugal pumps, geometric similarity is essential, but we need to operate them at such rotation speeds that the peripheral velocity and all fluid velocities are the same as in the prototype, because this is the only way we can detect cavitation.

We should scale down the roughness of a model in the same ratio as the other linear dimensions, which means that a small model should have surfaces that are much smoother than those in its prototype. But this requirement will impose a limit on the scale that we can use if we are to achieve true geometric similarity. However, in the case of a distorted model with a vertical scale larger than the horizontal scale, we may need to make the model surface rough in order to simulate the flow conditions in the prototype. As any distorted model lacks the proper similitude, there is no simple rule for this; we must adjust the roughness by trial until we judge the flow conditions are typical of those in the prototype. Most distorted models have metal tabs (Fig. 7.2) to provide proper frictional boundary effects. Workers determine the size and spacing of the tabs by trial so as to create flow conditions in the model identical to those observed in the prototype. Vertical distortion disturbs circulation patterns, and the metal tabs create large-scale eddies. Hence we must interpret mixing (the disposition of pollutants) in distorted models with caution.

In models of systems, such as siphons, involving liquids where we expect large negative pressures and so cavitation could occur, we must place the model in an air-tight chamber which maintains a partial vacuum that produces the same absolute pressure in the model as in the prototype.[6]

When modeling a subsonic airplane in a wind tunnel, we commonly need to conduct the test under high pressure in order to satisfy the Reynolds criterion

$$\left(\frac{DV\rho}{\mu}\right)_m = \left(\frac{DV\rho}{\mu}\right)_p$$

without introducing compressibility effects. For example, suppose $L_r = D_p/D_m = 20$. If the viscosity μ and density ρ of the air were the same in the model and prototype then, to satisfy Reynolds' criterion, $V_m = 20 \times V_p$. For an airplane

[6] Hydraulic Modeling: Concepts and Practice, *Manual and Report No. 97,* American Society of Civil Engineers, 2000.

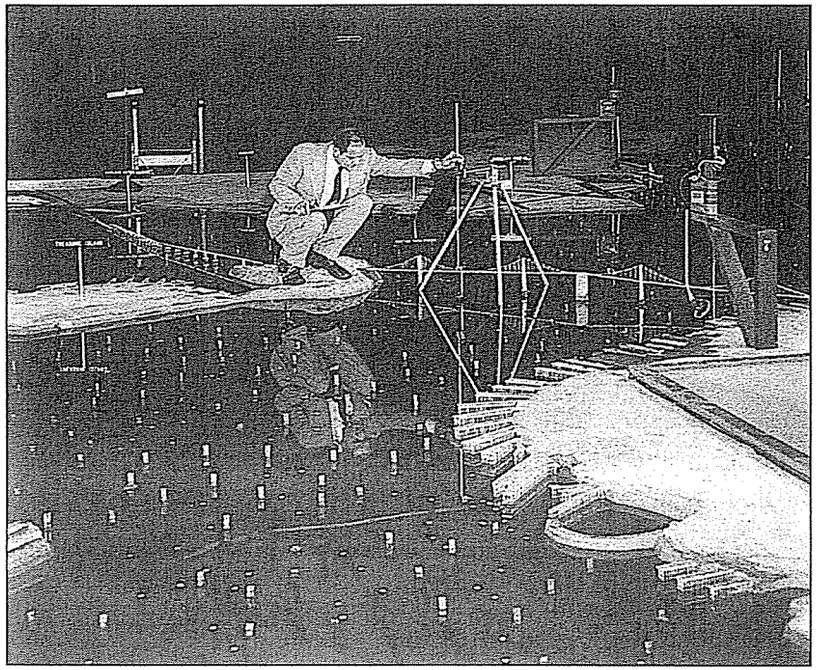


Figure 7.2
View of the Corps of Engineers Model of San Francisco Bay ($L_r = 1000$, $L_{r'} = 100$), showing the vertical metal tabs that were installed to provide proper frictional flow resistance. (U.S. Army Corps of Engineers)

operating at normal speed, this would make the model Mach number much greater than one, and compressibility effects would invalidate the behavior of the model. If, however, we conducted the test under a pressure of 20 atm with identical model and prototype temperatures, $\rho_m = 20 \times \rho_p$ and $\mu_m \approx \mu_p$, since the absolute viscosity of air changes very little with pressure (or density). In this case we should operate the model at a velocity equal to that of the prototype in order to have the same Reynolds numbers.

SAMPLE PROBLEM 7.3 A 1:50 model of a boat has a wave resistance of 0.02 N when operating in water at 1.0 m/s. Find the corresponding prototype wave resistance. Find also the horsepower requirement for the prototype. What velocity does this test represent in the prototype?

Solution
Gravity and inertia forces predominate; hence the Froude criterion is applicable.

$$\mathbf{F}_p = \mathbf{F}_m = \left(\frac{V}{\sqrt{gL}}\right)_p = \left(\frac{V}{\sqrt{gL}}\right)_m$$

Because the earth's gravitational field acts on both the model and the prototype, we can cancel out the gs. Thus

$$\frac{V_p^2}{L_p} = \frac{V_m^2}{L_m}$$

and

$$\frac{V_p^2}{V_m^2} = \frac{L_p}{L_m} = L_r = 50$$

Since

$$\frac{F_p}{F_m} = \frac{\rho L_p^2 V_p^2}{\rho L_m^2 V_m^2} = L_r^2 L_r = L_r^3$$

$$F_p = L_r^3 F_m = (50)^3(0.02) = 2500\ \text{N} = 562\ \text{lb} \qquad \textit{ANS}$$

$$V_p = \sqrt{L_r} \times V_m = \sqrt{50} \times 1 = 7.07\ \text{m/s} = 23.2\ \text{fps} \qquad \textit{ANS}$$

$$P_p = \frac{F_p V_p}{550} = \frac{562 \times 23.2}{550} = 23.7\ \text{hp} \qquad \textit{ANS}$$

7.7 DIMENSIONAL ANALYSIS

One approach we may take to fluid-mechanics problems is by using *dimensional analysis,* a mathematical technique that makes use of the study of dimensions. Dimensional analysis is related to similitude; however, the approach is different. In dimensional analysis, from a general understanding of fluid phenomena, we first predict the physical parameters that will influence the flow, and then we group these parameters into dimensionless combinations which enable a better understanding of the flow phenomena. Dimensional analysis is particularly helpful in experimental work because it provides a guide to those things that significantly influence the phenomena; thus it indicates the direction in which experimental work should go.

We may express physical quantities in either the force-length-time (FLT) system or the mass-length-time (MLT) system. This is because these two systems are interrelated through Newton's second law, which states that force equals mass times acceleration, $F = ma$, or

$$F = M\frac{L}{T^2}$$

Through this relation, we can convert from one system to the other. Other than convenience, it makes no difference which system we use, since the results are the same. The dimensions we use in either system may be in BG units or SI units. We give details on these systems of units and on conversion factors in Sec. 1.5 and on the inside covers of the book.

Basic Concepts

All theoretical equations that relate physical quantities must be dimensionally homogeneous. That is, all the terms in an equation must have the same dimensions. For example, as noted in Sec. 5.8, each of the terms in Bernoulli's equation (5.29) has the dimension of length. This principle is known as the ***principle of dimensional homogeneity*** (PDH), which was first formalized in 1822 by Baron Joseph Fourier (1768–1830), a French mathematician and physicist now best known for his Fourier series. The PDH is a valuable tool for checking equation derivations and engineering calculations, and, as we shall see shortly, it can be of great help in deriving the forms of physical equations.

We do, however, sometimes use nonhomogeneous equations, the best-known example in fluid mechanics being the Manning equation (8.64). Often such equations are the result of fitting equations to observed data. Generally their use is limited to specialized areas.

Because all the terms in a theoretical (dimensionally homogeneous) equation have the same dimensions, if we divide them all by a quantity that has the same dimensions then all the terms will become dimensionless. When we do this using various quantities with the proper dimensions, we often find certain dimensionless groups that recur frequently, such as the dimensionless numbers described in Sec. 7.4. As a result, we can express many equations more simply as relationships between dimensionless groups or numbers.

Dimensional analysis is a powerful scientific procedure that formalizes this process. It readily arranges all the variables involved into an equation containing independent dimensionless groups, so avoiding the experimentation. Another benefit it yields is that the number of independent groups so obtained is less than the number of variables.

To illustrate the basic principles of dimensional analysis, let us explore the equation for the speed V with which a pressure wave travels through a fluid. We must visualize the physical problem to consider what physical factors probably influence the speed. Certainly the compressibility E_v (Secs. 2.5 and 2.8) must be a factor; also the density ρ and the kinematic viscosity ν of the fluid (Secs. 2.3 and 2.11) might be factors. The dimensions of these quantities, written in square brackets, are

$$V = \left[\frac{L}{T}\right], \qquad E_v = \left[\frac{F}{L^2}\right] = \left[\frac{M}{LT^2}\right], \qquad \rho = \left[\frac{M}{L^3}\right], \qquad \nu = \left[\frac{L^2}{T}\right]$$

Here we converted the dimensions of E_v into the MLT system using $F = ML/T^2$ as noted earlier. Clearly, adding or subtracting such quantities will not produce dimensionally homogeneous equations. We must therefore multiply them in such a way that their dimensions balance. So let us write

$$V = CE_v^a \rho^b \nu^d$$

where C is a dimensionless constant, and let us solve for the exponents a, b, and d. Substituting the dimensions, we get

$$\frac{L}{T} = \left(\frac{M}{LT^2}\right)^a \left(\frac{M}{L^3}\right)^b \left(\frac{L^2}{T}\right)^d$$

To satisfy dimensional homogeneity, the exponents of each dimension must be identical on both sides of this equation. Thus

For M: $\quad 0 = a + b$
For L: $\quad 1 = -a - 3b + 2d$
For T: $\quad -1 = -2a - d$

Solving these three equations, we get

$$a = \tfrac{1}{2}, \qquad b = -\tfrac{1}{2}, \qquad d = 0$$

so that

$$V = C\sqrt{\frac{E_v}{\rho}}$$

This identifies the basic form of the relationship, which we will treat later as Eqs. (12.9) and (13.17), and it also determines that the wave speed is not affected by the fluid's kinematic viscosity, ν.

Dimensional analysis along such lines was developed by Lord Rayleigh, who we mentioned in Sec. 7.4. Although it is a very serviceable method, it has been superseded.

The Pi Theorem

A more generalized method of dimensional analysis developed by E. Buckingham[7] and others is most popular now. This arranges the variables into a lesser number of dimensionless groups of variables. Because Buckingham used an upper case Π (pi) to represent the product of variables in each group, we now call this method the ***pi theorem*** or the Buckingham pi theorem.

Let $X_1, X_2, X_3, \ldots, X_n$ represent *n dimensional* variables, such as velocity, density, and viscosity, which are involved in some physical phenomenon. We can write the dimensionally homogeneous equation relating these variables as

$$f(X_1, X_2, X_3, \ldots, X_n) = 0$$

in which the dimensions of each *term* are the same. It follows that we can rearrange this equation into

$$\phi(\Pi_1, \Pi_2, \ldots, \Pi_{n-k}) = 0$$

where ϕ (phi) is another function and each Π is an independent *dimensionless* product of some of the Xs. The reduction k in the number of terms (from n to $n - k$), is usually equal to and sometimes less than the number of fundamental dimensions m involved in all the variables.

We need to follow a series of seven steps when applying the pi theorem. As we review these, let us relate them to an example problem, that of the drag force F_D exerted on a submerged sphere as it moves through a viscous fluid.

Step 1: Visualize the physical problem, consider the factors that are of influence, and list and count the n variables.

[7] E. Buckingham, Model Experiments and the Form of Empirical Equations, *Trans. ASME,* Vol. 37, pp. 263–296, 1915.

In our example we must first consider which physical factors influence the drag force. Certainly, the size of the sphere must enter the problem; also, the velocity of the sphere must be important.The fluid properties involved are the density ρ and the viscosity μ. Thus we can write

$$f(F_D, D, V, \rho, \mu) = 0$$

Here we use D, the sphere diameter, to represent sphere size, and f stands for "some function." We see that $n = 5$. Note that the procedure cannot work if any relevant variables are omitted. Experimentation with the procedure and experience will help determine which variables are relevant.

Step 2: Choose a dimensional system (MLT or FLT) and list the dimensions of each variable. Find m, the number of fundamental dimensions involved in all the variables.

In our example, choosing the MLT system, the dimensions are, respectively

$$\frac{ML}{T^2}, \quad L, \quad \frac{L}{T}, \quad \frac{M}{L^3}, \quad \frac{M}{LT}$$

We can get help in identifying these dimensions from Table 7.1 and from the units of quantities on the inside covers and in the appendices, etc. We see that M, L, and T are involved in our example, so $m = 3$.

Step 3: Find the reduction number k. Usually this equals m, which it cannot exceed, but rarely it is less than m. To check this, try to find m dimensional variables that *cannot* be formed into a dimensionless group. If m are found, $k = m$; if not, reduce k by one and retry.

In our example we find that three of the dimensional variables, namely, ρ, D, and V, with dimensions M/L^3, L, and L/T, will *not* form into a dimensionless Π group because M and L cannot cancel among them, so $k = 3$. The rare circumstances that reduce k arise when some of the dimensions (usually M and T) occur in the parameters only in fixed combinations.

Step 4: Determine $n - k$, the number of dimensionless Π groups needed. In our example this is $5 - 3 = 2$. So we can write $\phi(\Pi_1, \Pi_2) = 0$.

Step 5: From the list of dimensional variables, select k of them to be so-called primary (repeating) variables. These must contain all of the m fundamental dimensions, and must not form a Π among themselves (see step 3). Generally it helps to choose primary variables that relate to mass, geometry, and kinematics (flow without forces or energy). Form the Π groups by multiplying the product of the primary variables, with unknown exponents, by each of the remaining variables, one at a time.

We choose ρ, D, and V (as in step 3) as the primary variables for our example. Then the Π terms are

$$\Pi_1 = \rho^{a_1} D^{b_1} V^{c_1} \mu$$
$$\Pi_2 = \rho^{a_2} D^{b_2} V^{c_2} F_D$$

Step 6: To satisfy dimensional homogeneity, equate the exponents of each dimension on both sides of each pi equation, and so solve for the exponents and the forms of the dimensionless groups. Since the Πs are dimensionless, we can replace them with $M^0L^0T^0$. Experience in fluid mechanics tells us that these dimensionless groups commonly take the form of a Reynolds number, Froude number, or Mach number. So we should always watch out for them when using dimensional analysis.

For our example, working with Π_1,

$$M^0L^0T^0 = \left(\frac{M}{L^3}\right)^{a_1} L^{b_1} \left(\frac{L}{T}\right)^{c_1} \left(\frac{M}{LT}\right)$$

M: $\quad 0 = a_1 + 1$
L: $\quad 0 = -3a_1 + b_1 + c_1 - 1$
T: $\quad 0 = -c_1 - 1$

Solving for a_1, b_1, and c_1,

$$a_1 = -1, \quad b_1 = -1, \quad c_1 = -1$$

Thus
$$\Pi_1 = \rho^{-1} D^{-1} V^{-1} \mu = \frac{\mu}{\rho DV} = \left(\frac{\rho DV}{\mu}\right)^{-1}$$

Noting that $(\rho DV/\mu)$ is a Reynolds number, therefore

$$\Pi_1 = \mathbf{R}^{-1}$$

Working in a similar fashion with Π_2, we get

$$\Pi_2 = \frac{F_D}{\rho D^2 V^2}$$

Check that all Πs are in fact dimensionless.

Step 7: Rearrange the pi groups as desired. The pi theorem states that the Πs are related, and may be expressed as $f_1(\Pi_1, \Pi_2, \ldots, \Pi_{n-k}) = 0$ or as $\Pi_1 = f_2(\Pi_2, \Pi_3, \ldots, \Pi_{n-k})$, etc. It does not predict the functional form of f_1 or f_2; we must determine these relations experimentally. Further, we can raise each Π parameter to any power, as this will not affect their dimensionless status. Note that if there is only one dimensionless group, $f(\Pi) = 0$ and so we must have $\Pi =$ constant.

In our example, since $f_1\left(\frac{F_D}{\rho D^2 V^2}, \mathbf{R}\right) = 0$ this indicates that $\frac{F_D}{\rho D^2 V^2}$ depends only on $\mathbf{R}$. Alternatively, since we are interested in F_D, if we write $\Pi_2 = \phi(\Pi_1^{-1})$ we get

$$\frac{F_D}{\rho D^2 V^2} = \phi(\mathbf{R})$$

so that
$$F_D = \rho D^2 V^2 \phi(\mathbf{R})$$

We must emphasize that dimensional analysis does not provide a complete solution to fluid problems. It provides a partial solution only. The success of

dimensional analysis depends entirely on the ability of the individual using it to define the parameters that are applicable. If we omit an important variable, the results are incomplete, and this may lead to incorrect conclusions. For example, with a compressible fluid at high velocities, compressibility effects may be significant, in which case we must consider the volume modulus E_v of the fluid an important physical property. Introducing E_v into the previous example of dimensional analysis of the drag on a sphere will show that for the more general case the drag may depend on the Mach number as well as the Reynolds number. If we include a variable that is totally unrelated to the problem, an additional insignificant dimensionless group will result. Thus, to use dimensional analysis successfully, we must be familiar with the fluid phenomena involved.

SAMPLE PROBLEM 7.4 Derive an expression for the flow rate q over the spillway shown in Fig. S7.4 per foot of spillway perpendicular to the sketch. Assume that the sheet of water is relatively thick, so that surface-tension effects are negligible. Assume also that gravity effects predominate so strongly over viscosity that we can neglect viscosity.

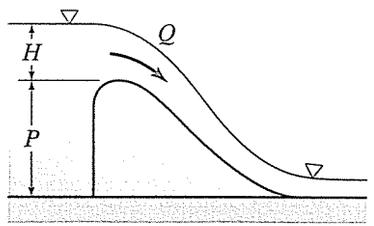


Figure S7.4

Solution

Under the assumed conditions the variables that affect q would be the head H, the acceleration of gravity g, and possibly the spillway height P. Thus

$$q = f(H, g, P)$$

or

$$f_1(q, H, g, P) = 0$$

In this case there are $n = 4$ variables and $m = 2$ dimensions. We can easily find two variables that we cannot form into a dimensionless group; therefore $k = m = 2$, and so there are $n - k = 2$ pi groups and

$$\phi(\Pi_1, \Pi_2) = 0$$

Using q and H as the primary variables,

$$\Pi_1 = q^{a_1} H^{b_1} g$$

$$\Pi_2 = q^{a_2} H^{b_2} P$$

Working with Π_1,

$$L^0T^0 = \left(\frac{L^3}{TL}\right)^{a_1} L^{b_1} \left(\frac{L}{T^2}\right)$$

L: $\quad 0 = 2a_1 + b_1 + 1$

T: $\quad 0 = -a_1 - 2$

So $\quad a_1 = -2, \quad b_1 = 3$

and

$$\Pi_1 = q^{-2} H^3 g = \frac{gH^3}{q^2}$$

Working with Π_2,

$$L^0T^0 = \left(\frac{L^3}{TL}\right)^{a_2} L^{b_2} L$$

L: $\quad 0 = 2a_2 + b_2 + 1$

T: $\quad 0 = -a_2$

So $\quad a_2 = 0, \quad b_2 = -1$

and

$$\Pi_2 = q^0 H^{-1} P = \frac{P}{H}$$

Finally, we can write $\phi(\Pi_1, \Pi_2) = 0$ as

$$\Pi_1^{-1/2} = \phi_1\left(\Pi_2^{-1}\right)$$

i.e.,

$$\frac{q}{\sqrt{g} H^{3/2}} = \phi_1\left(\frac{H}{P}\right)$$

or

$$q = \sqrt{g} H^{3/2} \phi_1\left(\frac{H}{P}\right)$$

Thus dimensional analysis indicates that the flow rate per unit length of spillway is proportional to $\sqrt{g}$ and to $H^{3/2}$. The flow rate also depends on the H/P ratio. We discuss this relationship in Secs. 11.11–11.13.

If we included viscosity as one of the variables, we would have obtained another dimensionless group, which would have had the form of a Reynolds number. If we included surface tension, the resulting dimensionless group would have been a Weber number.

EXERCISES

7.7.1 Use dimensional analysis to arrange the following groups into dimensionless parameters: (*a*) τ, V, ρ; (*b*) V, L, ρ, σ. Use the MLT system.

7.7.2 Use dimensional analysis to arrange the following groups into dimensionless parameters: (*a*) $\Delta p, V, \gamma, g$; (*b*) F, ρ, L, V. Use the MLT system.

7.7.3 Use dimensional analysis to derive an expression for the power developed by an engine in terms of the torque T and rotative speed ω.

7.7.4 Derive an expression for the shear stress at the pipe wall when an incompressible fluid flows through a pipe under pressure. Use dimensional analysis with the following significant parameters: pipe diameter D, flow velocity V, and viscosity μ and density ρ of the fluid.

7.7.5 Refer to the example of the drag on a submerged sphere, which illustrates the seven-step procedure of Sec. 7.7. Using the same parameters, but also including the acceleration due to gravity g to account for the effect of wave action, derive an expression for the drag on a surface vessel.

PROBLEMS

7.1 What is the Reynolds number for 110°F air at a pressure of 90 psia when it flows at a velocity of 120 fps through a 4-in-diameter pipe? Note that Appendix A gives physical properties of air.

7.2 What is the Reynolds number for 50°C air at a pressure of 650 kPa abs when it flows at a velocity of 35 m/s through a 100-mm-diameter pipe? Note that Appendix A gives physical properties of air.

7.3 The linear dimensions of a model airplane are $\frac{1}{15}$ those of its prototype. If the prototype is to fly at 450 mph, what must be the air velocity in a wind tunnel for the same Reynolds number if the air temperature and pressure are the same?

7.4 Air at 150°C and a pressure of 240 kPa abs flows at a velocity of 16 m/s through an 180-mm-diameter pipe. What is the Reynolds number?

7.5 Models are to be built of the following prototypes: (*a*) tides; (*b*) oil flowing through a full pipeline; (*c*) a water jet; (*d*) flow over the spillway of a dam; (*e*) a deep submersible vehicle; (*f*) an airplane flying at low speed; (*g*) a supersonic aircraft; (*h*) a supersonic missile. For dynamic similarity, indicate which single dimensionless ratio will govern, and give reasons why.

7.6 Air at 68°F and 60 psia flows in a 1.5-in-diameter pipe. What weight flow rate of this air will give dynamic similarity to 60°F water flowing at 200 gpm through a 3-in-diameter pipe?

7.7 When a submerged sphere moves through 15°C water at 2.0 m/s (Fig. P7.7), the drag force exerted on it is 16 N. In wind tunnel tests with another sphere of three times the diameter the air pressure and temperature are 1.8 MPa abs and 290 K, respectively. What must the air velocity be for dynamic similarity, and what will the drag force then be on the larger sphere?

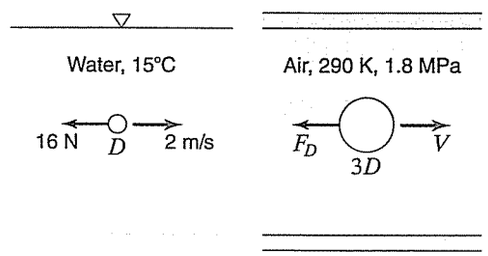


Figure P7.7

7.8 Air at 80°C and 475 kPa abs pressure flows in a 50-mm-diameter pipe. What air flow rate (kg/s) will give dynamic similarity to 70 L/s of 60°C water flowing in a 540-mm-diameter pipe?

7.9 A 500-ft-long ship will operate at a speed of 20 mph in ocean water whose viscosity is 1.2 cP and specific weight is 64 lb/ft^3. What should be the kinematic viscosity of the liquid used with a 10-ft-long model of this ship so that both the Reynolds number and the Froude number would be the same? Does such a liquid exist?

7.10 Water flows over a spillway at 5000 cfs. For dynamic similarity, what should the model scale be if the flow rate over the model is to be 45 cfs? The force exerted on a certain area of the model is 1.0 lb. What would the force be on the corresponding area of the prototype?

7.11 A 1:600 scale model is built to study tides. (*a*) What length of time in the model corresponds to one day in the prototype? (*b*) Suppose this model could be tested on the moon where g is one-sixth of that on earth. What then would be the time relationship between the model and prototype?

7.12 A vertical jet of water issuing upward from a nozzle at a velocity of 44 fps will rise to a height of approximately 30 ft on the earth. To get a water jet to rise to a height of 120 ft on the moon, where the gravity is one-sixth of that on earth, what must the jet velocity be? Neglect atmospheric resistance.

7.13 A 3-ft-high sectional model of a spillway is built in a 1-ft-wide laboratory flume. The flow is 0.86 cfs under a head of 0.380 ft. If the model scale is 1:20 and the prototype spillway is 600 ft long, what flow does this represent in the prototype?

7.14 A model spillway has a flow of 100 L/s per m of width. What is the actual flow for the prototype spillway if the model scale is 1:20?

7.15 When traveling at a velocity of 3 fps a 1:50 scale model of a ship has a wave resistance of 0.07 lb. This is kinematically similar to the design velocity of the prototype ship. (*a*) What is the design velocity of the prototype in the same water? and (*b*) what is its wave resistance at that velocity?

7.16 An aircraft company is investigating the flow about a model of a supersonic plane in a variable-density wind tunnel at 1500 fps; the air, at 80°F, has a pressure of 20 psia. What should the velocity be to maintain dynamic similarity if the air temperature is raised to 90°F and the pressure is increased to 30 psia? Solve this two ways: (*a*) using specific weights and densities, and (*b*) not using them.

7.17 An aircraft company is investigating the flow about a model of a supersonic plane in a variable density wind tunnel at 400 m/s; the air, at 40°C, has a pressure of 150 kPa abs. What should the velocity be to maintain dynamic similarity if the air temperature is raised to 75°C and the pressure is increased to 200 kPa abs?

7.18 Researchers plan to test a 1:13 model of a ballistic missile in a high-speed wind tunnel. The prototype missile will travel at 1250 fps through air at 68°F and 14.5 psia. (*a*) If the air in the wind tunnel test section has a temperature of 32°F and a pressure of 12.1 psia, what must its velocity be? (*b*) Estimate the drag force on the prototype if the drag force on the model is 95 lb.

7.19 Researchers plan to test a 1:13 model of a ballistic missile in a high-speed wind tunnel. The prototype missile will travel at 380 m/s through air at 23°C and 95.0 kPa abs. (*a*) If the air in the wind tunnel test section has a temperature of −20°C at a pressure of 89 kPa abs, what must its velocity be? (*b*) Estimate the drag force on the prototype if the drag force on the model is 400 N.

7.20 Find the dimensions of pressure, energy, and momentum in the FLT system. Repeat for the MLT system.

7.21 Find the dimensions of force, torque, and power in the FLT system. Repeat for the MLT system.

7.22 Develop the scale ratios given in Table 7.1 for the case where prototype and model Reynolds numbers are the same.

7.23 Develop the scale ratios given in Table 7.1 for the case where prototype and model Froude numbers are identical.

7.24 Develop the scale ratios given in Table 7.1 for the case where prototype and model Mach numbers are the same.

7.25 Gas ($\gamma = 0.32$ lb/ft^3, $\mu = 2.0 \times 10^{-6}$ lb·sec/ft^2) is flowing in an 0.75-in-diameter pipe. When a gas flowmeter measures the flow as being 0.13 lb/sec, it registers a pressure drop of 1.17 psi. Investigators plan to test an enlarged model that is geometrically similar in a 6-in-diameter pipe. (*a*) What flow rate of 80°F water will achieve dynamic similarity? (*b*) What would the pressure drop across the water meter be?

7.26 Gas ($\rho = 5.25$ kg/m^3, $\nu = 2.0 \times 10^{-5}$ m^2/s) is flowing in a 20-mm-diameter pipe. When a gas flowmeter measures the flow as being 0.064 kg/s, it registers a pressure drop of 8.5 kPa. Investigators plan to test an enlarged model that is geometrically similar in a 180-mm-diameter pipe. (*a*) What flow rate of 25°C water will achieve dynamic similarity? (*b*) What would the pressure drop across the water meter be?

7.27 (*a*) Use dimensional analysis and the MLT system to arrange the following into a dimensionless number: g, L, ρ, and μ. (*b*) Name the dimensionless number.

7.28 (*a*) Use dimensional analysis and the MLT system to arrange the following into a dimensionless number: g, ρ, μ, and σ. (*b*) Name the dimensionless number.

7.29 (*a*) Use dimensional analysis and the MLT system to arrange the following into a dimensionless number: L, ρ, μ, and σ. (*b*) Name the dimensionless number.

7.30 Use dimensional analysis to derive an expression for the height of capillary rise in a glass tube.

7.31 Derive an expression for the drag on an aircraft flying at supersonic speed.

7.32 Derive an expression for small flow rates over a spillway, in the form of a function including dimensionless quantities. Use dimensional analysis with the following parameters: height of spillway P, head on the spillway H, viscosity of liquid μ, density of liquid ρ, surface tension σ, and acceleration due to gravity g.

7.33 Derive an expression for the velocity of rise of an air bubble in a stationary liquid. Consider the effect of surface tension as well as other variables.

CHAPTER 8
Steady Incompressible Flow in Pressure Conduits

In this chapter we will discuss some aspects of steady flow in pressure conduits. We shall limit our discussion to ***incompressible fluids,*** that is, to those that have $\rho \approx$ constant. This includes all liquids. When gases flow with very small pressure changes we can consider them incompressible, for then $\rho \approx$ constant. In this chapter we will assume isothermal conditions so as to eliminate thermodynamic effects, some of which we discuss in Chap. 13.

8.1 LAMINAR AND TURBULENT FLOW

If we measure the head loss in a given length of uniform pipe at different velocities, we will find that, as long as the velocity is low enough to secure laminar flow (Sec. 4.2), the head loss, due to friction, is directly proportional to the velocity, as shown in Fig. 8.1. But with increasing velocity, at some point B, where visual observation of dye injected in a transparent tube would show that the flow changes from laminar to turbulent (Sec. 4.2), there will be an abrupt increase in the rate at which the head loss varies. If we plot the logarithms of these two variables on linear scales or, in other words, if we plot the values directly on log–log paper, we will find that, after passing a certain transition region (BCA in Fig. 8.1), the lines will have slopes ranging from about 1.75 to 2.00.

Thus we see that for laminar flow the drop in energy due to friction varies as V, while for turbulent flow the friction varies as V^n, where n ranges from about 1.75 to 2. The lower value of 1.75 for turbulent flow occurs for pipes with very smooth walls; as the wall roughness increases, the value of n increases up to its maximum value of 2.

The points in Fig. 8.1 are plotted directly from measurements made by Osborne Reynolds (Sec. 7.4), and show decided curves in the transition zone where values of n are even greater than 2. If we gradually reduce the velocity from a high value, the points will not return along line BC. Instead, the points will lie along curve CA. We call point B the ***higher critical point,*** and A the ***lower critical point.***

However, velocity is not the only factor that determines whether the flow is laminar or turbulent. The criterion is Reynolds number, which we discussed in Sec. 7.4. For a circular pipe we usually take the significant linear dimension L as

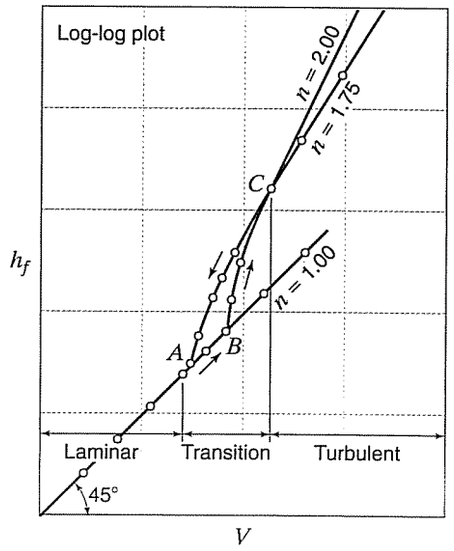


Figure 8.1
Log–log plot for flow in a uniform pipe ($n = 2.00$, rough-wall pipe; $n = 1.75$, smooth-wall pipe).

the diameter D, and thus from Eq. (7.6)

Pipe flow:
$$\mathbf{R} = \frac{DV\rho}{\mu} = \frac{DV}{\nu} \tag{8.1}$$

where we can use any consistent system of units, because **R** is a dimensionless number.[1]

8.2 CRITICAL REYNOLDS NUMBER

The upper critical Reynolds number, corresponding to point B of Fig. 8.1, is really indeterminate and depends on the care taken to prevent any initial disturbance from affecting the flow. Its value is normally about 4000, but experimenters have maintained laminar flow in circular pipes up to values of **R** as high as 50,000. However, in such cases this type of flow is inherently unstable, and

[1] Sometimes we find it convenient to use a "hybrid" set of units and compensate with a correction factor. Thus, by substituting $V = Q/A$ and $V = \dot{m}/\rho A$ from Eq. (4.6) into Eq. (8.1), we get

$$\mathbf{R} = 1.273Q/\nu D = 1.273\dot{m}/\mu D,$$

where Q and $\dot{m}$ are the volume flow rate and the mass flow rate, respectively. The last form is especially convenient in the case of gases; it shows that in a pipe of uniform diameter the Reynolds number is constant along the pipe, even for a compressible fluid, where the density and velocity vary, if there is no appreciable variation in temperature to alter the viscosity of the gas.

the least disturbance will transform it instantly into turbulent flow. On the other hand, it is practically impossible for turbulent flow in a straight pipe to persist at values of **R** much below 2000, because any turbulence that occurs is damped out by viscous friction. This lower value is thus much more definite than the higher one, and is the real dividing point between the two types of flow. So we define this lower value as the ***true critical Reynolds number.*** However, it may vary slightly. It will be higher in a converging pipe and lower in a diverging pipe than in a straight pipe. Also, it will be less for flow in a curved pipe than in a straight one, and even for a straight uniform pipe it may be as low as 1000, where there is excessive roughness. However, for normal cases of flow in straight pipes of uniform diameter and usual roughness, we can take the critical value as

$$\mathbf{R}_{\text{crit}} = 2000 \tag{8.2}$$

For water at 59°F (15°C), using Eqs. (8.1) and (8.2) we find:

when $D = 1$ in (25 mm), $V_{\text{crit}} = 0.30$ fps (0.091 m/s)

and when $V = 3$ fps (0.91 m/s), $D_{\text{crit}} = 0.10$ in (2.5 mm)

We seldom encounter velocities or pipe diameters as small as these with water flowing in practical engineering, though they may occur in certain laboratory instruments. Therefore, for such fluids as water and air, practically all cases of engineering importance are in the turbulent-flow region. But if the fluid is a viscous oil, laminar flow often occurs.

SAMPLE PROBLEM 8.1 In a refinery oil ($s = 0.85$, $\nu = 1.8 \times 10^{-5}$ m^2/s) flows through a 100-mm-diameter pipe at 0.50 L/s. Is the flow laminar or turbulent?

Solution

Eq. (4.7):
$$V = \frac{4Q}{\pi D^2} = \frac{4(0.0005 \text{ m}^3/\text{s})}{\pi(0.1 \text{ m})^2} = 0.0637 \text{ m/s}$$

Eq. (8.1):
$$\mathbf{R} = \frac{DV}{\nu} = \frac{0.10 \text{ m}(0.0637 \text{ m/s})}{1.8 \times 10^{-5} \text{ m}^2/\text{s}} = 354$$

Using Eq. (8.2), since $\mathbf{R} < \mathbf{R}_{\text{crit}} = 2000$, the flow is laminar. ***ANS***

EXERCISES

8.2.1 Oil with a kinematic viscosity of 0.00015 ft^2/sec is flowing through a 3-in-diameter pipe. Below what velocity will the flow be laminar?

8.2.2 Oil with a kinematic viscosity of 0.185 St is flowing through a 150-mm-diameter pipe. Below what velocity will the flow be laminar?

8.2.3 Oil with a kinematic viscosity of 0.0035 ft^2/sec flows through a 4-in-diameter pipe with a velocity of 15 fps. Is the flow laminar or turbulent?

8.3 HYDRAULIC RADIUS, HYDRAULIC DIAMETER

For conduits having noncircular cross sections, we need to use some value other than the diameter for the linear dimension in the Reynolds number. The characteristic dimension we use is the ***hydraulic radius,*** defined as

$$R_h = \frac{A}{P} \tag{8.3}$$

where A is the cross-sectional area of the flowing fluid, and P is the ***wetted perimeter,*** that portion of the perimeter of the cross section where the fluid contacts the solid boundary, and therefore where friction resistance is exerted on the flowing fluid. For a circular pipe flowing full,

Full-pipe flow:
$$R_h = \frac{\pi r^2}{2\pi r} = \frac{r}{2} = \frac{D}{4} \tag{8.4}$$

Thus R_h is not the radius of the pipe, and so the term "radius" is misleading. If a circular pipe is exactly half full, both the area and the wetted perimeter are half the preceding values; so R_h is $r/2$, the same as if it were full. But if the depth of flow in a circular pipe is 0.8 times the diameter, for example, $A = 0.674D^2$ and $P = 2.21D$, then $R_h = 0.304D$, or $0.608r$. We discuss part-full pipe flow further in Sec. 10.7.

The hydraulic radius is a convenient means for expressing the shape as well as the size of a conduit, since, for the same cross-sectional area, the value of R_h will vary with the shape.

In evaluating the Reynolds number for flow in a noncircular conduit (Sec. 8.6), customarily we substitute $4R_h$ for D in Eq. (8.1).

Workers in some engineering fields define $4R_h$ to be the ***hydraulic diameter*** D_h. Then, for a pipe, $D_h = D$, which is fine, but $D_h = 4R_h$, which seems strange.

EXERCISES

8.3.1 What is the hydraulic radius of a 12 in by 16 in rectangular air duct?

8.3.2 What is the percentage difference between the hydraulic radii of a 300-mm-diameter duct and a 300-mm-square duct?

8.4 FRICTION HEAD LOSS IN CONDUITS OF CONSTANT CROSS SECTION

This discussion applies to either laminar or turbulent flow and to any shape of cross section.

Consider steady flow in a conduit of uniform cross section A, not necessarily circular (Fig. 8.2). The pressures at sections 1 and 2 are p_1 and p_2, respectively. The distance between the sections is L. For equilibrium in steady flow, the

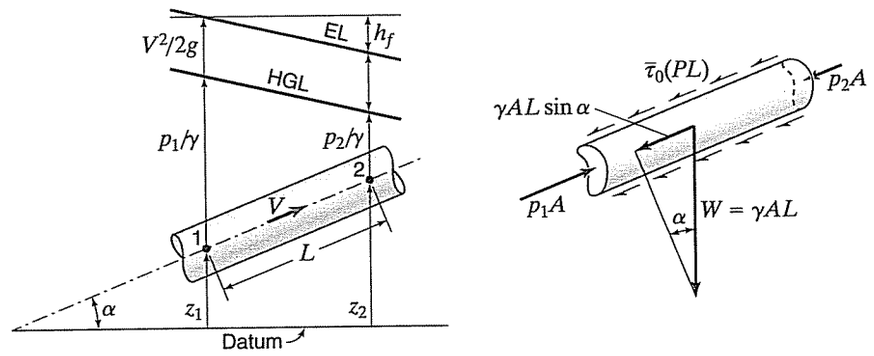


Figure 8.2

summation of forces acting on any fluid element must be equal to zero (i.e., $\sum F = ma = 0$). Thus, in the direction of flow,

$$p_1A - p_2A - \gamma LA\sin\alpha - \bar{\tau}_0(PL) = 0 \tag{8.5}$$

where we define $\bar{\tau}_0$, the ***average shear stress*** (average shear force per unit area) at the conduit wall, by

$$\bar{\tau}_0 = \frac{\int_0^P \tau_0 dP}{P} \tag{8.6}$$

in which τ_0 is the local shear stress[2] acting over a small incremental portion dP of the wetted perimeter.

Noting that $\sin\alpha = (z_2 - z_1)/L$ and dividing each term in Eq. (8.5) by γA gives

$$\frac{p_1}{\gamma} - \frac{p_2}{\gamma} - z_2 + z_1 = \bar{\tau}_0\frac{PL}{\gamma A} \tag{8.7}$$

From the left-hand sketch of Fig. 8.2, we can see that the head loss due to friction at the wetted perimeter is

$$h_f = \left(z_1 + \frac{p_1}{\gamma}\right) - \left(z_2 + \frac{p_2}{\gamma}\right)$$

This equation indicates that h_f depends only on the values of z and p on the centerline, and so it is the same regardless of the size of the cross-sectional area A. Substituting h_f for the right-hand side of this expression and replacing A/P in

[2] The local shear stress varies from point to point around the perimeter of all conduits (regardless of whether the wall is smooth or rough), except for the case of a circular pipe flowing full where the shear stress at the wall is the same at all points on the perimeter.

Eq. (8.7) by R_h from Eq. (8.3), we get

$$h_f = \bar{\tau}_0 \frac{L}{R_h \gamma} \tag{8.8}$$

This equation is applicable to any shape of uniform cross section, regardless of whether the flow is laminar or turbulent. Its derivation is very similar to that in Sec. 5.3, in which the conduit cross section was not constant, however.

For a smooth-walled conduit, where we can neglect wall roughness (discussed in Sec. 8.10), we might assume that the average fluid shear stress $\bar{\tau}_0$ at the wall is some function of ρ, μ, and V and some characteristic linear dimension, which we will here take as the hydraulic radius R_h. Thus

$$\bar{\tau}_0 = f(\rho, \mu, V, R_h) \tag{8.9}$$

Using the pi theorem of dimensional analysis (Sec. 7.7) to better determine the form of this relationship, we choose ρ, R_h, and V as primary variables, so that

$$\Pi_1 = \mu \rho^{a_1} R_h^{b_1} V^{c_1}$$

$$\Pi_2 = \bar{\tau}_0 \rho^{a_2} R_h^{b_2} V^{c_2}$$

With the dimensions of the variables being $ML^{-1}T^{-1}$ for μ, $ML^{-1}T^{-2}$ for $\bar{\tau}_0$, ML^{-3} for ρ, L for R_h, and LT^{-1} for V, the dimensions for Π_1 are

For M: $0 = 1 + a_1$

For L: $0 = -1 - 3a_1 + b_1 + c_1$

For T: $0 = -1 - c_1$

The solution of these three simultaneous equations is $a_1 = b_1 = c_1 = -1$, from which

$$\Pi_1 = \frac{\mu}{\rho R_h V} = \mathbf{R}^{-1}$$

where $R_h V \rho / \mu$ is a Reynolds number with R_h as the characteristic length. In a similar manner, we obtain

$$\Pi_2 = \frac{\bar{\tau}_0}{\rho V^2}$$

According to Sec. 7.7, step 7, we can write $\Pi_2 = \phi(\Pi_1^{-1})$, which results in $\bar{\tau}_0 = \rho V^2 \phi(\mathbf{R})$. Setting the dimensionless term $\phi(\mathbf{R}) = \frac{1}{2} C_f$, this yields

$$\bar{\tau}_0 = C_f \rho \frac{V^2}{2} \tag{8.10}$$

Inserting this value of $\bar{\tau}_0$ into Eq. (8.8), and noting from Eq. (2.1) that $\gamma = \rho g$,

$$h_f = C_f \frac{L}{R_h} \frac{V^2}{2g} \tag{8.11}$$

which we can apply to any shape of smooth-walled cross section. From this

equation, we may easily obtain an expression for the slope of the energy line,

$$S = \frac{h_f}{L} = \frac{C_f}{R_h} \frac{V^2}{2g} \tag{8.12}$$

which we also know as the ***energy gradient.***

Later, in Sec. 8.12, we shall see that Eqs. (8.11) and (8.12) also apply to rough-walled conduits.

8.5 FRICTION IN CIRCULAR CONDUITS

In Sec. 8.3 we saw that for a circular pipe flowing full $R_h = D/4$. Substituting this value into Eqs. (8.11) and (8.12), we obtain (for both smooth-walled and rough-walled conduits) the well-known equation for ***pipe-friction head loss,***

Circular pipe, flowing full (laminar or turbulent flow):
$$h_f = f \frac{L}{D} \frac{V^2}{2g} \tag{8.13}$$

and
$$\frac{h_f}{L} = S = \frac{f}{D} \frac{V^2}{2g} \tag{8.14}$$

where
$$f = 4C_f = 8\phi(\mathbf{R}) \tag{8.15}$$

Equation (8.13) is known as the ***pipe-friction equation,*** and as the ***Darcy-Weisbach equation.***[3] Like the coefficient C_f, the ***friction factor*** f (also sometimes called the ***Darcy friction factor***) is dimensionless and some function of Reynolds number. Much research has gone into determining how f varies with $\mathbf{R}$ and also with pipe roughness (see Sec. 8.12). The pipe-friction equation states that the head lost in friction in a given pipe is proportional to the velocity head. The equation is dimensionally homogeneous, and we may use it with any consistent system of units.

Dimensional analysis gives us the proper form for an equation, but does not yield a numerical result, since it does not deal with abstract numerical factors. It also shows that Eq. (8.13) is a rational expression for pipe friction. But we must determine the exact form of $\phi(\mathbf{R})$ and numerical values for C_f and f by experiment or other means.

For a circular pipe flowing full, by substituting $R_h = r_0/2$, where r_0 is the radius of the pipe, we can write Eq. (8.8) as

$$h_f = \bar{\tau}_0 \frac{L}{R_h \gamma} = \frac{2\tau_0 L}{r_0 \gamma} \tag{8.16}$$

where the local shear stress at the wall, τ_0, is equal to the average shear stress $\bar{\tau}_0$ because of symmetry.

[3] In a slightly different form where the hydraulic radius R_h replaces D, so that f changes, Eq. (8.13) is known as the ***Fanning equation,*** which chemical engineers use widely.

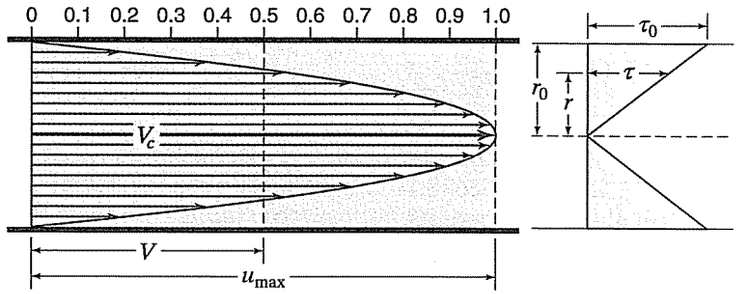


Figure 8.3
Velocity profile in laminar flow, and distribution of shear stress.

For a cylindrical fluid body of radius r concentric with the pipe, if we follow a development similar to that of Eqs. (8.5)–(8.8) and note that $A = \pi r^2$ and $P = 2\pi r$, we can show that

$$h_f = \frac{2\tau L}{r\gamma} \tag{8.17}$$

where τ is the shear stress in the fluid at radius r; this also follows from Eq. (5.12). Since h_f is the same for all r (Sec. 8.4), equating the right sides of Eqs. (8.16) and (8.17) yields

$$\tau = \tau_0 \frac{r}{r_0} \tag{8.18}$$

or the shear stress is zero at the center of the pipe and increases linearly with the radius to a maximum value τ_0 at the wall as in Fig. 8.3. This is true regardless of whether the flow is laminar or turbulent.

From Eqs. (8.8) and (8.13) and substituting $R_h = D/4$ for a circular pipe, we obtain

$$\tau_0 = \frac{f}{4}\rho\frac{V^2}{2} = \frac{f}{4}\gamma\frac{V^2}{2g} \tag{8.19}$$

With this equation, we can compute τ_0 for flow in a circular pipe for any experimentally determined value of f.

EXERCISES

8.5.1 Steam with a specific weight of 0.32 lb/ft^3 is flowing with a velocity of 94 fps through a circular pipe with $f = 0.0171$. What is the shear stress at the pipe wall?

8.5.2 Steam with a specific weight of 38 N/m^3 is flowing with a velocity of 35 m/s through a circular pipe with $f = 0.0154$. What is the shear stress at the pipe wall?

8.5.3 Oil ($s = 0.93$) of viscosity 0.004 ft^2/sec flows in a 4-in-diameter pipe at a rate of 6.5 gpm. Find the head loss per unit length.

8.5.4 Oil ($s = 0.92$) of viscosity 0.00038 m^2/s flows in a 100-mm-diameter pipe at a rate of 0.64 L/s. Find the head loss per unit length.

8.6 FRICTION IN NONCIRCULAR CONDUITS

Most closed conduits we use in engineering practice are of circular cross section; however, we do occasionally use rectangular ducts and cross sections of other geometry. We can modify many of the equations in this chapter for application to noncircular sections by using the concept of hydraulic radius.

In Sec. 8.3 we defined the hydraulic radius as $R_h = A/P$, where A is the cross-sectional area and P is the wetted perimeter. Some equations include R_h, as in Sec. 8.18, in which case we can simply determine R_h for that conduit. But, because circular pipes are so common, many more equations instead use the pipe diameter, D. If the conduit of interest is not circular, we have no diameter to use in such equations, but we can find its hydraulic radius, R_h. Then we notice for a circular pipe flowing full, that

$$R_h = \frac{A}{P} = \frac{\pi D^2/4}{\pi D} = \frac{D}{4} \tag{8.20}$$

or

$$D = 4R_h \tag{8.21}$$

This provides us with an *equivalent* diameter, which we can substitute into Eq. (8.13) for example to yield

$$h_f = f\frac{L}{4R_h}\frac{V^2}{2g} \tag{8.22}$$

and into Eq. (8.1) for Reynolds number to give

Pipe flow:
$$\mathbf{R} = \frac{(4R_h)V\rho}{\mu} = \frac{(4R_h)V}{\nu} \tag{8.23}$$

and into any other pipe flow equations and charts that use D instead of R_h. We will meet many such equations and an important such chart later in this chapter, and there are more in Chap. 13.

This approach gives reasonably accurate results for turbulent flow, but the results are poor for laminar flow, because in such flow viscous action causes frictional phenomena throughout the body of the fluid, while in turbulent flow the frictional effect occurs largely in the region close to the wall; i.e., it depends on the wetted perimeter.

EXERCISES

8.6.1 When fluid of specific weight 46 lb/ft^3 flows in a 8-in-diameter pipe, the frictional stress between the fluid and the pipe wall is 0.65 lb/ft^2. Calculate the friction head loss per foot of pipe. If the flow rate is 1.8 cfs, how much power is lost per foot of pipe?

8.6.2 When fluid of specific weight 8.2 kN/m^3 flows in a 150-mm-diameter pipe, the frictional stress between the fluid and the pipe wall is 25 N/m^2. Calculate the friction head loss per meter of pipe. If the flow rate is 42 L/s, how much power is lost per meter of pipe?

8.7 LAMINAR FLOW IN CIRCULAR PIPES

In Sec. 2.11 we noted that for laminar flow $\tau = \mu\, du/dy$, where u is the value of the velocity at a distance y from the boundary. As $y = r_0 - r$, we also see that $\tau = -\mu\, du/dr$; in other words, the minus sign indicates that u decreases as r increases. The coefficient of viscosity μ is a constant for any particular fluid at a constant temperature, and therefore if the shear varies from zero at the center of the pipe to a maximum at the wall, it follows that the velocity profile must have a zero slope at the center and have a continuously steeper velocity gradient approaching the wall.

To determine the velocity profile for laminar flow in a circular pipe, we substitute the expression $\tau = \mu\, du/dy$ into Eq. (8.17). Thus

$$h_f = \frac{2\tau L}{r\gamma} = \mu \frac{du}{dy}\frac{2L}{r\gamma} = -\mu \frac{du}{dr}\frac{2L}{r\gamma}$$

From this,

$$du = -\frac{h_f \gamma}{2\mu L} r\, dr$$

Integrating and determining the constant of integration from the fact that $u = u_{\max}$ when $r = 0$, we obtain

$$u = u_{\max} - \frac{h_f \gamma}{4\mu L} r^2 = u_{\max} - kr^2 \tag{8.24}$$

From this equation we see that the velocity profile is a parabola, as shown in Fig. 8.3. Note that $k = h_f\gamma/4\mu L$.

At the wall we have the no-slip boundary condition (Sec. 2.11) that $u = 0$ when $r = r_0$. Substituting this into the second expression of Eq. (8.24) and noting that $u_{\max} = V_c$, the centerline velocity, we find $k = V_c/r_0^2$. Thus we can express Eq. (8.24) as

$$u = V_c - \frac{V_c}{r_0^2} r^2 = V_c\left(1 - \frac{r^2}{r_0^2}\right) \tag{8.25}$$

Combining Eqs. (8.24) and (8.25), we get an expression for the centerline velocity as follows

$$V_c = u_{\max} = \frac{h_f \gamma}{4\mu L} r_0^2 = \frac{h_f \gamma}{16\mu L} D^2 \tag{8.26}$$

We can multiply Eq. (8.24) by a differential area $dA = 2\pi r\, dr$ and integrate the product from $r = 0$ to $r = r_0$ to find the rate of discharge. From Eq. (4.3), the rate of discharge is equivalent to the volume of a solid bounded by the velocity profile. In this case the solid is a paraboloid with a maximum height of $u_{\max}$. The mean height of a paraboloid is one-half the maximum height, and hence the mean velocity V is $0.5u_{\max}$. Thus

$$V = \frac{h_f \gamma}{32\mu L} D^2 \tag{8.27}$$

From this last equation, noting that $\gamma = g\rho$ and $\mu/\rho = \nu$, the loss of head due to friction is

Laminar flow:

$$h_f = 32\frac{\mu}{\gamma}\frac{L}{D^2}V = 32\nu\frac{L}{gD^2}V \tag{8.28}$$

which is the ***Hagen-Poiseuille law*** for laminar flow in tubes. G. Hagen, a German engineer, experimented with water flowing through small brass tubes, and published his results in 1839. J. L. Poiseuille, a French scientist, experimented with water flowing through capillary tubes in order to determine the laws of flow of blood through the veins of the body, and published his studies in 1840.

From Eq. (8.28) we see that in laminar flow the loss of head is proportional to the first power of the velocity. This is verified by experiment, as shown in Fig. 8.1. The striking feature of this equation is that it involves no empirical coefficients or experimental factors of any kind, except for the physical properties of the fluid such as viscosity and density (or specific weight). From this, it would appear that in laminar flow the friction is independent of the roughness of the pipe wall. Experiments also bear out that this is true.

Dimensional analysis shows that we may express the friction loss by Eq. (8.13). Equating (8.13) and (8.28) and solving for the friction factor f, we obtain for laminar flow under pressure in a circular pipe,

Laminar flow:

$$f = \frac{64\nu}{DV} = \frac{64}{\mathbf{R}} \tag{8.29}$$

Thus, if $\mathbf{R}$ is less than 2000, we may use Eq. (8.28) to find pipe friction head loss, or we may use the pipe-friction equation (8.13) with the value of f as given by Eq. (8.29).

EXERCISES

8.7.1 An oil with kinematic viscosity of 0.004 ft^2/sec weighs 62 lb/ft^3. What will be its flow rate and head loss in a 2750-ft length of a 3-in-diameter pipe when the Reynolds number is 950?

8.7.2 With laminar flow in a circular pipe, at what distance from the centerline (in terms of the pipe radius) does the average velocity occur?

8.7.3 For laminar flow in a two-dimensional passage, find the relation between the average and maximum velocities.

8.8 ENTRANCE CONDITIONS IN LAMINAR FLOW

In the case of a pipe leading from a reservoir, if the entrance is rounded so as to avoid any initial disturbance of the entering stream, all particles will start to flow with the same velocity, except for a very thin film in contact with the wall. Particles in contact with the wall have zero velocity (the no-slip boundary condition, Sec. 2.11), but the velocity gradient there is extremely steep, and, with this slight exception, the velocity is uniform across the diameter, as shown in

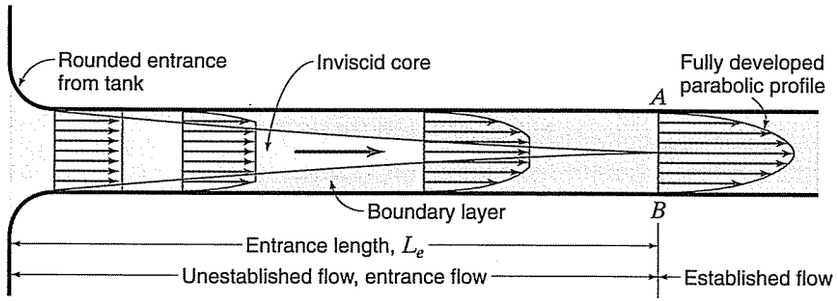


Figure 8.4
Velocity profiles and development of the boundary layer along a pipe in laminar flow.

Fig. 8.4. As the fluid progresses along the pipe, friction originating from the wall slows down the streamlines in the vicinity of the wall, but since Q is constant for successive sections, the velocity in the center must accelerate, until the final velocity profile is a parabola, as shown in Fig. 8.3. Theoretically, this requires an infinite distance, but both theory and observation have established that the maximum velocity in the center of the pipe will reach 99% of its ultimate value in a distance[4]

$$L_e = 0.058\mathbf{R}D \tag{8.30}$$

We call this distance the ***entrance length*** (Fig. 8.4). For the critical (maximum) value of $\mathbf{R} = 2000$ (Sec. 8.2), the entrance length L_e equals 116 pipe diameters. In other cases of laminar flow with Reynolds numbers less than 2000, the distance L_e will be correspondingly less, in accordance with Eq. (8.30).

Within the entrance length the flow is ***unestablished;*** that is, the velocity profile is changing. In this region (Fig. 8.4), we can visualize the flow as consisting of a central ***inviscid core*** in which there are no frictional effects, i.e., the flow is uniform, and an outer, annular zone extending from the core to the pipe wall. This outer zone increases in thickness as it moves along the wall, and is known as the ***boundary layer.*** Viscosity in the boundary layer acts to transmit the effect of boundary shear inwardly into the flow. At section AB the boundary layer has grown until it occupies the entire cross section of the pipe. At this point, for laminar flow, the velocity profile is a perfect parabola. Beyond section AB, for the same straight pipe the velocity profile does not change, and the flow is known as (laminar) ***established flow,*** or (laminar) ***fully developed flow.*** The flow will continue as fully developed so long as no change occurs to the straight pipe surface. When a change occurs, such as at a bend or other pipe fitting, the velocity profile will deform and will require some more flow length to return to established flow. Usually such fittings are so far apart that fully developed flow is common; but when they are close enough it is possible that established flow never occurs.

The concept of a boundary layer within which viscosity is important, and outside of which friction is unimportant and we can consider the fluid to be

[4] H. L. Langhaar, Steady Flow in the Transition Length of a Straight Tube, *J. Appl. Mech.,* Vol. 10, p. 55, 1942.

ideal, originated in 1904 with Ludwig Prandtl (1875–1953), a German engineering professor. Perhaps the single most significant contribution to fluid mechanics, this concept is particularly important with turbulent flow; we will discuss it further in Secs. 8.10–8.12 and Chap. 9.

As we saw in Sample Prob. 5.1 for a circular pipe, the kinetic energy of a stream with a parabolic velocity profile is $2V^2/2g$ (because $\alpha = 2$), where V is the mean velocity. At the entrance to the pipe the velocity is uniformly V across the diameter, except for an extremely thin layer next to the wall. Thus at the entrance to the pipe the kinetic energy per unit weight is practically $V^2/2g$. So in the distance L_e there is a continuous increase in kinetic energy accompanied by a corresponding decrease in pressure head. Therefore, at a distance L_e from the entrance with laminar flow, the piezometric head is less than the static value in the reservoir by $2V^2/2g$ plus the friction loss in this distance.

We have dealt with laminar flow rather fully, not merely because it is important in problems involving fluids of very high viscosity, but especially because it permits a simple and accurate rational analysis. This general approach will also help with the study of *turbulent* flow, where conditions are so complex that rigid mathematical treatment is impossible.

SAMPLE PROBLEM 8.2 For the case of Sample Prob. 8.1, find the centerline velocity, the velocity at $r = 20$ mm, the friction factor, the shear stress at the pipe wall, and the head loss per meter of pipe length.

Solution

From the solution to Sample Prob. 8.1: The flow is laminar.

Therefore, per Sec. 8.7: $V_c = 2V = 0.1273$ m/s ***ANS***

Eq. (8.24): $u = u_{\max} - kr^2 \qquad u_{\max} = V_c = 0.1273$ m/s

When $r = r_0 = 50$ mm, $u = 0$; hence $0 = 0.1273 - k(0.05)^2$, from which

$k = 50.9/(\text{m}\cdot\text{s}) \qquad u_{20\,\text{mm}} = 0.1273 - 50.9(0.02)^2 = 0.1070$ m/s ***ANS***

Eq. (8.29): $$f = \frac{64}{\mathbf{R}} = \frac{64}{354} = 0.1810 \qquad \textbf{ANS}$$

Eq. (8.19): $$\tau_0 = \frac{f}{4}\rho\frac{V^2}{2} = \frac{0.1810}{4}(850\ \text{kg/m}^3)\frac{(0.0637\ \text{m/s})^2}{2}$$
$$= 0.0779\ \text{kg/(m}\cdot\text{s}^2)$$
$$\tau_0 = 0.0779\frac{\text{kg}}{(\text{m}\cdot\text{s}^2)}\frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}} = 0.0779\ \text{N/m}^2 \qquad \textbf{ANS}$$

Eq. (8.14): $$S = \frac{h_f}{L} = f\frac{1}{D}\frac{V^2}{2g} = 0.1810\frac{1}{0.10\ \text{m}}\frac{(0.0637\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)}$$
$$= 0.000374\ \text{m/m} \qquad \textbf{ANS}$$

EXERCISES

8.8.1 In Exer. 8.2.3 what will be the approximate distance from the pipe entrance to the first point at which the flow is established?

8.8.2 In Exer. 8.5.4 what will be the approximate distance from the pipe entrance to the first point at which the flow is established?

8.9 TURBULENT FLOW

In Sec. 4.2 we saw that in laminar flow the fluid particles move in straight lines while in turbulent flow they follow random paths. Consider the case of laminar flow shown in Figs. 8.5(*a*, *b*), where the velocity u increases with y. Even though the fluid particles are moving horizontally to the right, because of molecular motion, molecules will cross line *ab* and transport momentum with them. On the average, the velocities of the molecules in the slower-moving fluid below the line will be less than those of the faster-moving fluid above; the result is that the molecules that cross from below tend to slow down the faster-moving fluid. Likewise, the molecules that cross the line *ab* from above tend to speed up the slower-moving fluid below. The result is a shear stress τ (tau) along the surface whose trace is *ab*. As given in Sec. 2.11, $\tau = \mu du/dy$. This equation is applicable to laminar flow only.

Let us examine some of the characteristics of turbulent flow to see how it differs from laminar flow. In turbulent flow the velocity at a point in the flow field fluctuates in both magnitude and direction.[5] We may observe these fluctuations in accurate velocity measurements (Sec. 11.4), and we commonly see their effects on pressure gages and manometers. The fluctuations result from a multitude of small eddies (Sec. 4.2), created by the viscous shear between adjacent particles. These eddies grow in size and then disappear as their particles

[5] We can best visualize the velocity at a point in a so-called "steady" turbulent flow as a vector that fluctuates in both direction and magnitude. The mean temporal velocity at that point corresponds to the "average" of those vectors.

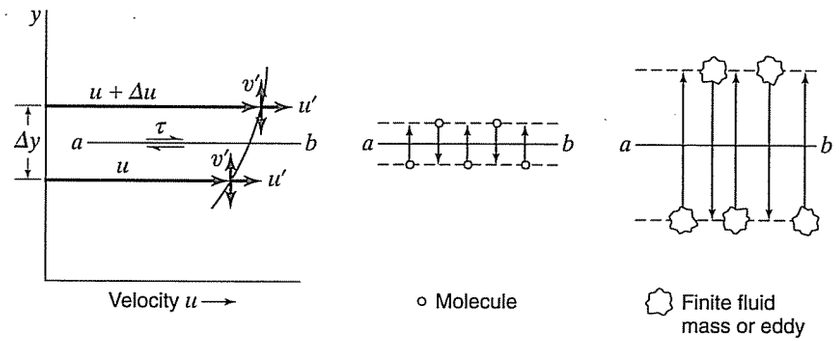


Figure 8.5
(*a*) Velocity profile. (*b*) Laminar flow (transfer of molecules across *ab*). (*c*) Turbulent flow (transfer of finite fluid masses across *ab*).

merge into adjacent eddies. Thus there is a continuous mixing of particles, with a consequent transfer of momentum. Viscosity dissipates mechanical energy, generating small amounts of heat.

First Expression

In the modern conception of turbulent flow, we assume a mechanism similar to that just described for laminar flow. However, we replace the molecules by minute but finite masses or eddies (Fig. 8.5*c*). So, by analogy, for turbulent flow we may define the shear stress along the plane through *ab* in Fig. 8.5 as

$$\text{Turbulent shear stress} = \eta \frac{du}{dy} \tag{8.31}$$

But unlike μ, the ***eddy viscosity*** η (eta) is not a constant for a given fluid at a given temperature, but it depends on the turbulence of the flow. We may view it as a coefficient of momentum transfer, expressing the transfer of momentum from points where the velocity is low to points where it is higher, and vice versa. Its magnitude may range from zero to many thousand times the value of μ. However, its numerical value is of less interest than its physical concept. In dealing with turbulent flow it is sometimes convenient to use ***kinematic eddy viscosity*** ε (epsilon) $= \eta/\rho$, which is a property of the flow alone, analogous to kinematic viscosity.

In general, the total shear stress in turbulent flow is the sum of the laminar shear stress plus the turbulent shear stress, i.e.,

$$\tau = \mu \frac{du}{dy} + \eta \frac{du}{dy} = \rho(\nu + \varepsilon)\frac{du}{dy} \tag{8.32}$$

With turbulent flow the second term of this equation is usually many times larger than the first term.

For turbulent flow we saw in Sec. 4.5 and Fig. 4.6 that the local axial velocity has fluctuations of plus and minus u', and there are also fluctuations of plus and minus v' and w' normal to u as shown in Fig. 8.6*b*. As it is obvious that there

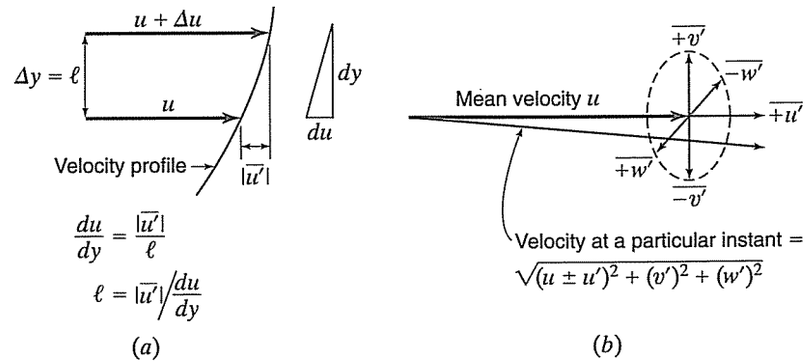


Figure 8.6
(*a*) Mixing length ℓ. (*b*) Instantaneous local velocity in turbulent flow.

can be no values of v' next to and perpendicular to a smooth wall, turbulent flow cannot exist there. Hence, near a smooth wall, the shear is due to laminar flow alone, and $\tau = \mu\, du/dy$. Note that the shear stress always acts to cause the velocity distribution to become more uniform.

At some distance from the wall, such as $0.2r$, the value of du/dy becomes small in turbulent flow, and so the viscous shear becomes negligible in comparison with the turbulent shear. The latter can be large, even though du/dy is small, because of the possibility of η being very large. This is because of the great turbulence that may exist at an appreciable distance from the wall. But at the center of the pipe, where du/dy is zero, there can be no shear at all. So, in turbulent flow as well as in laminar flow, the shear stress is a maximum at the wall and decreases linearly to zero at the axis, as shown in Fig. 8.3 and proved in Sec. 8.5.

Second Expression

We can obtain another expression for turbulent shear stress that is different from Eq. (8.31). In Fig. 8.5*a*, if a mass m of fluid below ab, where the temporal mean axial velocity is u, moves upward into a zone where the temporal mean axial velocity is $u + \Delta u$, its initial momentum in the axial direction must increase by $m\,\Delta u$. Conversely, when a mass m moves from the upper zone to the lower its axial momentum will decrease by $m\,\Delta u$. So this transfer of momentum back and forth across ab will produce a shear in the plane through ab proportional to Δu. This shear is possible only because of the velocity profile shown. If the latter were vertical, Δu would be zero and there could be no shear.

If the distance Δy in Fig. 8.5*a* is chosen so that the average value of $+u'$ in the upper zone over a time period long enough to include many velocity fluctuations is equal to Δu, i.e., $\Delta u = \overline{|u'|}$, the distance between the two streams will be what is known as the ***mixing length*** ℓ, which we will discuss shortly. Consider, over a short time interval, a mass moving upward from below ab with a velocity v'; it will transport into the upper zone, where the velocity is $u + u'$, a momentum per unit time which is on the average equal to $\rho(v'dA)(u)$. The slower-moving mass from below ab will tend to retard the flow above ab; this creates a shear force along the plane of ab. We can find this force by applying the momentum principle, Eq. (6.6),

$$F = \tau dA = \rho Q(\Delta V) = \rho(v'dA)(u + u' - u) = \rho u'v'dA$$

So, over a period of time sufficiently long to include a large number of velocity fluctuations, the shear stress is given by

$$\tau = F/dA = -\rho\overline{u'v'} \tag{8.33}$$

where $\overline{u'v'}$ is the temporal average of the product of u' and v'. This is an alternate form for Eq. (8.31), and in modern turbulence theory $-\rho\overline{u'v'}$ is known as the ***Reynolds stress.***

The minus sign appears in Eq. (8.33) because the product $\overline{u'v'}$ on the average is negative. By inspecting Fig. 8.5*a*, we can see that $+v'$ is associated with $-u'$ values more than with $+u'$ values. The opposite is true for $-v'$. Even though the temporal mean values of u' and v' are individually equal to zero, the

temporal mean value of their product is not zero. This is because combinations of $+v'$ and $-u'$ and of $-v'$ and $+u'$ predominate over combinations of $+v'$ and $+u'$ and $-v'$ and $-u'$, respectively.

Prandtl (Sec. 8.8) reasoned that in any turbulent flow $\overline{|u'|}$ and $\overline{|v'|}$ must be proportional to each other and of the same order of magnitude. He also introduced the concept of mixing length ℓ, which is the distance transverse to the flow direction such that $\Delta u = \overline{|u'|}$. From Fig. 8.6*a* we can see that $\Delta u = \ell\, du/dy$ and so $\overline{|u'|} = \ell\, du/dy$. If $\overline{|u'|} \propto \overline{|v'|}$ and if we allow ℓ to account for the constant of proportionality, Prandtl[6] showed that $-\overline{u'v'}$ varies as $\ell^2(du/dy)^2$. Thus

$$\tau = -\rho\overline{u'v'} = \rho\ell^2\left(\frac{du}{dy}\right)^2 \tag{8.34}$$

This equation expresses terms that we can measure. Thus in any experiment that determines the pipe friction loss, we can compute $\overline{\tau}_0$ from Eq. (8.8), and then we can find τ at any radius from Eq. (8.18). A traverse of the velocity across a pipe diameter will give u at any radius, and the velocity profile will give du/dy at any radius. Thus Eq. (8.34) allows us to find the mixing length ℓ as a function of the pipe radius. The purpose of all of this is to help us develop theoretical equations for the velocity profile in turbulent flow, and from this in turn to develop theoretical equations for f, the friction coefficient.

EXERCISES

8.9.1 Tests on 70°F water flowing through a 9-in-diameter pipe showed that when $V = 13$ fps, $f = 0.0162$. (*a*) If, at a distance of 3 in from the center of the pipe, $\tau = 0.388$ psf, and the velocity profile gives a value for du/dy of 6.97/sec, find at that radius (*a*) the viscous shear, (*b*) the turbulent shear, and (*c*) the mixing length ℓ.

8.9.2 Water at 20°C flows through a 240-mm-diameter pipe. Tests have determined that at a distance of 60 mm from the pipe centerline the mixing length ℓ is 19.8 mm and from the velocity profile $du/dy = 5.33\ \text{s}^{-1}$. Find at that radius (*a*) the total shear stress, (*b*) the eddy viscosity, (*c*) the viscous shear, and (*d*) the turbulent shear.

8.10 VISCOUS SUBLAYER IN TURBULENT FLOW

In Fig. 8.4 we saw that, for *laminar* flow, if the fluid enters a pipe with no initial disturbance, the velocity is uniform across the diameter except for an exceedingly thin film at the wall, because the velocity touching any wall is zero (the no-slip condition, Sec. 2.11). But as flow proceeds down the pipe, the velocity profile changes because of the growth of a laminar boundary layer, which continues until the boundary layers from opposite sides meet at the pipe axis and then there is fully developed (or established) laminar flow.

[6] H. Schlichting, *Boundary Layer Theory,* 7th ed., McGraw-Hill Book Co., 1987, p. 605.

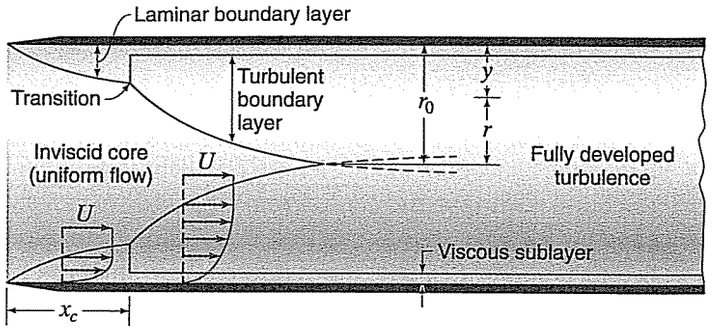


Figure 8.7
Development of boundary layer in a pipe where fully developed flow is turbulent (scales much distorted).

If the Reynolds number is above the critical value (Sec. 8.2), so that the developed flow is turbulent, the initial condition is much like that in Fig. 8.4. But as the laminar boundary layer increases in thickness, at a certain point a transition occurs and the boundary layer becomes turbulent (Fig. 8.7). This transition occurs where the length x_c of the laminar portion of the boundary layer is about equal to $500{,}000\nu/U$, where U is the uniform velocity (i.e., where $\mathbf{R}_x = Ux/\nu \approx 500{,}000$). After the transition, the ***turbulent boundary layer*** generally increases in thickness much more rapidly. So the length of the ***inviscid core,*** to where the two opposite layers meet, is relatively shorter.

The development of turbulent flow is considerably more complex than that of laminar flow (Sec. 8.8). Although the length of the inviscid core is relatively short with turbulent flow, it takes about four times this length for the velocity profile to become fully developed, and 8 to 12 times this length for the detailed structure of the turbulence to become fully developed. Only when all these aspects are complete do we have ***fully developed turbulent flow.*** These various developments depend on a variety of features, including the entrance conditions (Sec. 8.21) and the wall roughness (Secs. 8.10–8.12); as a result no single equation exists to predict the ***entrance length*** for turbulent flow, i.e., the length over which the flow is developing. But as an approximate guide, we may observe that the velocity profile is generally fully developed within 20 to 40 pipe diameters.

In all that now follows we shall consider only *fully* developed turbulent flow.

As v' must be zero at a smooth wall, turbulence there is inhibited so that a laminar-like sublayer occurs immediately next to the wall. However, the adjacent turbulent flow does repeatedly induce random transient effects that momentarily disrupt this sublayer, even though they fade rapidly. Because it is therefore not a true laminar layer, and because shear in this layer is predominantly due to viscosity alone, we call it a ***viscous sublayer*** (see Fig. 8.7). This viscous sublayer is extremely thin, usually only a few hundredths of a millimeter, but its effect is great because of the very steep velocity gradient within it and because $\tau = \mu\, du/dy$ in that region. At a greater distance from the wall the

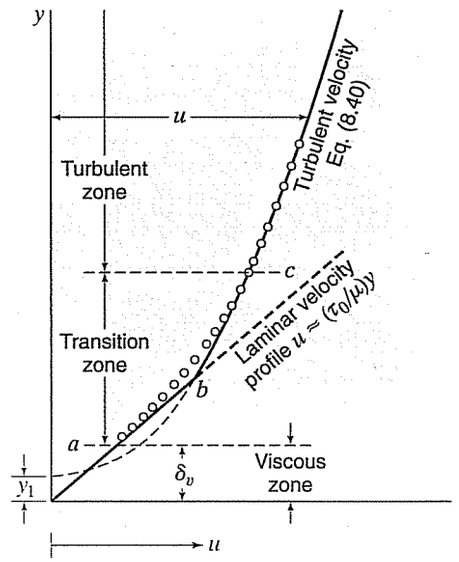


Figure 8.8
Velocity profile near a solid wall (vertical scale greatly exaggerated). Theoretical relations (solid lines) are compared with experimental data (small circles).

viscous effect becomes negligible, but the turbulent shear is then large. Between the two, there must be a transition zone where both types of shear are significant. It is evident that there cannot be sharp lines of demarcation separating these three zones, instead one must merge gradually into the other.

By plotting one velocity profile from the wall on the assumption that the flow is entirely laminar (Sec. 8.7) and plotting another velocity profile on the assumption that the flow is entirely turbulent (Sec. 8.11), the two will intersect, as shown in Fig. 8.8. It is obvious that there can be no abrupt change in profile at this point of intersection, but that one curve must merge gradually into the other with some kind of transition, which is in fact what happens as shown by the experimental points.

When studying such velocity profiles, we find that the quantity $\sqrt{\tau_0/\rho}$ frequently occurs. Because it has the dimensions of velocity, scientists have named it the ***shear-stress velocity*** (or ***friction velocity***) u_*, although it is not actually a flow velocity.

When the flow in a circular pipe is entirely laminar, we have seen that the velocity profile is a parabola (Fig. 8.3). But when there is only an extremely thin film closest to the wall where viscous shear dominates, we can scarcely distinguish the velocity profile in it from a straight line. If we ignore the momentary fluctuations in this viscous sublayer, with such a linear velocity profile Eq. (2.9) at the wall becomes

$$\tau_0 = \mu \frac{u}{y} \qquad \text{or} \qquad \frac{\tau_0}{\rho} = \frac{\nu u}{y}$$

But from the definition of u_*, we have $u_*^2 = \tau_0/\rho$, so, by eliminating τ_0/ρ, we obtain

$$\frac{\nu u}{y} = u_*^2$$

or

$$\frac{u}{u_*} = \frac{y u_*}{\nu} \tag{8.35}$$

which is known as a ***law of the wall.*** This linear relation for $u(y)$ approximates experimental data well in the range $0 \leqslant yu_*/\nu \leqslant 5$. If we call this imprecise, but commonly accepted, upper limiting value of y the thickness of the viscous sublayer, δ_v, then

$$\delta_v = 5\nu/u_* \tag{8.36}$$

The transition zone appears to extend from a to c in Fig. 8.8. For the latter point, the value of y seems to be about $70\nu/u_*$ or $14\delta_v$. Beyond this, the flow is so turbulent that viscous shear is negligible.

Noting from Eq. (8.19) that

$$u_* = \sqrt{\frac{\tau_0}{\rho}} = V\sqrt{\frac{f}{8}} \tag{8.37}$$

and that the Reynolds number $\mathbf{R} = DV/\nu$, we see that when $yu_*/\nu = 5$, or $y = \delta_v$,

$$\delta_v = \frac{14.14\nu}{V\sqrt{f}} = \frac{14.14D}{\mathbf{R}\sqrt{f}} \tag{8.38}$$

From this, we see that the higher the velocity or the lower the kinematic viscosity, the thinner is the viscous sublayer. Thus, for a given constant pipe diameter, the thickness of the viscous sublayer decreases as the Reynolds number increases.

Now we can consider what is meant by a smooth wall and a rough wall. There is no such thing in reality as a mathematically smooth surface. But if the irregularities on any actual surface are small enough that the effects of the projections do not pierce through the viscous sublayer (Fig. 8.8), the surface is ***hydraulically smooth*** from the fluid-mechanics viewpoint. If the effects of the projections extend beyond the sublayer, the laminar layer is broken up and the surface is no longer hydraulically smooth. If the surface roughness projections are large enough to protrude right through the transition layer, it is totally broken up. The resulting flow is completely turbulent, known as ***fully rough (-pipe) flow,*** and friction is independent of Reynolds number. We shall discuss the significance of this in Sec. 8.11. If the roughness projections protrude only partially into the transition layer, we say the flow is ***transitionally rough,*** and there is a moderate Reynolds number effect.

To be more specific, if e is the equivalent height of the roughness projections then for $eu_*/\nu < 5$ (or $e < \delta_v$) the viscous sublayer completely buries the

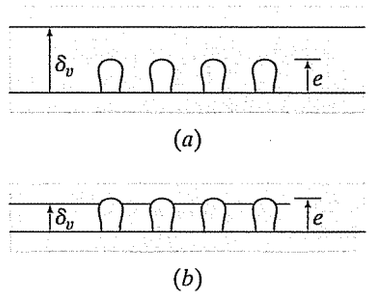


Figure 8.9
Turbulent flow near a boundary. (a) Low $\mathbf{R}$, $\delta_v > e$; the pipe behaves as a smooth pipe. (b) Relatively high $\mathbf{R}$, $\delta_v < e$; if $\delta_v < \frac{1}{14}e$, the pipe behaves as a fully rough pipe.

surface roughness, the roughness has no effect on friction, and the pipe is hydraulically smooth. If $eu_*/\nu > 70$ (or $e > 14\delta_v$), the pipe will behave as fully rough. In the region between these values, i.e., when the roughness projections are such that $5 \leqslant eu_*/\nu \leqslant 70$ (or $\delta_v \leqslant e \leqslant 14\delta_v$), the pipe will behave in a transitional mode, neither hydraulically smooth nor fully rough. Most engineering pipe flows fall in this range.

Because the thickness of the viscous sublayer in a given pipe decreases with an increase in Reynolds number, we see that the same pipe may be hydraulically smooth at low Reynolds numbers and rough at high Reynolds numbers. Thus even a relatively smooth pipe may behave as a rough pipe if the Reynolds number is high enough. It is also apparent that, with increasing Reynolds number, there is a gradual transition from smooth to rough-pipe flow. Figure 8.9 depicts these concepts schematically.

EXERCISES

8.10.1 Compute δ_v for the data of Sample Prob. 8.3.

8.10.2 Water in a pipe ($f = 0.018$) is at a temperature of 70°F. (a) If the mean velocity is 14 fps, what is the nominal thickness δ_v of the viscous sublayer? (b) What will δ_v be if we increase the velocity to 24 fps and f does not change?

8.10.3 Water in a pipe ($f = 0.012$) is at a temperature of 15°C. (a) If the mean velocity is 3.2 m/s, what is the nominal thickness δ_v of the viscous sublayer? (b) What will δ_v be if we increase the velocity to 5.5 m/s and f does not change?

8.10.4 For the data of Exer. 8.10.2(a), what is the distance from the wall to the assumed limit of the transition region where true turbulent flow begins?

8.10.5 For the data in Exer. 8.10.3(a), what is the distance from the wall to the assumed limit of the transition region where true turbulent flow begins?

8.10.6 Water at 50°C flows in a 150-mm-diameter pipe with $V = 6.5$ m/s and $e = 0.14$ mm. Head loss measurements indicate that $f = 0.020$. (a) What is the thickness of the viscous sublayer? (b) Is the pipe behaving as a fully rough pipe?

8.11 VELOCITY PROFILE IN TURBULENT FLOW

Prandtl (Secs. 8.8 and 8.9) reasoned that turbulent flow in a pipe is strongly influenced by the flow phenomena near the wall. In the vicinity of the wall, $\tau \approx \tau_0$. He assumed that the mixing length ℓ (Sec. 8.9) near the wall was proportional to the distance from the wall, that is, $\ell = Ky$. Experiments have confirmed this, and have determined that $K = 0.40$. Using this relationship in Eq. (8.34), we get

$$\tau \approx \tau_0 = \rho\ell^2\left(\frac{du}{dy}\right)^2 = \rho K^2 y^2\left(\frac{du}{dy}\right)^2$$

or

$$du = \frac{1}{K}\sqrt{\frac{\tau_0}{\rho}}\frac{dy}{y} = \frac{u_*}{K}\frac{dy}{y}$$

Integrating, and inserting 0.4 for K, we obtain

$$u = 2.5u_* \ln y + C$$

We can evaluate the constant C by noting that $u = u_{max}$ = the centerline velocity when $y = r_0$ = the pipe radius. Substituting the expression for C, we get

$$\frac{u_{max} - u}{u_*} = 2.5\ln\frac{r_0}{y} \tag{8.39}$$

This is known as the ***velocity defect law,*** because we call $(u_{max} - u)$ the velocity defect. Replacing y by $r_0 - r$, and changing the base e logarithm (ln) to a base 10 logarithm (log), the equation becomes

$$u = u_{max} - 2.5u_*\ln\frac{r_0}{r_0 - r} = u_{max} - 5.76u_*\log\frac{r_0}{r_0 - r} \tag{8.40}$$

Although this equation is derived by assuming certain relations very near to the wall, it holds almost as far out as the pipe axis.

Starting with the derivation of Eq. (8.34), this entire development is open to argument at nearly every step. But the fact remains that Eq. (8.40) agrees very closely with actual measurements of velocity profiles for both smooth and rough pipes. However, there are two zones in which the equation is defective. At the axis of the pipe, *du/dy* must be zero. But Eq. (8.40) is logarithmic and does not have a zero slope at $r = 0$, and hence the equation gives a velocity profile with a sharp point (or cusp) at the axis, whereas in reality it is rounded at the axis. This discrepancy affects only a very small area and causes only a very slight error when computing the rate of discharge using Eq. (8.40).

Equation (8.40) is also not applicable very close to the wall. In fact, it indicates that when $r = r_0$, the value of u is minus infinity. The equation indicates that $u = 0$, not at the wall, but at a small distance from it, shown as y_1 in Fig. 8.8.

However, this discrepancy is well within the confines of the viscous sublayer, where the equation is not supposed to apply, and where we have Eq. (8.35). In the intervening transition or overlap zone (Fig. 8.8), where both viscous and turbulent shear are important, investigators have found that experimental velocity profile data follow a logarithmic relation

$$\frac{u}{u_*} = 2.5\ln\left(\frac{yu_*}{\nu}\right) + 5.0 \tag{8.41}$$

Although Eq. (8.40) is not perfect, it reliably fits the data except for the two small areas mentioned, where it is still close. So we can determine the rate of discharge Q with a high degree of accuracy by using the value of u given by Eq. (8.40) and integrating over the area of the pipe. Thus

$$Q = \int u\,dA = 2\pi\int_0^{r_0} ur\,dr$$

Substituting from the first expression of Eq. (8.40) for u, integrating and dividing by the pipe area πr_0^2, the mean velocity is[7]

$$V = u_{max} - 2.5u_*\left[\ln r_0 - \frac{2}{r_0^2}\int_0^{r_0} r\ln(r_0 - r)\,dr\right]$$

Making use of Eq. (8.37), this equation reduces to

$$V = u_{max} - \tfrac{3}{2}\times 2.5u_* = u_{max} - 1.326V\sqrt{f} \tag{8.42}$$

From Eq. (8.42) we can obtain the ***pipe factor,*** which is the ratio of the mean to the maximum velocity. It is

$$\text{Pipe factor} = \frac{V}{u_{max}} = \frac{1}{1 + 1.326\sqrt{f}} \tag{8.43}$$

Using Eq. (8.43) to eliminate u_{max} from Eq. (8.40) and using Eq. (8.37) to eliminate u_*, the result is

$$u = (1 + 1.326\sqrt{f})V - 2.04\sqrt{f}\,V\log\frac{r_0}{r_0 - r} \tag{8.44}$$

[7] The integral results in indeterminate values at $r = r_0$, as we should expect, since the equation for u does not really apply close to the wall. However, for all practical purposes, these reduce to negligible quantities.

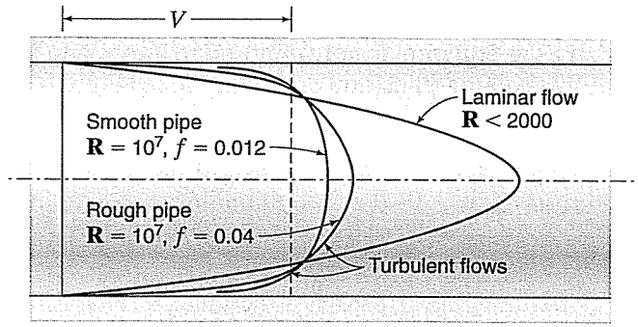


Figure 8.10
Velocity profiles across a pipe for equal flow rates. The turbulent profiles are plotted from Eq. (8.44).

which enables us to plot a velocity profile for any mean velocity and any value of f in turbulent flow. In Fig. 8.10 profiles for both a smooth and a rough pipe are plotted from this equation. The only noticeable difference between these and measured profiles is that the latter are more rounded at the axis of the pipe.[8] Of course, the measured turbulent profiles also exhibit turbulent fluctuations everywhere except near the walls.

Comparing the turbulent-flow velocity profiles with the laminar-flow velocity profile in Fig. 8.10 we see the turbulent-flow profiles are much flatter near the central portion of the pipe and steeper near the wall. We also notice that the turbulent profile for the smooth pipe is flatter near the center (i.e., blunter) than for the rough pipe. In contrast, the velocity profile in laminar flow is independent of pipe roughness.

As we have now derived a theoretical equation for the velocity profile for turbulent flow in circular pipes, we can also derive equations for the kinetic-energy- and momentum-correction factors (Secs. 5.1 and 6.3) using mean velocities. Respectively, these equations are[9]

$$\alpha = 1 + 2.7f \tag{8.45a}$$

$$\beta = 1 + 0.98f \tag{8.45b}$$

[8] Although the preceding theory agrees very well with experimental data, it is not absolutely correct throughout the entire range from the axis to the pipe wall, and some slight shifts in the numerical constants could improve agreement with test data. Thus in Eqs. (8.43) and (8.44) we can replace the 1.326 by 1.44, and in Eq. (8.44) although many writers use 2 instead of 2.04, a better practical value seems to be 2.15.

[9] L. F. Moody, Some Pipe Characteristics of Engineering Interest, *Houille Blanche,* May–June, 1950.

SAMPLE PROBLEM 8.3 The pipe friction head loss in 200 ft of 6-in-diameter pipe is 25 ft·lb/lb when oil ($s = 0.90$) of viscosity 0.0008 lb·sec/ft^2 flows at 2.0 cfs. Determine the centerline velocity, the shear stress at the wall of the pipe, and the velocity at 2 in from the centerline.

Solution
First determine whether the flow is laminar or turbulent:

Eq. (4.7): $$V = \frac{4Q}{\pi D^2} = \frac{4(20)}{\pi(0.5)^2} = 10.19 \text{ fps}$$

Eq. (8.1): $$\mathbf{R} = \frac{DV\rho}{\mu} = \frac{0.5(10.19)(0.9 \times 1.940)}{0.0008} = 11{,}120$$

Since $\mathbf{R} > 2000$, the flow is turbulent.

From Eq. (8.13): $$f = \frac{h_f D(2g)}{LV^2} = \frac{25(0.5)2(32.2)}{200(10.19)^2} = 0.0388$$

From Eq. (8.43), $u_{\max} = 10.19(1 + 1.326\sqrt{0.0388}) = 12.85$ fps ***ANS***

Eq. (8.19): $$\tau_0 = \frac{f\rho V^2}{8} = \frac{0.0388(0.9 \times 1.940)(10.19)^2}{8} = 0.878 \text{ lb/ft}^2$$ ***ANS***

Eq. (8.37): $$u_* = V\sqrt{\frac{f}{8}} = 10.19\sqrt{\frac{0.0388}{8}} = 0.709 \text{ fps}$$

Finally, from Eq. (8.40),

$$u_{2\text{in}} = u_{\max} - 5.76(0.709)\log\tfrac{3}{1} = 12.85 - 1.948 = 10.90 \text{ fps}$$ ***ANS***

Note: $u_{\max}/V = 12.85/10.19 = 1.261$. If the flow had been laminar, the velocity profile would have been parabolic and $u_{\max}/V$ would have been 2 (Sec. 8.7).

EXERCISES

8.11.1 In a 1.00-m-diameter pipe velocities are measured as 5.35 m/s on the centerline and 4.91 m/s at $r = 70$ mm. Approximately what is the flow rate?

8.11.2 For turbulent flow in a circular pipe, find r/r_0 at the radial distance from the centerline where the mean velocity occurs.

8.11.3 Oil ($s = 0.92$) with a viscosity of 0.000 65 lb·sec/ft^2 flows at a rate of 6 cfs through a 4-in-diameter pipe having $f = 0.040$. Find the friction head loss. Determine the shear stress at the pipe wall and the velocity at 1.5 in from the centerline.

8.12 PIPE ROUGHNESS

Unfortunately, there is as yet no scientific way of measuring or specifying the roughness of commercial pipes. Several experimenters have worked with pipes with artificial roughness produced by various means so that the roughness could be measured and described by geometric factors, and they have proved that the friction depends not only on the size and shape of the projections, but also on their distribution or spacing. We have much more to do before we completely solve this problem.

Noteworthy efforts in this direction were made in 1933 by a German engineer, J. Nikuradse, a student of Prandtl's. He coated several different sizes of pipe with sand grains that he had sorted by sieving into different grain sizes of reasonably uniform diameters. Let us represent the diameters of the sand grains by e, which is known as the ***absolute roughness.*** In Sec. 8.4 dimensional analysis of pipe flow showed that for a smooth-walled pipe the friction factor f is a function of Reynolds number. A more general approach, including e as a parameter, reveals that $f = \phi(\mathbf{R}, e/D)$. The term e/D is known as the ***relative roughness.*** In his experimental work Nikuradse had values of e/D ranging from 0.000985 to 0.0333.

In the case of artificial roughness such as this, the roughness is uniform, whereas in commercial pipes it is irregular both in size and in distribution. However, we may describe the irregular roughness of commercial pipe by a single e value, if we understand this means that the pipe has the same value of f at a high Reynolds number that it would have for a smooth pipe coated with sand grains of uniform size e. This "equivalent" grain size e must be close to the mean size of the irregular roughness elements.

Investigators working with pipes have found that if the thickness of the viscous sublayer $\delta_v > e$ (i.e., $eu_*/\nu < 5$), the viscous sublayer completely submerges the effect of e (see Sec. 8.10). Prandtl (Secs. 8.8 and 8.11), using information from Eq. (8.40) and data from Nikuradse's experiments, developed an equation for the friction factor for such a case:

$$\text{Smooth-pipe flow:} \qquad \frac{1}{\sqrt{f}} = 2\log\left(\frac{\mathbf{R}\sqrt{f}}{2.51}\right) \tag{8.46}$$

This equation applies to turbulent flow in any pipe as long as $\delta_v > e$; when this condition prevails, the flow is known as ***smooth-pipe flow.*** Since e does not appear in the equation, it is as though $e = 0$. Users have found the equation to be reliable for smooth pipes for all values of $\mathbf{R}$ over 4000. For such pipes, i.e., drawn tubing, brass, glass, etc., we can extrapolate it with confidence for values of $\mathbf{R}$ far beyond any present experimental values because it is functionally correct, assuming the wall surface is so smooth that the effects of the projections do not pierce the viscous sublayer, which becomes increasingly thinner with increasing $\mathbf{R}$. That this is so is evident from the fact that the formula yields a value of $f = 0$ for $\mathbf{R} = \infty$. This agrees with the facts, because $\mathbf{R}$ is infinite for a fluid of zero viscosity, and for such a case f must be zero.

Because of the way that f appears in two places in Eq. (8.46), it is ***implicit*** in f (see Appendix B) and hard to solve; we must use either iteration or a graph of f versus $\mathbf{R}$. However, as suggested by Colebrook,[10] we can approximate it by the ***explicit*** equation

$$\text{Smooth-pipe flow:} \qquad \frac{1}{\sqrt{f}} = 1.8\log\left(\frac{\mathbf{R}}{6.9}\right) \tag{8.47}$$

which differs from Eq. (8.35) by less than $\pm 1.5\%$ for $4000 \leq \mathbf{R} \leq 10^8$.

Blasius[11] has shown that for Reynolds numbers between 3000 and 10^5 we can approximately express the friction factor for a smooth pipe as

$$\text{Blasius, smooth pipe, } 3000 \leq \mathbf{R} \leq 10^5: \qquad f = \frac{0.316}{\mathbf{R}^{0.25}} \tag{8.48}$$

Sometimes we can use this very conveniently to simplify equations. Blasius also found that over the same range of Reynolds numbers, the velocity profile in a smooth pipe closely agrees with the expression

$$\text{Blasius, smooth pipe, } 3000 \leq \mathbf{R} \leq 10^5: \qquad \frac{u}{u_{\max}} = \left(\frac{y}{r_0}\right)^{1/7} \tag{8.49}$$

where $y = r_0 - r$, the distance from the pipe wall. We call this equation the ***seventh-root law*** for turbulent-velocity distribution. Though it is not absolutely accurate, it is useful because it is easy to work with mathematically. At Reynolds numbers above 10^5 we must use an exponent somewhat smaller than $\frac{1}{7}$ to give good results.

At high Reynolds numbers δ_v becomes much smaller, and the roughness elements protrude through the viscous sublayer as in Fig. 8.9*b*. If $\delta_v < \frac{1}{14}e$ (i.e., $eu_*/\nu > 70$), investigators have found that the flow behaves as ***fully-rough-pipe flow,*** i.e., the friction factor is independent of the Reynolds number. For such a case von Kármán found that we can express the friction factor as

$$\text{Fully-rough-pipe flow } (f_{\min}): \qquad \frac{1}{\sqrt{f}} = 2\log\left(\frac{3.7}{e/D}\right). \tag{8.50}$$

The values of f from this equation correspond to the right-hand side of the Moody chart (Fig. 8.11), where the curves become horizontal. We sometimes refer to these values as $f_{\min}$.

[10] C. F. Colebrook, Turbulent Flow in Pipes, with Particular Reference to the Transition Region between the Smooth and Rough Pipe Laws, *J. Inst. Civil Engrs.* (*London*), Vol. 11, February 1939.

[11] H. Blasius, Das Ähnlichkeitsgesetz bei Reibungsvorgängen in Flüssigkeiten, *Forsch. Gebiete Ingenieurw.,* Vol. 131, 1913.

In the interval where $e > \delta_v > \frac{1}{14}e$ (i.e., $5 < eu_*/\nu < 70$) neither smooth flow Eq. (8.46) nor fully rough flow Eq. (8.50) applies. In 1939 Colebrook[12] combined Eqs. (8.46) and (8.50) to yield

Turbulent flow, all pipes (Colebrook):
$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{e/D}{3.7} + \frac{2.51}{\mathbf{R}\sqrt{f}}\right) \tag{8.51}$$

Besides providing a good approximation to conditions in the intermediate range, for $e = 0$ the Colebrook equation reduces to the smooth-pipe equation (8.46), and for large $\mathbf{R}$ it reduces to the fully-rough-pipe equation (8.50). Thus it applies to *all* turbulent flow conditions. Values of friction factor f that it predicts are generally accurate to within 10–15% of experimental data. This equation is so useful that engineers have long used it as the accepted design formula for turbulent flow; however, it has one major disadvantage. Like Eq. (8.46), it is implicit in f (Appendix B), which makes it inconvenient to use to manually evaluate f. More recently, in 1983 Haaland[13] combined Eqs. (8.47) and (8.50) to provide another approximation,

Turbulent flow, all pipes (Haaland):
$$\frac{1}{\sqrt{f}} = -1.8\log\left[\left(\frac{e/D}{3.7}\right)^{1.11} + \frac{6.9}{\mathbf{R}}\right] \tag{8.52}$$

which has the advantage of being explicit in f; it has the same asymptotic behavior as Eq. (8.51), from which it differs by less than ±1.5% for $4000 \leq \mathbf{R} \leq 10^8$.

EXERCISES

8.12.1 Using the implicit equation (8.46), the approximate equation (8.47), and Blasius' equation (8.48), solve for the smooth-pipe friction factor f using Reynolds numbers of (*a*) 4000, (*b*) 20,000, and (*c*) 10^5. (*d*) For which of these three values do the equations show the most variation in f?

8.12.2 Substitute into Eq. (8.51) the given and computed data of Sample Prob. 8.5*a*. How well does the right-hand side of the equation agree with the left-hand side?

8.12.3 Repeat Exer. 8.12.2 using Eq. (8.52).

8.13 CHART FOR FRICTION FACTOR

The preceding equations for f have been very inconvenient to use in a number of circumstances, which we will discuss further in coming sections, and this was especially true before Haaland's equation appeared. The inconvenience was largely overcome by reading numerical values from a chart (Fig. 8.11), prepared

[12] See Footnote 10.

[13] S. E. Haaland, Simple and Explicit Formulas for the Friction Factor in Turbulent Pipe Flow, *J. Fluids Eng.*, Vol. 105, March 1983.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

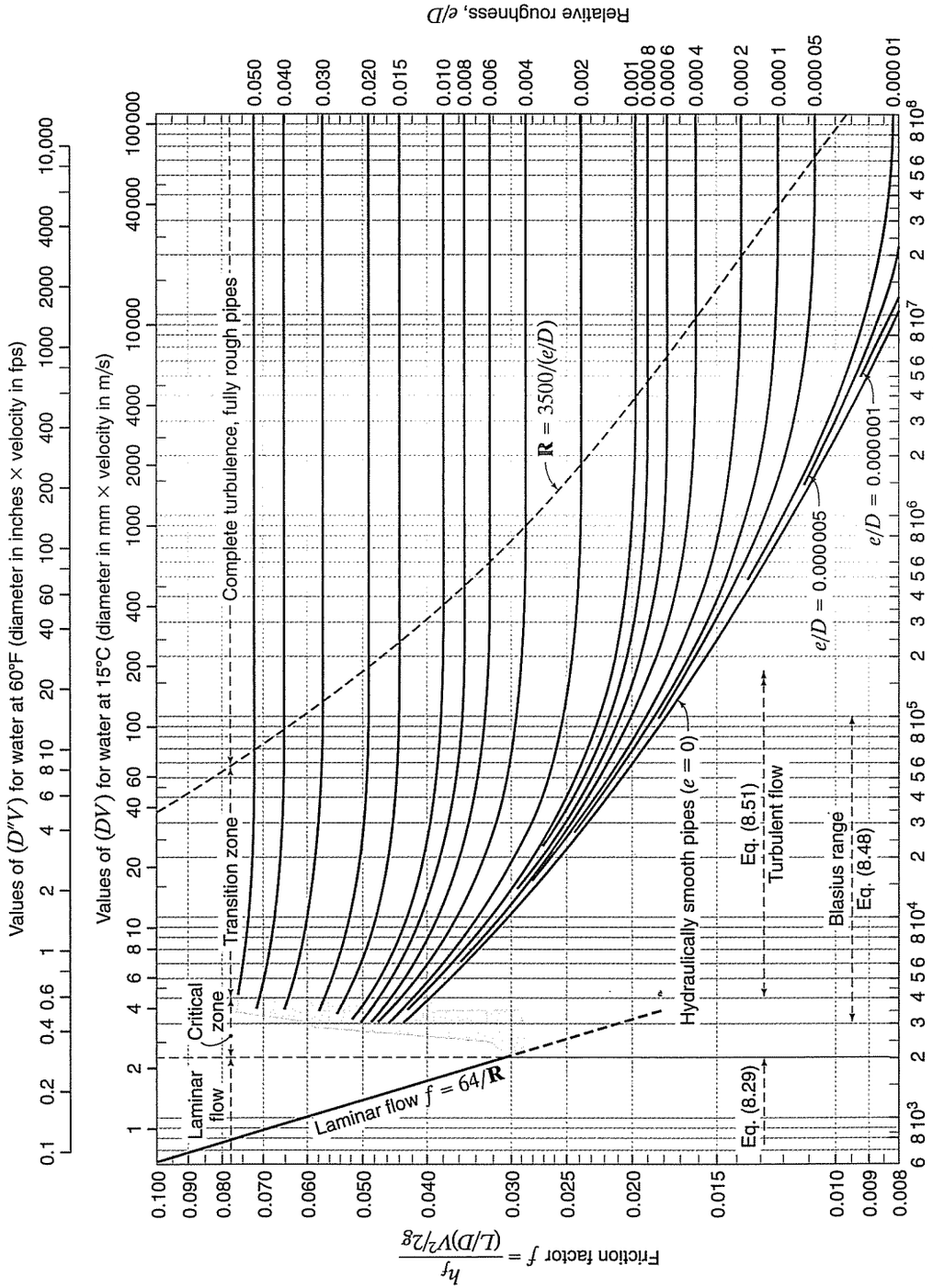


Figure 8.11
Moody chart for pipe friction factor (Stanton diagram).

by Moody[14] in 1944. The chart, often called the ***Moody diagram,***[15] is based on the best information available, and was plotted with the aid of Eqs. (8.29) and (8.51). All the quantities involved are dimensionless, so the chart may be used for both BG and SI unit systems. For convenience, BG values of DV (diameter times velocity) for water at 60°F and similar SI values for water at 15°C are given across the top of the chart to save the need to compute Reynolds number for those common cases.

The Moody chart, and the various flow conditions that it represents, divides into four zones: the ***laminar-flow zone;*** a ***critical zone*** where values are uncertain because the flow might be either laminar or turbulent; a ***transition zone,*** where f is a function of both Reynolds number and relative pipe roughness; and a ***zone of complete turbulence*** (fully-rough-pipe flow), where the value of f is independent of Reynolds number and depends solely upon the relative roughness, e/D.

There is no sharp line of demarcation between the transition zone and the zone of complete turbulence. The dashed line of Fig. 8.11 that separates the two zones was suggested by R. J. S. Pigott; the equation of this line is $\mathbf{R} = 3500/(e/D)$. On the right-hand side of the chart the given values of e/D correspond to the curves and *not* to the grid. Note how their spacing varies. The lowest of the curves in the transition zone is the smooth-pipe ($e = 0$) curve given by Eqs. (8.47) and (8.48); notice how many of the other curves blend asymptotically into the smooth-pipe curve.

EXERCISES

8.13.1 Oil ($s = 0.90$) with viscosity 1.6×10^{-4} lb·sec/ft^2 flows in a 4-in-diameter welded-steel pipe (see Table 8.1) at 0.25 cfs. What is the friction head loss per foot of pipe?

8.13.2 Oil ($s = 0.94$) with viscosity 0.0096 N·s/m^2 flows in a 90-mm-diameter welded-steel pipe (see Table 8.1) at 7.2 L/s. What is the friction head loss per meter of pipe?

8.13.3 Water at 20°C flows through a 150-mm-diameter pipe with $e = 0.015$ mm. (*a*) If the mean velocity is 5 m/s, what is the nominal thickness δ_v of the viscous sublayer? (*b*) What will δ_v be if the velocity is increased to 6.2 m/s?

8.13.4 A straight, new 48-in-diameter asphalted cast-iron pipe (see Table 8.1) 700 ft long carries 78°F water at an average velocity of 12 fps. (*a*) Using the value of f as determined from Fig. 8.11, find the shear force on the pipe. (*b*) What will be the shear force if the average velocity is reduced to 4.2 fps?

[14] L. F. Moody, Friction Factors for Pipe Flows, *ASME Trans.,* Vol. 66, 1944. Lewis F. Moody (1880–1953), an eminent American engineer and professor, also contributed greatly to our understanding of similitude and cavitation as applied to hydraulic machinery (Secs. 15.4, 15.10, 16.10, and 16.12).

[15] Fig. 8.11 is also called a ***Stanton diagram,*** because Stanton first proposed such a plot.

8.14 SINGLE-PIPE FLOW: SOLUTION BASICS

The methods of solution we will use for single-pipe flow are extensions of the method summarized in Sec. 5.14. We recommend that you review that section now, together with Secs. 5.11 and 5.13, which discuss the energy line and the hydraulic grade line.

Governing Equations

Four simultaneous equations govern flow at a point in a single pipe. Three of these are the equations of continuity, energy loss, and the Reynolds number:

$$V = \frac{4Q}{\pi D^2} \quad (4.7); \qquad h_f = f\frac{L}{D}\frac{V^2}{2g} \quad (8.13); \qquad \mathbf{R} = \frac{DV}{\nu} \quad (8.1)$$

If the flow is turbulent, as is most common, the fourth is the Colebrook Eq. (8.51) or the Haaland Eq. (8.52) for the friction factor, as just presented in Sec. 8.12; if the flow is laminar, which is more rare, the friction factor is instead given by $f = 64/\mathbf{R}$, Eq. (8.29). The four unknowns are usually f, $\mathbf{R}$, D or h_f, and Q or V. If the pipeline is uniform in size, shape, and roughness, then these equations will yield the same results at all points along it.

In some cases we will need h_f as a known quantity, and if it is not given we can often obtain it by rearranging energy Eq. (5.28) to first find the total head loss h_L as follows:

$$h_L = H_1 - H_2 = \left(\frac{p}{\gamma} + z + \frac{V^2}{2g}\right)_1 - \left(\frac{p}{\gamma} + z + \frac{V^2}{2g}\right)_2$$
$$= \frac{\Delta p}{\gamma} + \Delta z + \frac{V_1^2 - V_2^2}{2g}$$

If, for example, points 1 and 2 are in contact with the atmosphere, $p_1 = p_2 = \Delta p = 0$; if they are also on reservoir water surfaces, $V_1 = V_2 = 0$, so that $h_L = \Delta z$. If they are in a pipeline of constant diameter, $V_1 = V_2$. As we noted in Sec. 5.3, the pipe friction head loss h_f must be either equal to or less than h_L; the difference results from what are known as minor losses, which are often negligible, and which we will discuss more fully in Secs. 8.20–8.26.

Solution of Special Cases

Laminar Flow. For laminar flow (see Fig. 8.11) we know from Eq. (8.29) that $f = 64/\mathbf{R}$. Substituting this into pipe-friction Eq. (8.13) to eliminate f, and using Eq. (8.1) for $\mathbf{R}$, yields Eq. (8.28). This equation is explicit (Appendix B) in each variable of interest, h_f, D, and V, so we can solve directly for any one of them. We must remember to check the validity of the solution we obtain by confirming that $\mathbf{R}$ is in the laminar range (normally < 2000, see Sec. 8.2).

Smooth-Pipe Flow. When e is so small that the roughness elements do not extend through the viscous sublayer or zone (i.e., $e < \delta_v$, see Sec. 8.10), so that the pipe behaves as a smooth pipe (Fig. 8.9*a*) even though most of the flow is turbulent, this is represented by the lowest curve on Fig. 8.11, for which $e = 0$.

From Sec. 8.12 we know we may express f along this curve by the Blasius equation (8.48) provided $3000 \leq \mathbf{R} \leq 10^8$. Combining Eq. (8.48) with pipe-friction Eq. (8.13) to eliminate f, and using Eq. (8.1) for $\mathbf{R}$, yields

Smooth pipe, $3000 \leq \mathbf{R} \leq 10^5$:

$$h_f = 0.1580\nu^{0.25}\frac{LV^{1.75}}{gD^{1.25}} \tag{8.53}$$

This equation has the advantage of being explicit in each variable of interest, h_f, D, or V. We must remember to check the validity of the solution it yields by confirming that $\mathbf{R}$ is in the required Blasius range.

Fully-Rough-Pipe Flow. When e is so large that the roughness elements extend through both the viscous zone and the transition zone (i.e., $e > 14\delta_v$; see Fig. 8.8), so that there is complete turbulence and the flow is fully rough (Secs. 8.10 and 8.13), this is represented by the horizontal lines at the right of Fig. 8.11, where $\mathbf{R} > 3500/(e/D)$. Then $f = f_{\min}$ = constant is given by Eq. (8.50), which when rearranged gives

Fully rough pipe:

$$f = f_{\min} = \left[2\log\left(\frac{3.7}{e/D}\right)\right]^{-2} \tag{8.54}$$

Substituting this into pipe-friction Eq. (8.13) to eliminate f, yields

Fully rough pipe ($f_{\min}$):

$$h_f = \frac{1}{\left[2\log\left(\frac{3.7}{e/D}\right)\right]^2}\frac{L}{D}\frac{V^2}{2g} \tag{8.55}$$

We notice that for fully-rough-pipe flow h_f is directly proportional to V^2 and is independent of $\mathbf{R}$. Equation (8.55) is explicit in h_f and V, but implicit in D (see Appendix B). The solution of implicit equations is discussed in Secs. 8.15–8.17 and 8.27, and in Appendixes C and D. Values of e are given in Table 8.1. We must check the validity of a solution we obtain from this equation by confirming that $\mathbf{R}$ is sufficiently large.

TABLE 8.1 **Values of absolute roughness e for new commercial pipes**

	Feet	Millimeters
Glass, plastic (smooth)	0.0	0.0
Drawn tubing, brass, lead, copper, centrifugally spun cement, bituminous lining, transite	0.000 005	0.0015
Commercial steel, wrought iron, welded-steel pipe	0.000 15	0.046
Asphalt-dipped cast iron	0.000 4	0.12
Galvanized iron	0.000 5	0.15
Cast iron, average	0.000 85	0.25
Wood stave	0.0006–0.003	0.18–0.9
Concrete	0.001–0.01	0.3–3
Riveted steel	0.003–0.03	0.9–9

Note: $\frac{e}{D} = \frac{e \text{ in feet}}{D \text{ in feet}} = 12 \times \frac{e \text{ in feet}}{D \text{ in inches}} = \frac{e \text{ in mm}}{D \text{ in mm}}$

In engineering applications, flows in any of these three special cases occur relatively rarely. In Fig. 8.11 we see that the only flow region not covered by these three cases is the transition zone. In addition to the fact that most engineering flows occur in the transition zone, flow solutions for this zone are more involved. We will discuss various ways to solve such flows, and flows in which the regimen (zone) is initially unknown, in Secs. 8.15–8.19 and 8.27.

8.15 SINGLE-PIPE FLOW: SOLUTION BY TRIALS

As we have already noted in Secs. 8.12–8.14, often some of the equations governing pipe flow are implicit in form (Appendix B), and therefore they do not readily permit direct solution. For example, although Haaland's equation (8.52) is explicit in f, it is still implicit in V and D, since these are involved in both $\mathbf{R}$ and in f through Eq. (8.13). The classical way to overcome this difficulty has been to use trial-and-error procedures in combination with the Moody chart described in Sec. 8.13. With the more recent availability of the Haaland equation, we can now also perform these procedures using equations alone. We will explore both methods in this section, including applications to different types of problems.

To solve any problem (other than laminar flow or smooth-pipe flow) using the friction factor f, we will require a value of e, and we can obtain this from Table 8.1. As the ratio of e/D is dimensionless, we may use any units, provided they are the same for both e and D. Note also that we should use exact values of the internal pipe diameter. Exact diameters often differ from the nominal sizes, particularly for smaller pipes. If the pipe is not of circular cross section, we can replace D in all instances by $4R_h$ as we noted in Sec. 8.6.

With regard to the values of e, note that those given in Table 8.1 are for new, clean pipes, and even in such cases there may be considerable variation in the values. Consequently, in practical cases, the value of f may be in error by $\pm 5\%$ for smooth pipes and by $\pm 10\%$ for rough ones. For old pipes, values of e may be much higher, but there is much variation in the degree with which pipe roughness increases with age, since so much depends on the nature of the fluid transported. In addition, deposits in small pipes materially reduce the internal diameter. The effect of the roughness of pipe joints may also increase the value of f substantially. So we must use judgment in estimating a value of e, and consequently of f.

Most engineering problems fall within the transition zone (see Fig. 8.11). When solving single-pipe flow problems in this zone, where conditions depend on both e/D and the Reynolds number, we need a definite value of $\mathbf{R}$. But when V or D is unknown, so is $\mathbf{R}$. The solution procedure varies with the type of problem, and we can categorize most single-pipe flow problems into one of the following three types:

Type	Find	Given
1. Head-loss problem	h_f	D, Q or V, and g, L, e, ν
2. Discharge problem	Q or V	D, h_f, and g, L, e, ν
3. Sizing problem	D	Q, h_f, and g, L, e, ν

Here g is the gravitational acceleration, L is the pipe length, Q is the flow rate, and ν is the kinematic viscosity. Of course, we may know μ and ρ instead of ν, in which case we may obtain ν from Eq. (2.11).

Type 1 problems are the simplest. If Q is given, we can easily obtain V from continuity Eq. (4.7). Then we can easily obtain the Reynolds number and e/D from the given data. Next we enter Fig. 8.11 vertically with $\mathbf{R}$ and along a curve (possibly interpolated) for e/D from the right, to identify an operating point for which we can read off the friction factor f horizontally to the left. Alternatively, we can compute f from Eq. (8.52) or by trial and error from Colebrook Eq. (8.51). With this value of f, we can directly compute the friction head loss h_f from Eq. (8.13).

In ***Type 2*** and ***Type 3*** problems, because either V or D is unknown, the Reynolds number $\mathbf{R}$ is not known at the outset and so a direct solution is not possible. However, we notice in Fig. 8.11 that the value of f changes very slowly with large changes in $\mathbf{R}$. So we can usually solve the problem quite effectively by assuming an initial value of f, and then obtaining the final solution by successive trials (trial and error; see Sample Prob. 3.5). If D is known (Type 2), the fully-rough-pipe f value (f_{min}) given on the right-hand side of Fig. 8.11 or by Eq. (8.54) provides a good starting point. If D is not known (Type 3), a value near the middle of the f range on Fig. 8.11, such as 0.03, makes a good start. Each succeeding trial is started with the f value obtained from the previous trial. The value of f can be considered close enough when the first three significant figures of the required answer (Q, V, or D) no longer change. Note that if we assume values of some other variable besides f, convergence is usually much slower.

For ***Type 2*** problems (to find V or Q), we rearrange Eq. (8.13) into the form $V = K/\sqrt{f}$, where $K = \sqrt{2gDh_f/L}$ is known. Assuming an f (as just discussed) therefore yields a V, which enables us to calculate $\mathbf{R}$ and enter the diagram or to use Eq. (8.51) or (8.52) to obtain an improved value of f. If this is different from the assumed f, we must repeat the procedure assuming the just-obtained value, and successively repeat it until the two values converge. This usually only requires two or three trials, by which time all the values are correct, including the required value of V.

For ***Type 3*** problems, since the D to be found is unknown, neither e/D nor $\mathbf{R}$ are known initially. We substitute $V = 4Q/(\pi D^2)$ into Eq. (8.13) and rearrange it to obtain $D = (fK)^{1/5}$, where $K = 8LQ^2/(\pi^2 g h_f)$ is known. Then assuming an f (as previously discussed) yields a D, which enables us to proceed with repetition in a manner similar to that used for Type 2.

Experience has shown that new users of the Moody diagram frequently misread it. This most probably occurs because none of the scales are linear, and because the intervals between grid lines and chart curves keep changing. So take special care in reading the chart, and confirm any interpolated values by comparing them with nearby grid values or curves in both directions.[16]

[16] Charts involving the same functional relations have been plotted with different coordinates from those in Fig. 8.11 and may be more convenient for certain specific purposes, but we believe that the form shown is best both for purposes of instruction and for general use.

Note that in all cases we should assume an f value whenever possible, rather than some other variable. This is because f varies the least, so trial calculations are less sensitive to its changes, and as a result they converge faster.

Sample Probs. 8.4–8.6 illustrate solutions to Type 1, 2, and 3 problems; the last two of these use trial and error.

SAMPLE PROBLEM 8.4 A 20-in-diameter galvanized iron pipe 2 miles long carries 4 cfs of water at 60°F. Find the friction head loss: (*a*) using Fig. 8.11 and the Reynolds number; (*b*) using Fig. 8.11 and its $D''V$ scale; (*c*) using only a basic scientific calculator,[17] without Fig. 8.11.

Solution
This is a Type 1 problem, to find h_f. From Table 8.1 for galvanized iron:

$$e = 0.0005 \text{ ft}: \quad \text{so} \quad e/D = 0.0005(12)/20 = 0.0003$$

$$L = 2 \text{ mi}(5280 \text{ ft/mi}) = 10{,}560 \text{ ft}$$

Eq. (4.7):
$$V = \frac{4Q}{\pi D^2} = \frac{4(4)}{\pi(20/12)^2} = 1.833 \text{ fps}$$

Table A.1 for water at 60°F: $\nu = 1.217 \times 10^{-5}$ ft^2/sec

$$\mathbf{R} = \frac{DV}{\nu} = \frac{(20/12)1.833}{1.217 \times 10^{-5}} = 2.51 \times 10^5 \ (> \mathbf{R}_{\text{crit}}, \text{ i.e., flow is turbulent})$$

(*a*) Enter Fig. 8.11 at the right-hand side with $e/D = 0.0003$, by interpolating between 0.0002 and 0.0004; note that the e/D spacing varies. Follow this (unplotted) e/D curve to the left until it crosses a vertical line at $\mathbf{R} = 2.51 \times 10^5$ (*caution:* this is between 10^5 and 10^6). For this operating point, reading horizontally to the left, $f = 0.0172$.

Eq. (8.13):
$$h_f = f\frac{L}{D}\frac{V^2}{2g} = 0.0172\frac{(10{,}560)}{(20/12)}\frac{1.833^2}{2(32.2)} = 5.69 \text{ ft} \qquad \textbf{\textit{ANS}}$$

(*b*) $D''V = 20(1.833) = 36.7$. Find this value on the scale across the top of Fig. 8.11; note that this scale is varying. Find where the (interpolated) curve for $e/D = 0.0003$ crosses the vertical line at $D''V = 36.7$. From this point, read horizontally to the left, to find $f = 0.0172$. Compute h_f as for part (*a*).

Note: From the operating point on Fig. 8.11, we see that flow conditions are in the transition zone of turbulent flow, which is typical.

[17] We define a basic scientific calculator as one that is not programmable and does not have automatic equation solving capabilities.

(*c*) Eq. (8.41): $$\frac{1}{\sqrt{f}} = -1.8\log\left[\left(\frac{0.0003}{3.7}\right)^{1.11} + \frac{6.9}{2.51\times10^{5}}\right] = 7.65$$

from which $f = 0.01709$

Eq. (8.10): $$h_f = 0.01709\frac{10{,}560}{(20/12)}\frac{(1.883)^2}{2(32.2)} = 5.65\text{ ft} \quad \textbf{\textit{ANS}}$$

Note: This differs from answer (*a*) by only 0.04 ft, or 0.70%.

SAMPLE PROBLEM 8.5 Water at 20°C flows in a 500-mm-diameter welded-steel pipe. If the friction loss gradient is 0.006, determine the flow rate: (*a*) using Fig. 8.11; (*b*) using only a basic scientific calculator,[17] without Fig. 8.11.

Solution
This is a Type 2 problem, to find Q.

Table 8.1 for welded steel: $e = 0.046$ mm; $e/D = 0.046/500 = 0.000092$

Table A.1 at 20°C: $\nu = 1.003\times10^{-6}$ m²/s; $h_f/L = 0.006$ is given

Eq. (8.14): $$S = \frac{h_f}{L} = \frac{f}{D}\frac{V^2}{2g}, \quad \text{i.e.,} \quad 0.006 = \frac{fV^2}{0.5(2)9.81}$$

from which $V = 0.243/f^{1/2}$.

(*a*) Fig. 8.11 for $e/D = 0.000092$: $f_{\min} \approx 0.0118$.
Try $f = 0.0118$. Then $V = 0.243/(0.0118)^{1/2} = 2.23$ m/s.

Eq. (8.1): $$\mathbf{R} = \frac{DV}{\nu} = \frac{0.5(2.23)}{1.003\times10^{-6}} = 1.114\times10^{6} \quad \text{(turbulent flow)}$$

Figure 8.11 with $e/D = 0.000092$ and $\mathbf{R} = 1.114\times10^6$: $f = 0.0131$. Assumed and obtained f values are different, so we must try again; use the obtained f for the next trial. Tabulating all the trials:

Try f	V, m/s	**R**	Obtained f	
0.0117	2.25	1.120×10^6	0.0131	Try again
0.0131	2.12	1.059×10^6	0.0131	Converged!

The f values now agree, so we have the true operating point. Convergence is rapid!

$$Q = AV = (\pi/4)D^2V = (\pi/4)(0.5)^2 2.12 = 0.416\text{ m}^3/\text{s} \quad \textbf{\textit{ANS}}$$

Caution: Take great care to read Fig. 8.11 correctly.

(*b*) Eq. (8.54) for $e/D = 0.000092$: $f_{\min} = 0.01179$. Calculate V and **R** as in (*a*), then obtain an improved f from Eq. (8.52). Use the obtained f for the next trial. Tabulating all the trials:

Try f	V, m/s	**R**	Obtained f	
0.01179	2.23	1.114×10^6	0.01309	Try again
0.01309	2.12	1.057×10^6	0.01315	Converged!

$$Q = AV = (\pi/4)(0.5)^2 2.12 = 0.416\text{ m}^3/\text{s} \quad \textbf{\textit{ANS}}$$

SAMPLE PROBLEM 8.6 A galvanized iron pipe 18,000 ft long must convey ethyl alcohol ($\nu = 2.3\times10^{-5}$ ft²/sec) at a rate of 135 gpm. If the friction head loss must be 215 ft, determine the pipe size theoretically required: (*a*) using Fig. 8.11; (*b*) using only a basic scientific calculator, without Fig. 8.11.

Solution
This is a Type 3 problem, to find D.

Inside cover: $Q = 135$ gpm(2.23 cfs/1000 gpm) $= 0.301$ cfs

Table 8.1 for galvanized iron: $e = 0.0005$ ft.

$$\frac{e}{D} = \frac{0.0005}{D}$$

Eq. (4.7): $$V = \frac{4Q}{\pi D^2} = \frac{4(0.301)}{\pi D^2} = \frac{0.383}{D^2}$$

Eq. (8.1): $$\mathbf{R} = \frac{DV}{\nu} = \frac{D}{2.3\times10^{-5}}\left(\frac{0.383}{D^2}\right) = \frac{16{,}590}{D}$$

Eq. (8.13): $$215 = h_f = f\frac{18{,}000}{2D(32.2)}\left(\frac{0.383}{D^2}\right)^2, \text{ from which } D = 0.718f^{1/5}$$

(*a*) Start by assuming a mid-range value of f

Try f	D, ft	e/D	**R**	Chart f	
0.0300	0.356	0.001404	4.68×10^4	0.0253	Try again
0.0253	0.344	0.001453	4.84×10^4	0.0253	Converged!

Values of f now agree, so we have the true operating point. Convergence is rapid!

$$D = 0.344\text{ ft} = 4.13\text{ in} \quad \textbf{\textit{ANS}}$$

(*b*) Start by assuming a mid-range value of f. Calculate V and $\mathbf{R}$ as before, then obtain an improved f from Eq. (8.52). Tabulating this and subsequent trials:

Try f	D, ft	e/D	$\mathbf{R}$	Eq. (8.41) f	
0.0300	0.356	0.001404	46,800	0.02501	Try again
0.0250	0.343	0.001456	48,500	0.02504	Converged!

$$D = 0.343 \text{ ft} = 4.12 \text{ in} \qquad \textit{ANS}$$

As noted earlier, in practice there can be a considerable amount of uncertainty in the size of the absolute roughness e. So it is important to have some idea what effects changes in e will have on h_f, Q, and D. From Eqs. (8.13) and (4.7), it follows that $h_f \propto f$, $Q \propto f^{-1/2}$, and $D \propto f^{1/5}$. The variation of f with e we may see in Fig. 8.11 or Eq. (8.51). As an example, in Sample Prob. 8.4, if e had been 20% larger then the h_f would have been 2.3% larger. This change in the head loss would be larger for larger f (or larger e/D), and vice versa. Changes in Q and D will be smaller than those in h_f.

We can avoid the repetitious labor required for the trial-and-error procedures just described by using direct methods and automated methods. We will review these alternative solution methods in Secs. 8.16 and 8.17.

EXERCISES

8.15.1 Compute the friction head per 100 ft of 3-in-diameter pipe for a Reynolds number of 50,000 if (*a*) the flow is laminar (achievable with great care); (*b*) the flow is turbulent in a smooth pipe; (*c*) the flow is turbulent in a rough pipe with $e/D = 0.05$. Consider two situations: one where the fluid is 70°F water, the other where the fluid is SAE 10 (Western lubricating) oil at 150°F.

8.15.2 California crude oil, warmed until its kinematic viscosity is 0.0004 ft^2/sec and its specific weight is 53.45 lb/ft^3, is pumped through a 3-in pipe ($e = 0.001$ in). (*a*) For laminar flow with $\mathbf{R} = \mathbf{R}_{crit} = 2000$, what would be the loss in energy head in psi per 1000 ft of pipe? (*b*) What would be the loss in head per 1000 ft if the velocity were three times the value in (*a*)?

8.15.3 Water at 50°F flowing through 80 ft of 4-in-diameter average cast-iron pipe causes a friction head loss of 0.27 ft. Find the flow rate.

8.15.4 When gasoline with a kinematic viscosity of 5×10^{-7} m^2/s flows in a 200-mm-diameter smooth pipe, the friction head loss is 0.43 m per 100 m. Find the flow rate.

8.16 SINGLE-PIPE FLOW: DIRECT SOLUTIONS[18]

With time and experience, engineers working on pipe-flow problems have developed some ingeneous ways to obtain direct solutions. These have the advantage of avoiding the tedious trial and error described in Sec. 8.15, and in addition they are more precise and reliable than reading off a chart that may easily be misread.

For people who do not have access to advanced programmable calculators or computers with mathematics software (Sec. 8.17), ways have been found to solve each of the three types of pipe-flow problems mentioned in Sec. 8.15 without iteration, so that we can evaluate them directly on a basic scientific calculator.[19] This of course requires an *explicit* equation in the sought quantity (Appendix B). We describe these methods further below.

The Head-Loss Problem (Type 1). The Haaland equation (8.52) makes possible a direct computation of f, from which we can find h_f using Eq. (8.13). Section 8.15 and Sample Prob. 8.4 explain this more fully. For a faster solution we may eliminate f between Eqs. (8.13) and (8.52).

The Discharge Problem (Type 2). We can obtain a single equation for the velocity (or discharge) as follows. First we rearrange Eq. (8.13) to obtain

$$\frac{1}{\sqrt{f}} = V\sqrt{\frac{L}{2gDh_f}}$$

Then, when we substitute this and $\mathbf{R} = DV/\nu$ into the Colebrook equation (8.51) and rearrange, we get

$$\text{Turbulent flow, Type 2:} \quad V = -2\sqrt{\frac{2gDh_f}{L}}\log\left(\frac{e/D}{3.7} + \frac{2.51\nu}{D}\sqrt{\frac{L}{2gDh_f}}\right) \tag{8.56a}$$

This equation is explicit in V, which fortunately results because the Vs from $\mathbf{R}$ and $\sqrt{f}$ cancel in the last term of Eq. (8.51) [the same substitution into the Haaland equation (8.52) does not produce such a desirable result]. Having this explicit equation available will save much repetitive work in solving more involved problems, like those of branching pipes (Sec. 8.26). If Q is required rather than V, we can use continuity equation (4.7) to eliminate V, and Eq. (8.56*a*) becomes

$$\text{Turbulent flow, Type 2:} \quad \frac{4Q}{\pi D^2} = -2\sqrt{\frac{2gDh_f}{L}}\log\left(\frac{e/D}{3.7} + \frac{2.51\nu}{D}\sqrt{\frac{L}{2gDh_f}}\right) \tag{8.56b}$$

[18] Some users may omit this section because it is not required to understand the governing equations and their relationships. Note, however, that the methods described here are of great benefit in Secs. 8.24 (single pipe with minor losses) and 8.26 (branching pipes).

[19] We define a basic scientific calculator to be one that is not programmable and does not have automatic equation solving capabilities.

Using Eq. (8.56), we can calculate Q or V directly with a basic scientific calculator. Remember to use Eq. (8.1) to confirm that $\mathbf{R}$ is in the turbulent range. If $\mathbf{R}$ turns out to be in the laminar range, i.e., less than 2000, we must instead find V or Q from Eq. (8.28) rearranged.

The Sizing Problem (Type 3). We can eliminate iteration by reformulating our equations as follows. Substituting from Eqs. (8.13), (4.7), and (8.1) into the dimensionless quantity

$$\mathbf{N}_1 = f\mathbf{R}^5 = \frac{h_f D 2g}{L}\left(\frac{\pi D^2}{4Q}\right)^2\left(\frac{D}{\nu}\frac{4Q}{\pi D^2}\right)^5 = \frac{128g}{\pi^3}\frac{h_f}{L}\frac{Q^3}{\nu^5} \tag{8.57}$$

indicates that it is independent of the unknown D, so that $\mathbf{N}_1$ is a known quantity. We can also transform the relative roughness into another known dimensionless quantity by dividing it by $\mathbf{R}$, i.e.,

$$\mathbf{N}_2 = \frac{e/D}{\mathbf{R}} = \frac{e}{D}\left(\frac{\nu}{DV}\right) = \frac{e}{D}\left(\frac{\nu}{D}\right)\frac{\pi D^2}{4Q} = \frac{\pi e \nu}{4Q} \tag{8.58}$$

Now we can use $\mathbf{N}_1$ to eliminate f from laminar flow Eq. (8.29) to obtain

Laminar flow, Type 3:
$$\mathbf{R} = \left(\frac{\mathbf{N}_1}{64}\right)^{0.25} \tag{8.59}$$

and we can use $\mathbf{N}_1$ and $\mathbf{N}_2$ to eliminate f and e/D from the Colebrook equation (8.51) for turbulent flow to obtain

$$\mathbf{R}^{2.5} = -2\mathbf{N}_1^{0.5}\log\left(\frac{\mathbf{N}_2\mathbf{R}}{3.7} + \frac{2.51}{\mathbf{N}_1^{0.5}}\mathbf{R}^{1.5}\right) \tag{8.60}$$

Here $\mathbf{R}$ occurs in three places, so this equation is strongly implicit (Appendix B), and Eq. (8.60) appears to be not a very useful result. However, if we plot $\mathbf{N}_1$ versus $\mathbf{R}$ we find it collapses the various flow curves on the Moody chart into a very narrow band that is closely approximated by the formula $\mathbf{R} \approx 1.43\mathbf{N}_1^{0.208}$. We can substitute this into the right-hand side of Eq. (8.60) to create the more useful equation:

Turbulent flow, Type 3:
$$\mathbf{R}^{2.5} = -2\mathbf{N}_1^{0.5}\log\left(\frac{\mathbf{N}_2\mathbf{N}_1^{0.208}}{2.59} + \frac{4.29}{\mathbf{N}_1^{0.188}}\right) \tag{8.61}$$

We see that this reformulation has converted the Colebrook equation into an explicit form.

Comparison of the two Reynolds numbers from Eqs. (8.59) and (8.61) with $\mathbf{R}_{crit} = 2000$ indicates which equation (laminar or turbulent) is applicable for a given Type 3 problem.

Finally, from Eq. (8.1) combined with Eq. (4.7), we have

$$\mathbf{R} = \frac{DV}{\nu} = \frac{D}{\nu}\left(\frac{4Q}{\pi D^2}\right) = \frac{4Q}{\pi \nu D}$$

and so, by rearrangement, we can obtain the required diameter D from

Type 3:
$$D = \frac{4Q}{\pi \nu \mathbf{R}} \tag{8.62}$$

SAMPLE PROBLEM 8.7 Solve Sample Prob. 8.5 without trial and error, using only a basic scientific calculator.

Solution

This is a Type 2 problem, to find Q. As in Sample Prob. 8.5, $D = 0.5$ m, $h_f/L = 0.006$, $e/D = 0.000092$, and $\nu = 1.003 \times 10^{-6}$ m^2/s. The quantity

$$\sqrt{\frac{2gDh_f}{L}} = \sqrt{2(9.81)0.5(0.006)} = 0.243 \text{ m/s},$$

so in turbulent-flow Eq. (8.56*a*):

$$V = -2(0.243)\log\left[\frac{0.000092}{3.7} + \frac{2.51(1.003\times10^{-6})}{0.5}\frac{1}{0.243}\right] = 2.11 \text{ m/s}$$

Check: $\mathbf{R} = \dfrac{DV}{\nu} = \dfrac{0.5(2.11)}{1.003\times10^{-6}} = 1.050\times10^6$. This is $> \mathbf{R}_{crit} = 2000$

so flow *is* turbulent, validating the use of Eq. (8.56*a*). Finally,

$$Q = AV = (\pi/4)D^2V = 0.25\pi(0.5)^2 2.11 = 0.414 \text{ m}^3\text{/s} \qquad \textit{ANS}$$

SAMPLE PROBLEM 8.8 Solve Sample Prob. 8.6 without trial and error using only a basic scientific calculator.

Solution

This is a Type 3 problem, to find D. As in Sample Prob. 8.6, $h_f = 215$ ft, $L = 18{,}000$ ft, $Q = 0.301$ cfs, $e = 0.0005$ ft, $g = 32.2$ ft/sec^2, and $\nu = 2.3\times10^{-5}$ ft^2/sec.

Eq. (8.57):
$$\mathbf{N}_1 = \frac{128(32.2)}{\pi^3}\left(\frac{215}{18{,}000}\right)\frac{(0.301)^3}{(2.3\times10^{-5})^5} = 6.71\times10^{21}$$

Eq. (8.58):
$$\mathbf{N}_2 = \frac{\pi e\nu}{4Q} = \frac{\pi(0.0005)2.3\times10^{-5}}{4(0.301)} = 3.00\times10^{-8}$$

Laminar flow Eq. (8.59):
$$\mathbf{R} = \left(\frac{\mathbf{N}_1}{64}\right)^{0.25} = 101{,}300 > \mathbf{R}_{crit} = 2000$$

Turbulent-flow Eq. (8.61):

$$\mathbf{R}^{2.5} = -2(6.73\times10^{21})^{0.5}\log\left[\frac{3.00\times10^{-8}(6.71\times10^{21})^{0.208}}{2.59} + \frac{4.29}{(6.71\times10^{21})^{0.188}}\right]$$

$$= 5.13\times10^{11}$$

▦: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

from which, $\mathbf{R} = 48{,}300$. Clearly the laminar flow equation is invalid, flow is turbulent, and $\mathbf{R} = 48{,}300$. Thus, from Eq. (8.62),

$$D = \frac{4Q}{\pi\nu\mathbf{R}} = \frac{4(0.301)}{\pi(2.3\times10^{-5})48{,}300} = 0.345 \text{ ft} \qquad \textit{ANS}$$

Note: This differs by only 0.00055 ft or 0.16% from the accurate answers in Sample Prob. 8.9.

EXERCISES

8.16.1 Solve Exer. 8.15.3 without trial and error, using only a basic scientific calculator.

8.16.2 Solve Exer. 8.15.4 without trial and error, using only a basic scientific calculator.

8.17 SINGLE-PIPE FLOW: AUTOMATED SOLUTIONS[20]

With progress in technology and solution techniques, automated methods have become available for solving pipe-flow problems that involve the complicated equations for roughness described in Sec. 8.12. These methods provide accurate and convenient alternatives to the manual methods described in Secs. 8.15 and 8.16.

In recent years, preprogrammed equation solvers have become available on certain more advanced programmable scientific calculators and in some spreadsheet and mathematics software packages (see Appendixes C and D). To use them, we must first enter the equation, explicit or implicit (Appendix B) and in any form, into the calculator's memory or into a computer file. Such calculators now often have "equation writers," which present the equation on the screen in the same form as we write it on paper. Some of the equation solving software packages also present the equation in the same form. In either case we can save the equations, and edit them if necessary. Then we assign values to each of the known variables, and assign estimated values to the unknown variables.

When the calculator or software solves for the unknown variable(s), it uses an internally programmed trial-and-error procedure to find them, to high precision. Of great practical convenience is the fact that, once everything is stored in memory, it is very easy to change which of the variables we wish to treat as the unknowns. An important difference between the advanced programmable

[20] Some users may omit this section because it is not required to understand the governing equations and their relationships. Note, however, that in Sec. 8.24 (single pipe with minor losses) and Secs. 8.26–8.29 (branching pipes, series and parallel pipes, and pipe networks) we discuss advantages of the methods described here.

calculators and the equation solving software is that at present (2001) the calculators can solve only one nonlinear (and implicit) equation for one unknown (Appendixes B and C.1), whereas a number of software packages like Mathcad and Excel can automatically solve a system of many nonlinear equations in an equal number of unknowns.

Next we will discuss ways to use automated equation solvers with each of the three principal types of single-pipe flow problem.

The Head-Loss Problem (Type 1). While equation solvers are not necessary for Type 1 problems, we can still use them, as described next for Types 2 and 3. They *are* necessary if we want to use the implicit Colebrook equation (8.51).

The Discharge Problem (Type 2). An equation solver on a programmable calculator is not *necessary* for Type 2 problems, because we can solve them directly with a basic scientific calculator, as described in Sec. 8.16. However, we can use a programmable calculator for convenience, in the same way as for Type 3 problems.

When using equation solving software such as Mathcad or Excel to solve the discharge problem, we enter the four governing equations in their familiar forms (Sec. 8.14). We must assign values to the known variables g, ν, h_f, L, e, and D, and initial estimates for the unknown variables f (0.03 suggested), V, $\mathbf{R}$ (> 2000), and Q. The solver then simply solves the four equations (some nonlinear, some implicit) for the four unknowns.

The Sizing Problem (Type 3). When using equation solving software, we follow the same procedure as for the discharge problem just described, with the one difference that we interchange Q and D on the lists of known and unknown variables.

With an equation solver on a programmable calculator, we can solve Eq. (8.56*b*) for D directly, even though it is implicit in D. We must then check the value of $\mathbf{R}$.



SAMPLE PROBLEM 8.9 Solve Sample Prob. 8.6 using (*a*) an equation solver on a programmable scientific calculator; (*b*) equation solving software.

Solution

This is a Type 3 problem, to find D. As in Sample Prob. 8.6, $h_f = 215$ ft, $L = 18{,}000$ ft, $Q = 0.301$ cfs, $e = 0.0005$ ft, $g = 32.2$ ft/sec^2, and $\nu = 2.3\times10^{-5}$ ft^2/sec.

(*a*) Using an "equation writer" on the programmable calculator, key in Eq. (8.56*b*) and store it permanently, assigning it a name such as EQ856B. Then activate the equation solving feature (see Appendixes C.1 and D.1), select this equation from the catalog of equations, and make it the "current" equation (note that it eliminates f, $\mathbf{R}$, and V).

Display the menu of variables in the equation, and enter numeric values for them all (g, ν, h_f, L, e, Q, D) as above, including an estimate of D (1 ft, say).

Instruct the solver to solve for D.

Result: $D = 0.344$ ft ***ANS***

Note: Appendix D.1 gives more information about using solvers on programmable scientific calculators.

(*b*) Using equation solving software, such as Excel or Mathcad (Appendixes C.2–C.3), open a file or "worksheet" and type in the four governing equations (Sec. 8.14). These equations involve ten variables, and of these we wish to find f, V, **R**, and D.

Assign numerical values to the six known variables (g, ν, h_f, L, e, Q), as before. Then assign estimated values to the four unknown variables, for example, $f = 0.03$, $V = 5$, $R = 100{,}000$, $D = 0.5$. It is helpful to keep these two sets of assignments on separate lines; they can be labeled with text. The success of the procedure may be sensitive to the estimated values, so if the solver is unable to find a solution, try different estimates.

Results: $f = 0.0253$, $V = 3.23$ ft/s, $\mathbf{R} = 48{,}400$, $D = 0.344$ ft ***ANS***

Notes: We can also automatically solve Type 2 problems in this way. Appendixes D.2 and D.3 include sample problems solved by Excel and Mathcad.

EXERCISES

8.17.1 Solve Sample Prob. 8.5 using an equation solver on (*a*) a programmable scientific calculator; (*b*) computer software.

8.17.2 Solve Exer. 8.15.3 using an equation solver on (*a*) a programmable scientific calculator; (*b*) computer software.

8.17.3 Solve Exer. 8.15.4 using an equation solver on (*a*) a programmable scientific calculator; (*b*) computer software.

8.18 EMPIRICAL EQUATIONS FOR SINGLE-PIPE FLOW

The presentation of friction loss in pipes given in Secs. 8.1–8.17 incorporates the best knowledge available on this subject, as far as application to Newtonian fluids (Sec. 2.11) is concerned. Admittedly, however, the trial-and-error type of solution, especially when encumbered with computations for relative roughness and Reynolds number, becomes tedious when repeated often for similar conditions. It is natural, therefore, that engineers developed simple and convenient-to-use design formulas, based on experiments and observations but limited to specific fluids and conditions. Such equations, in which the relations between the

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

UNIVERSIDAD POLITECNICA SALESIANA BIBLIOTECA

primary variables of interest (here V, R_h or D, and $S = h_f/L$) are based on observations rather than theory, are known as ***empirical equations*** (Appendix B). This is in contrast to the Darcy-Weisbach Eq. (8.13), for which we saw in Secs. 8.4 and 8.5 that these relations were developed from theory, primarily dimensional analysis; the fact that the roughness coefficients were determined experimentally does not affect the way in which the variable relations were developed.

Perhaps the best example of such an empirical equation is that of Hazen and Williams, applicable only to the flow of water in pipes larger than 2 in (50 mm) and at velocities less than 10 fps (3 m/s), but widely used in the waterworks industry. This formula takes the form

$$\text{BG units:} \qquad V = 1.318 C_{HW} R_h^{0.63} S^{0.54} \tag{8.63a}$$

$$\text{SI units:} \qquad V = 0.849 C_{HW} R_h^{0.63} S^{0.54} \tag{8.63b}$$

where R_h (ft or m) is the hydraulic radius (Sec. 8.3), and $S = h_f/L$, the energy gradient. The advantage of Eq. (8.63) over the standard pipe-friction formula is that the roughness coefficient C_{HW} is not a function of the Reynolds number, and so we need not make trial solutions. Values of C_{HW} range from 140 for very smooth, straight pipe down to 110 for new riveted-steel and vitrified pipe and to 90 or 80 for old and tuberculated pipe.

Another empirical formula, which we will discuss in detail in Sec. 10.2, is the Manning formula, which is

$$\text{BG units:} \qquad V = \frac{1.486}{n} R_h^{2/3} S^{1/2} \tag{8.64a}$$

$$\text{SI units:} \qquad V = \frac{1}{n} R_h^{2/3} S^{1/2} \tag{8.64b}$$

where n is a roughness coefficient, varying from 0.008 for the smoothest brass or plastic pipe, to 0.014 for average drainage tile or vitrified sewer pipe, to 0.021–0.030 for corrugated metal, and up to 0.035 for tuberculated cast-iron pipe (Table 10.1). The Manning formula applies to about the same flow range as does the Hazen-Williams formula.

For some problems, it is more convenient to work with Eqs. (8.63) and (8.64) in the form of expressions for head loss (see Sec. 8.19). Because we can also express the equations in terms of V or Q, depending on which is given or sought, and in BG or SI units, the number of alternative forms is quite large. Because each variable occurs only once in the above empirical equations, they are always explicit regardless of how they are rearranged or which variable is unknown. This gives them their distinct advantage, that we can always solve them directly.

Engineers have developed nomographic charts and diagrams for the application of Eqs. (8.63) and (8.64). The lack of accuracy that results from using these formulas is not important in the design of water distribution systems, since we can seldom predict the capacity requirements with high precision, and because flows vary considerably throughout the day.

EXERCISES

8.18.1 When water flows at 2.5 cfs through a 20-in-diameter pipeline, the head loss is 0.0004 ft/ft. Find the value of the Hazen-Williams coefficient.

8.18.2 Water flows at 0.32 m^3/s through a 600-mm-diameter pipeline with a head loss of 0.0029 m/m. Find the value of the Hazen-Williams coefficient.

8.18.3 When water flows at 70 cfs through a 5-ft-diameter pipeline the head loss is 3.0 ft per mile. Find the value of Manning's n.

8.19 NONRIGOROUS HEAD-LOSS EQUATIONS

If we rearrange the empirical Hazen-Williams and Manning's equations of Sec. 8.18 into the form of head-loss equations like Eq. (8.13), they will remain explicit in all unknowns and so easy to solve, but they will of course be less accurate than the rigorous equations that have a friction factor f that depends on the Reynolds number, **R**. We can create a third, similar situation and equation if we use the Darcy-Weisbach Eq. (8.13) with a given, *constant* value of f.

To get good results with these nonrigorous equations the user must select proper values of the friction factor, either C_{HW}, Manning's n, or the constant f. This is more uncertain than estimating the e/D value for the Darcy-Weisbach equation with varying f. In Sec. 8.18 we noted that such less-than-rigorous methods and less accurate results are acceptable for the design of water distribution networks; they could also serve as first estimates for repetitive, rigorous solution procedures.

When the friction factor is constant, i.e., it does not vary with **R**, we find in all three cases that we can conveniently represent the head-loss equations in the form

$$h_f = KQ^n \tag{8.65}$$

where we note that n is a constant exponent and not Manning's n, which we shall here represent by n_m. By rearranging Eq. (8.13) when f is constant and using Eq. (4.7) to replace V, we obtain

Darcy-Weisbach with constant f:
$$K = \frac{8fL}{\pi^2 g D^5}, \qquad n = 2 \tag{8.66}$$

By rearranging the empirical equation (8.63*a*), we obtain

Hazen-Williams BG units:
$$K = \frac{4.727L}{C_{HW}^{1.852} D^{4.87}}, \qquad n = 1.852 \tag{8.67}$$

For SI units we must replace the 4.727 by 10.675 in the Hazen-Williams K; recall from Sec. 8.18 the restrictions on D and V. By rearranging the empirical equation (8.64*a*), we obtain

Manning, BG units:
$$K = \frac{4.66Ln_m^2}{D^{16.3}}, \qquad n = 2 \tag{8.68}$$

For SI units, we must replace the 4.66 by 10.29 in the Manning K; recall the same restrictions on D and V.

We see that $h_f = KQ^2$ except when working with the Hazen-Williams coefficient, and we notice that K is a property of the pipe alone. When we need to solve for discharge, we can of course rearrange Eq. (8.65) into

$$Q = \left(\frac{h_f}{K}\right)^{1/n} \tag{8.69}$$

We shall make use of these nonrigorous equations in Secs. 8.29–8.32.

8.20 MINOR LOSSES IN TURBULENT FLOW

Losses due to *local* disturbances of the flow in conduits such as changes in cross section, projecting gaskets, elbows, valves, and similar items we call ***minor losses.*** In the case of a very long pipe or channel, these losses are usually insignificant in comparison with the loss due to pipe (wall) friction (Secs. 5.3, 8.5) in the length considered. But if the length of pipe or channel is very short, these so-called minor losses may actually be major losses. Thus, in the case of the suction pipe of a pump, the loss of head at entrance, especially if there is a strainer and a foot valve, may be very much greater than the friction loss in the short inlet pipe.

Whenever the average velocity of turbulent flow is altered either in direction or in magnitude, large ***eddies*** (Secs. 4.2, 8.9) or eddy currents are set up causing a loss of energy in excess of the pipe friction in that same length.[21] Head loss in decelerating (i.e., diverging) flow is much larger than that in accelerating (i.e., converging) flow (Sec. 8.24). In addition, head loss generally increases with an increase in the geometric distortion of the flow. Though the causes of minor losses are usually confined to a very short length of the flow path, the effects may not disappear for a considerable distance downstream. Thus an elbow in a pipe may occupy only a small length, but the disturbance in the flow will extend for a long distance downstream.

In the following Secs. 8.21–8.26, we describe the most common sources of minor loss. There are two ways we can represent such losses. We can express them as $kV^2/2g$, where we must determine the ***loss coefficient*** k for each case. Or we can represent them as being equivalent to a certain length of straight pipe, usually expressed in terms of the number of pipe diameters, N. Since

$$k\frac{V^2}{2g} = \frac{f(ND)}{D}\frac{V^2}{2g}$$

it follows that
$$k = Nf$$

To differentiate minor losses from other losses, we shall represent minor head losses by the symbol h' (with a prime). Because they are losses of energy, we will frequently relate them to the energy line (EL) and the hydraulic grade line (HGL), which we initially defined and discussed in Secs. 5.11 and 5.13. Note

[21] In laminar flow these losses are insignificant, because irregularities in the flow boundary create a minimal disturbance to the flow and separation is essentially nonexistent.

that these losses are really only degradations of mechanical energy to a less useful form (heat energy).

8.21 LOSS OF HEAD AT ENTRANCE

Referring to Fig. 8.12, we can see that, as fluid from the reservoir enters the pipe, the streamlines continue to converge for a while, much as though this were a jet issuing from a sharp-edged orifice (Sec. 11.6). As a result, we find a cross section with maximum velocity and minimum pressure at B. This minimum flow area is known as the ***vena contracta.*** At B, surrounding the contracted flowing stream there is fluid in a state of turbulence but having very little forward motion. Between B and C the fluid is very disturbed because the stream expands (is less constrained) and the velocity decreases while the pressure rises. From C to D the flow is normal.

We see that the loss of energy at entrance occurs over the length AC, a distance of several diameters. The increased turbulence and vortex motion in this portion of the pipe cause the friction loss to be much greater than in a corresponding length where the flow is normal, as we can see from the drop of the total-energy line. Of this total loss, a small portion h_f would be due to the normal pipe friction (see Fig. 8.12). Hence the difference between this and the total, or h_e', is the true value of the extra loss caused at entrance.

The loss of head at entrance we can express as

$$h_e' = k_e \frac{V^2}{2g} \tag{8.70}$$

where V is the mean velocity in the pipe, and k_e is the loss coefficient, whose general values are given in Fig. 8.13.

The entrance loss results primarily from the turbulence created by the enlargement of the stream after it passes section B, and this enlargement in turn

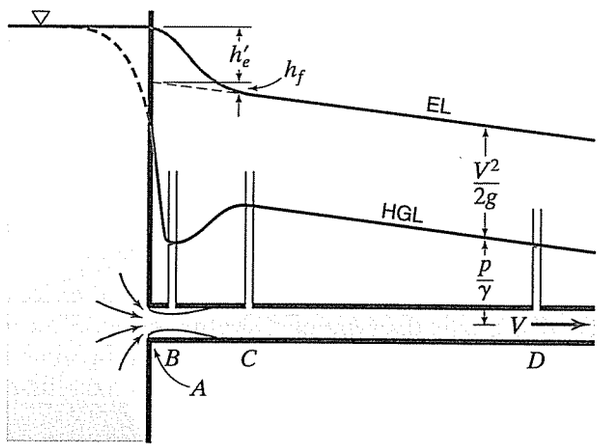


Figure 8.12
Conditions at entrance.

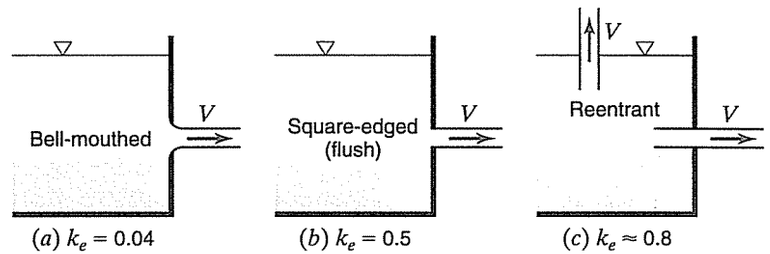


Figure 8.13
Entrance loss coefficients.

depends on how much the stream contracts as it enters the pipe. Thus it very much depends on the conditions at the entrance to the pipe. Experiments have determined values of the entrance loss coefficients. If the entrance to the pipe is well rounded or ***bell-mouthed*** (Fig. 8.13*a*), there is no contraction of the stream entering and the coefficient of loss is correspondingly small. For a ***flush*** or ***square-edged entrance,*** such as shown in Fig. 8.13*b*, k_e has a value of about 0.5. A ***reentrant tube,*** such as that shown in Fig. 8.13*c*, produces a maximum contraction of the entering stream, because the streamlines come from around the outside wall of the pipe, as well as more directly from the fluid in front of the entrance. The degree of the contraction depends upon how far the pipe may project within the reservoir and also upon how thick the pipe walls are, compared with its diameter. With very thick walls, the conditions approach that of a square-edged entrance. For these reasons, the loss coefficients for reentrant tubes vary; for very thin tubes, $k_e \approx 0.8$.

8.22 LOSS OF HEAD AT SUBMERGED DISCHARGE[22]

Discharge into Still Water

When a fluid with a velocity V is discharged from the end of a pipe into a closed tank or reservoir which is so large that the velocity within it is negligible, the entire kinetic energy of the flow is dissipated. Thus the discharge loss is

$$h_d' = \frac{V^2}{2g} \tag{8.71}$$

We can confirm that this is true by writing an energy equation between (a) and (c) in Fig. 8.14. Taking the datum plane through (a) and recognizing that the pressure head of the fluid at (a) is y, its depth below the surface, $H_a = y + 0 + V^2/2g$ and $H_c = 0 + y + 0$. Therefore we obtain

$$h_d' = H_a - H_c = \frac{V^2}{2g}$$

[22] We first discussed this topic in Sec. 5.12.

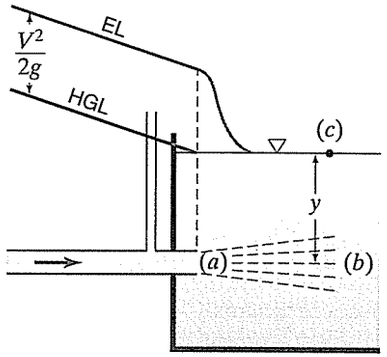


Figure 8.14
Loss due to submerged discharge into still water.

Thus the discharge loss coefficient $k_d = 1.0$ under all conditions; so the only way to reduce the discharge loss is to reduce the magnitude of V by means of a diverging tube. This is the reason for a diverging draft tube that discharges the flow from a reaction turbine (Sec. 16.8).

As contrasted with entrance loss, note that discharge loss occurs *after* the fluid *leaves* the pipe,[23] while entrance loss occurs *after* the fluid *enters* the pipe.

Discharge into Moving Water

Let us now find the head loss when a submerged discharge enters a body of water moving away in a channel, such as from a culvert with a submerged outlet (Sec. 10.23), or as in the tailrace of a reaction turbine (Sec. 16.8).

At the outlet, the streamlines are straight and parallel, so, as demonstrated in Sec. 5.4, the pressure distribution across it must be hydrostatic. After a sufficient distance from the outlet, as at section *cf* in Fig. 8.15, the streamlines will again be essentially straight and parallel, and the flow velocity will be practically uniform ($= V_c$) over the full channel depth. So the pressure distribution at section *cf* must also be hydrostatic, and therefore per Eq. (3.6) (using gage pressures)

$$\frac{p_e}{\gamma} + z_e = \frac{p_f}{\gamma} + z_f = \frac{p_c}{\gamma} + z_c = 0 \tag{8.72}$$

[23] In a short pipe, where the discharge loss may be a major factor, greater accuracy is obtained by using the correction factor α, as explained in Sec. 5.1 (see also Eq. 8.45*a*).

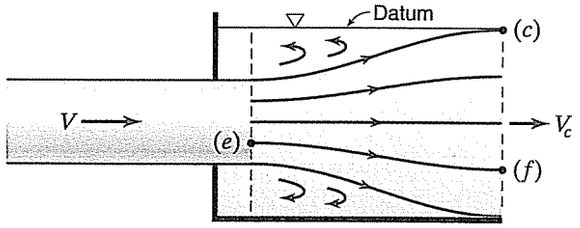


Figure 8.15
Loss due to submerged discharge into moving water.

For a general streamline such as *ef* in Fig. 8.15, from energy Eq. (5.28) we have

$$\left[\frac{p_e}{\gamma} + z_e + \frac{V^2}{2g}\right] - h'_d = \left[\frac{p_f}{\gamma} + z_f + \frac{V_c^2}{2g}\right] \tag{8.73}$$

Substituting from Eq. (8.72) into (8.73) and rearranging we obtain

$$h'_d = \frac{V^2}{2g} - \frac{V_c^2}{2g} \tag{8.74}$$

This equation states that the discharge loss is equal to the *difference* between the discharge and ultimate velocity heads. Equation (8.71) is a special case of this equation.

EXERCISES

8.22.1 A 12-in-diameter pipe ($f = 0.02$) carries fluid at 8 fps between two tanks. The entrance and exit conditions to and from the pipe are square-edged and flush with the wall of the tank. Find the ratio of the minor losses divided by the pipe friction loss if the length of the pipe is (*a*) 4 ft; (*b*) 80 ft; (*c*) 1600 ft.

8.22.2 A 375-mm-diameter pipe ($f = 0.017$) carries fluid at 3.6 m/s between two tanks. The entrance and exit conditions to and from the pipe are reentrant. Find the ratio of the minor losses divided by the pipe friction loss if the length of the pipe is (*a*) 2 m; (*b*) 50 m; (*c*) 1000 m.

8.22.3 Water leaves a turbine at 18.5 fps and enters a tailrace having an average velocity of 1.5 fps. (*a*) What is the submerged discharge head loss? (*b*) By what percentage is this loss reduced if the provision of a draft tube increases the discharge flow area to six times the size?

8.22.4 A smooth 300-mm-diameter pipe is 90 m long and has a flush entrance and a submerged discharge. It carries 15°C water at a velocity of 3 m/s. What is the total head loss?

8.22.5 Oil with a kinematic viscosity of 0.001 ft^2/sec and a specific gravity of 0.92 flows at 8 fps through a smooth 15-in-diameter pipe which is 400 ft long with a flush entrance and submerged discharge. What is the head loss in feet of oil and in psi?

8.23 LOSS DUE TO CONTRACTION

Sudden Contraction

The phenomena accompanying the sudden contraction of a flow are shown in Fig. 8.16. There is a marked drop in pressure due to the increase in velocity and to the loss of energy in turbulence. Note that in the corner upstream at section *C* there is a rise in pressure because the streamlines here are curving, so that the centrifugal action causes a greater pressure at the pipe wall than in the center of the stream. The dashed line indicates the pressure variation along the central streamline from sections *B* to *C*.

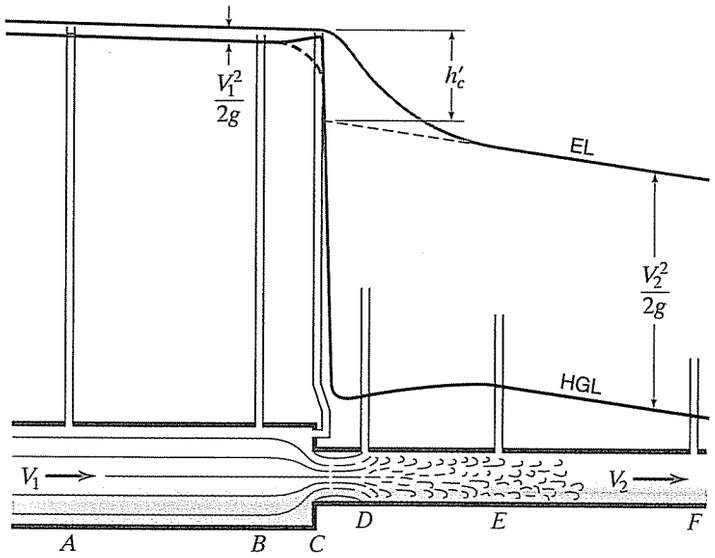


Figure 8.16
Loss due to sudden contraction. (*Plotted to scale.*)

TABLE 8.2 **Loss coefficients for sudden contraction**

D_2/D_1	0.0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1.0
k_c	0.50	0.45	0.42	0.39	0.36	0.33	0.28	0.22	0.15	0.06	0.00

From C to E, the conditions are similar to those described for entrance (Sec. 8.21). We can represent the loss of head for a sudden contraction by

$$h_c' = k_c \frac{V_2^2}{2g} \tag{8.75}$$

where k_c has the values given in Table 8.2.

The entrance loss of Sec. 8.21 is a special case where $D_2/D_1 = 0$.

Gradual Contraction

In order to reduce the foregoing losses, we should avoid abrupt changes of cross section. We could achieve this by changing from one diameter to the other with a smoothly curved transition or with the frustrum of a cone. With a smoothly curved transition, a loss coefficient k_c as small as 0.05 is possible. For conical reducers, a minimum k_c of about 0.10 is possible, with a total cone angle of 20–40°. Smaller or larger total cone angles result in higher values of k_c.

The nozzle at the end of a pipeline (see Fig. S8.11*b*) is a special case of gradual contraction. An equation like Eq. (8.75) also governs the head loss through a nozzle at the end of a pipeline, where k_c becomes the ***nozzle loss coefficient*** k_n whose value commonly ranges from 0.04 to 0.20 and $V_2 = V_j$ is the jet velocity.[24] However, we can *not* regard the head loss through a nozzle h_n as a minor loss, because the jet velocity head is usually quite large (see, e.g., Sample Prob. 8.12). More details on the flow through nozzles is presented in Sec. 11.6.

8.24 Loss Due to Expansion

Sudden Expansion

The conditions at a sudden expansion are shown in Fig. 8.17. There is a rise in pressure because of the decrease in velocity, but this rise would be even greater if there were not the loss in energy. There is excessive turbulence in the flow from C to F, beyond which the flow is normal. The drop in pressure just beyond section C, which was measured by a piezometer not shown in the figure, is due to the fact that the pressures at the wall of the pipe are in this case less than those in the center of the pipe because of centrifugal effects.

Figures 8.16 and 8.17 are both drawn to scale from test measurements for the same diameter ratios and the same velocities, and they show that the loss due to sudden expansion is greater than the loss due to a corresponding contraction. This is so because of the inherent instability of flow in an expansion, where diverging flow paths encourage the formation of eddies *within* the flow. Moreover, separation of the flow from the wall of the conduit creates pockets of

[24] See also Eq. (11.14).

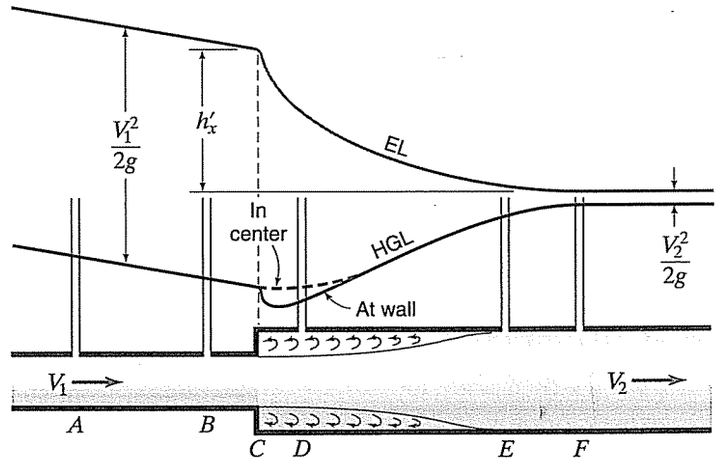


Figure 8.17
Loss due to sudden enlargement. (*Plotted to scale. Velocity is the same as in Fig. 8.16.*)

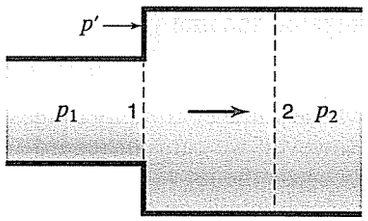


Figure 8.18

eddying turbulence outside the flow region. In converging flow, on the other hand, there is a dampening effect on eddy formation, and the conversion from pressure energy to kinetic energy is more efficient.

We can derive an expression for the loss of head in a sudden enlargement as follows. In Fig. 8.18, section 2 corresponds to section F in Fig. 8.17, which is a section where the velocity profile has become normal again, and marks the end of the region of excess energy loss due to the turbulence created by the sudden enlargement. In Fig. 8.18 let us assume that the pressure at section 2 for ideal flow (without friction) is p_0. Then from Eq. (5.30) in this ideal case

$$\frac{p_0}{\gamma} = \frac{p_1}{\gamma} + \frac{V_1^2}{2g} - \frac{V_2^2}{2g}$$

If in the actual case, however, the pressure at section 2 is p_2, while the average pressure on the annular ring is p' then, equating the resultant force on the body of fluid between sections 1 and 2 to the time rate of change of momentum between sections 1 and 2 per Eq. (6.7a), we obtain

$$p_1A_1 + p'(A_2 - A_1) - p_2A_2 = \frac{\gamma}{g}(A_2V_2^2 - A_1V_1^2)$$

From this,

$$\frac{p_2}{\gamma} = \frac{A_1}{A_2}\frac{p_1}{\gamma} + \frac{A_2 - A_1}{A_2}\frac{p'}{\gamma} + \frac{A_1}{A_2}\frac{V_1^2}{g} - \frac{V_2^2}{g}$$

The loss of head is caused by friction, and so it is given by the difference between the ideal and actual pressure heads (or total heads) at section 2. Thus $h_x' = (p_0 - p_2)/\gamma$, and noting that

$$A_1V_1 = A_2V_2$$

and that $A_1V_1^2 = A_1V_1V_1 = A_2V_2V_1$, we obtain, from substituting the preceding expressions for p_0/γ and p_2/γ into $(p_0 - p_2)/\gamma$,

$$h_x' = \frac{(V_1 - V_2)^2}{2g} + \left(1 - \frac{A_1}{A_2}\right)\left(\frac{p_1}{\gamma} - \frac{p'}{\gamma}\right)$$

We usually assume that $p' = p_1$, in which case the loss of head due to sudden enlargement is

$$h_x' = \frac{(V_1 - V_2)^2}{2g} \tag{8.76}$$

Although it is possible that, under some conditions, p' will equal p_1, it is also possible for it to be either more or less than that value, in which case the loss of head will be either less or more than that given by Eq. (8.76). The exact value of p' will depend on the manner in which the fluid eddies move around in the corner adjacent to this annular ring. However, the deviation from Eq. (8.76) is quite small and of negligible importance.

The discharge loss into still water of Sec. 8.22 is a special case where A_2 is infinite compared with A_1, or $V_2 = 0$, so that Eq. (8.76) will reduce to Eq. (8.71).

On occasion we may need to express this minor loss all in terms of one velocity (see, e.g., Sec. 8.27). If so, we can use continuity, $(\pi/4)D_1^2V_1 = (\pi/4)D_2^2V_2$, to obtain

$$(V_1 - V_2)^2 = \left(1 - \frac{D_1^2}{D_2^2}\right)^2 V_1^2 = \left(\frac{D_2^2}{D_1^2} - 1\right)^2 V_2^2 \tag{8.77}$$

Gradual Expansion

To minimize the loss accompanying a reduction in velocity, we can use a ***diffuser*** such as that shown in Fig. 8.19. The diffuser may have a curved outline, or it may be a frustum of a cone (straight sided). In Fig. 8.19 the loss of head will be some function of the angle of divergence and also of the ratio of the two areas, the length of the diffuser being determined by these two variables.

In flow through a diffuser we can consider the total loss as made up of two components. One is the ordinary pipe-friction loss, which we can represent by

$$h_f = \int \frac{f}{D}\frac{V^2}{2g}\,dL$$

In order to integrate the foregoing, we need to express the variables f, D, and V as functions of L. For our present purpose, it is sufficient, however, merely to

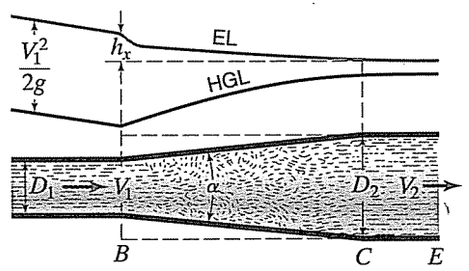


Figure 8.19
Loss due to gradual enlargement.

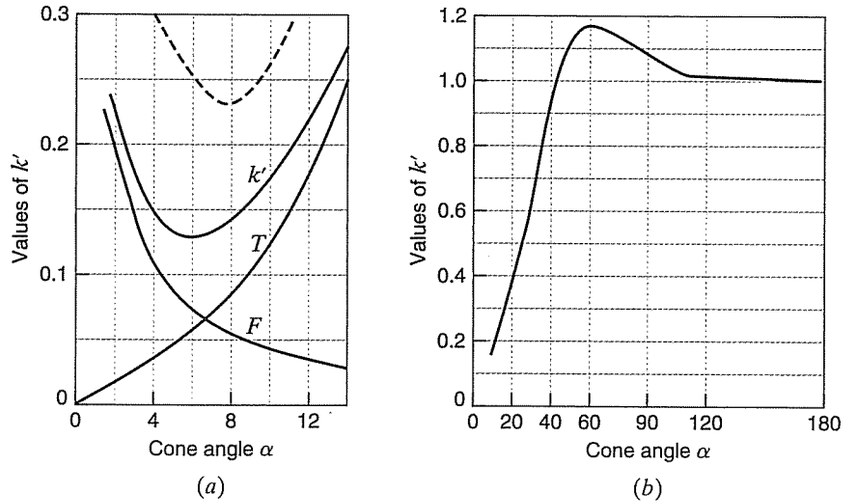


Figure 8.20
Loss coefficient for conical diffusers.

note that the friction loss increases with the length of the cone. Hence, for given values of D_1 and D_2, the larger the angle of the cone, the less its length and the less the pipe friction, which we can see from the curve marked F in Fig. 8.20*a*. However, in flow through a diffuser there is an additional loss component, due to turbulence set up by induced currents that produce a vortex motion over and above that which normally exists. This additional turbulence loss will naturally increase with the degree of divergence, as we can see from the curve marked T in Fig. 8.20*a*, and if the rate of divergence is great enough then the flow may separate from the walls with eddies flowing backward along the walls (see Sec. 9.6). The total loss in the diverging cone consists of the sum of these two losses, marked k'. This has a minimum value at 6° for the particular case chosen, which is for a very smooth surface. If the surface were rougher, the value of the friction F would be larger. This increases the value of k', as in the dashed curve, and also shifts the angle for minimum loss to 8°. Thus the best angle of divergence increases with the roughness of the surface.

We have seen that the loss due to a sudden enlargement is very nearly equal to $(V_1 - V_2)^2/2g$. The loss due to a gradual enlargement is

$$h' = k'\frac{(V_1 - V_2)^2}{2g} \tag{8.78}$$

Values of k' as a function of the cone angle α are given in Fig. 8.20*b*, for a wider range than appears in Fig. 8.20*a*. Note, interestingly, that at an angle slightly above 40° the loss is the same as that for a sudden enlargement, which is 180°, and that between these two the loss is greater than for a sudden enlargement, being a maximum at about 60°. This is a result of the induced currents that develop.

SAMPLE PROBLEM 8.10 Water flows through a 150-mm-diameter pipe at 100 L/s. (*a*) If the pipe suddenly enlarges to 200 mm diameter, what is the head loss? (*b*) If the same enlargement is via a diffuser with a total cone angle of 20°, what is the head loss then?

Solution

(*a*)
$$V_1 = \frac{Q}{A_1} = \frac{4Q}{\pi D_1^2} = \frac{4(0.100)}{\pi(0.15)^2} = 5.66 \text{ m/s}$$

$$V_2 = \frac{4(0.100)}{\pi(0.20)^2} = 3.18 \text{ m/s}$$

Eq. (8.76):
$$h'_x = \frac{(5.66 - 3.18)^2}{2(9.81)} = 0.312 \text{ m} \qquad \textit{ANS}$$

(*b*) Fig. 8.20 for $\alpha = 20°$: $k' = 0.38$

Eq. (8.78):
$$h' = 0.38(0.312) = 0.1187 \text{ m} \qquad \textit{ANS}$$

EXERCISES

8.24.1 Two pipes with a diameter ratio of 1:2 are connected in series (Fig. X8.24.1). With a velocity of 6.8 m/s in the smaller pipe, find the loss of head due to (*a*) sudden contraction; (*b*) sudden enlargement; (*c*) expansion in a conical diffuser with a total angle of 30°, and of 10°.

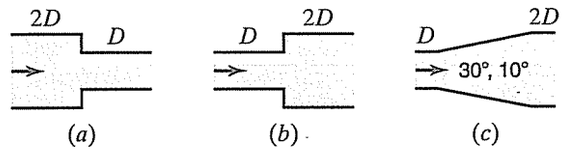


Figure X8.24.1

8.24.2 A 5-in-diameter pipe ($f = 0.033$) 110 ft long connects two reservoirs whose water-surface elevations differ by 12 ft (Fig. X8.24.2). The pipe entrance is flush, and the discharge is submerged. (*a*) Compute the flow rate. (*b*) If the last 10 ft of pipe were replaced with a conical diffuser with a cone angle of 10°, compute the flow rate.

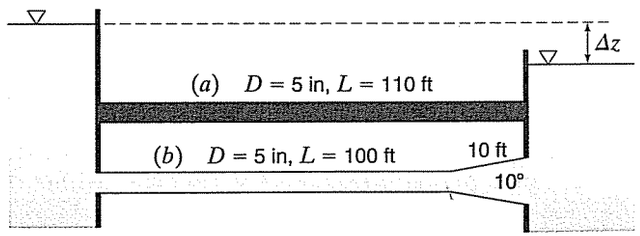


Figure X8.24.2

8.25 LOSS IN PIPE FITTINGS

For pipe fittings we can express the loss of head as $kV^2/2g$, where V is the velocity in a pipe of the nominal size of the fitting. Table 8.3 gives typical values of k. As an alternative, we may account for the head loss due to a fitting by increasing the pipe length by amounts given by L/D in the table. However, we should recognize that these fittings create so much turbulence that the loss caused by them is proportional to V^2, and so we should restrict this latter method to cases where the pipe friction itself is in the zone of complete turbulence (fully-rough-pipe flow). For very smooth pipes it is better to use the k values when determining the loss through fittings.

TABLE 8.3 **Values of loss factors for pipe fittings**

Fitting	k	L/D
Globe valve, wide open	10	350
Angle valve, wide open	5	175
Close-return bend	2.2	75
T, through side outlet	1.8	67
Short-radius elbow	0.9	32
Medium-radius elbow	0.75	27
Long-radius elbow	0.60	20
45° elbow	0.42	15
Gate valve, wide open	0.19	7
half open	2.06	72

8.26 LOSS IN BENDS AND ELBOWS

In flow around a bend or elbow, because of centrifugal effects (Sec. 5.16), there is an increase in pressure along the outer wall and a decrease in pressure along the inner wall. In Fig. 8.21 we see the centrifugal forces acting on a number of fluid particles, each of mass m on pipe diameter CE that is normal to the pipe's plane of curvature. The centrifugal force mV^2/r on the particles near the center of the pipe, where the velocities are high, is larger than the centrifugal force on the particles near the walls of the pipe, where the velocities are low (see Fig. 8.10). Because of this unbalanced condition, a secondary flow[25] develops, as shown in the cross section of Fig. 8.21. This combines with the axial velocity to form a double spiral flow, which persists for some distance. Thus not only is there some loss of energy within the bend itself, but this distorted flow persists for some distance downstream until viscous friction dissipates it. The velocity in the pipe may not become normal again within as much as 100 pipe diameters

[25] Secondary flow in the bends of open channels is discussed in Sec. 10.21.

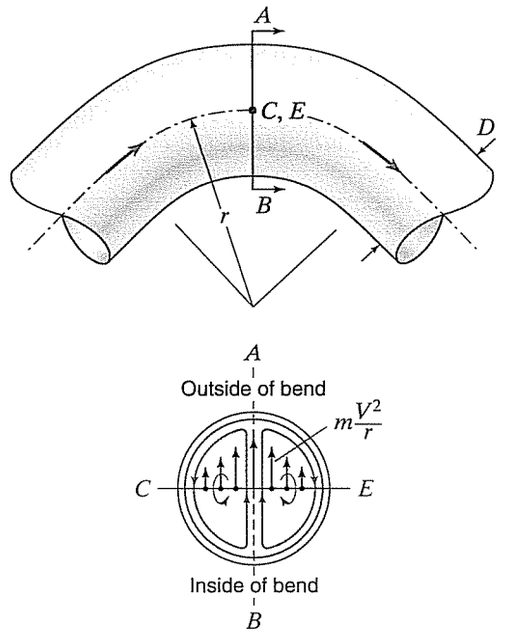


Figure 8.21
Secondary flow in a pipe bend.

downstream from the bend. In fact, more than half the friction loss of a bend or elbow occurs in the straight pipe following it.

The pressure difference that develops between A and B (Fig. 8.21), on the outside and inside of the bend, is used as the basis of the ***elbow meter*** (Sec. 11.16), which measures discharge.

We may eliminate most of the head loss due to a sharp bend by using a vaned elbow, such as that in Fig. 8.22. The vanes impede the formation of the secondary flows that would otherwise develop.

Note that the amount of head loss that a pipe bend or elbow causes, as given by

$$h_b = k_b \frac{V^2}{2g} \tag{8.79}$$

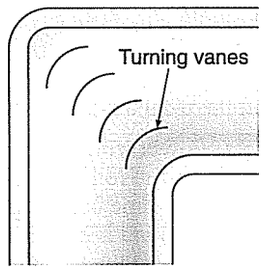


Figure 8.22
Vaned elbow.

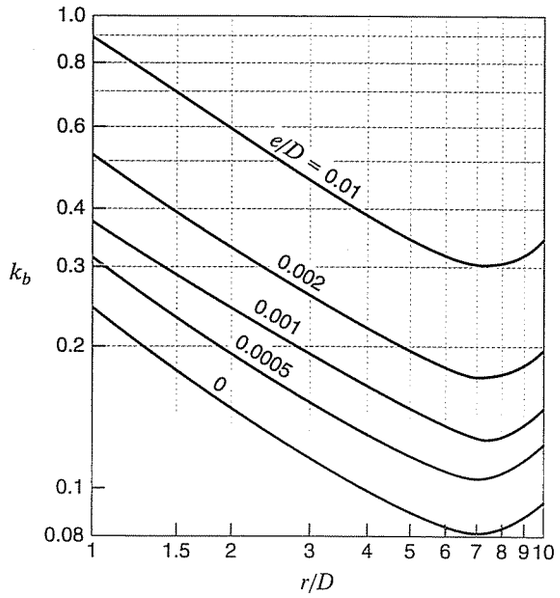


Figure 8.23
Resistance coefficients for 90° bends (resistance due to length of pipe $\pi r/2$ in the bend must be added, where r is the radius to the pipe centerline). (*From Ref. 46.*)

is the amount *in excess of the loss for an equal length of straight pipe.* It depends strongly on the ratio of the radius of curvature r (Fig. 8.21) to the diameter of the pipe D. Note also, that we cannot treat combinations of different bends placed close together by adding up the losses of each one considered separately. The total loss depends not only on the spacing between the bends, but also on the relative directions of the bends and the planes containing them. Bend loss is not proportional to the angle of the bend; for 22.5° and 45° bends the losses are respectively about 40% and 80% of the loss in a 90° bend. Figure 8.23 gives values of k_b for 90° bends, varying with bend radius and pipe roughness.

EXERCISES

8.26.1 Water at 72°F flows through a 110-ft-long, 5-in-diameter wrought-iron pipe that contains the following fittings: one open globe valve, one medium-radius elbow, and one 90° pipe bend ($k_b = 0.13$) with a radius of curvature of 45 in. The length of the bend is not included in the 110 ft. There are no entrance or discharge losses. Find the total head loss if the flow velocity is 4.8 fps.

8.26.2 Water at 18°C flows through a 25-m-long, 75-mm-diameter commercial steel pipe that contains the following fittings: one open angle valve, one short-radius elbow, and one 90° pipe bend ($k_b = 0.12$) with a radius of curvature of 600 mm. The length of the bend is not included in the 25 m. There are no entrance or discharge losses. Find the total head loss if the flow velocity is 2 m/s.

8.27 SINGLE-PIPE FLOW WITH MINOR LOSSES

We have examined the fundamental fluid mechanics associated with the frictional loss of energy in single-pipe flow, caused by both the wall roughness of the pipes and by pipe fittings and the like that disturb the flow (minor losses). While the interest of the scientist extends very little beyond this, it is the task of the engineer to apply these fundamentals to various types of practical problems.

Specialists generally concede that for pipes longer than 1000 diameters, the error from neglecting minor losses is less than that inherent in selecting a value for the friction factor (f, n, or C_{HW}). In applying this rule, one must of course use common sense and recall that a valve, for example, is a minor loss only when it is wide open; partially closed, it may be the most important loss in the system. When minor losses are negligible, as they often are, we can solve pipe-flow problems by the methods of Secs. 8.13–8.19.

When we *include* minor losses, the total head h_L loss between two points is the sum of the pipe (wall) friction head loss h_f plus the minor losses, or

$$h_L = h_f + \sum h' \tag{8.80}$$

We can represent the pipe friction loss by a number of different equations, and it may depend on a number of different factors, as noted in Secs. 8.1–8.19. And, as we noted in Sec. 8.20, we can represent the minor losses as a coefficient multiplied by the velocity head, $kV^2/2g$, or as an equivalent length of pipe, expressed as a number of diameters, ND. As a result, the right-hand side of Eq. (8.80) can take on many forms. Because of the extra term added by the minor losses, such forms of this equation are often more challenging to solve than those that omit minor losses (Secs. 8.14–8.19).

Nonrigorous Equations. In problems where a constant f value is given, Eq. (8.80) still has only one unknown, namely, h_L or V or Q or D. In most cases this equation is explicit in the unknown (Appendix B), and so is easy to solve. But for sizing problems, the resulting equation in D is of the fifth degree and implicit, requiring trial and error or an equation solver as discussed in Sec. 8.17.

The empirical equations (Secs. 8.18–8.19) do not necessarily continue to yield direct solutions when minor losses are included. By including $kV^2/2g$ or ND in Eq. (8.65) with Eqs. (8.67) and (8.68), we find that these equations are always explicit for Type 1 (find h_L now, not just h_f), usually explicit for Type 2 (find Q or V), and never explicit for Type 3 (find D). The exception with Type 2 occurs when using the Hazen-Williams equation and representing minor losses by $kV^2/2g$; this is because, in Eq. (8.65) with (8.67), we find that the pipe friction loss is proportional to $V^{1.852}$, so the two terms contain V to different inconvenient powers. Type 3 problems are more complex in part owing to the fact that Q is given rather than V, since this causes V to also be a function of D. We first discussed these three problem categories in Sec. 8.15.

Rigorous Equations. In practice, we may need to take into account the variation of f, which can vary by as much as a factor of five for smoother pipes

(see Fig. 8.11). When including minor losses, methods involving the Darcy-Weisbach equation (8.13) and an unknown friction factor f remain explicit or direct for Type 1 problems (find h_L), but for the other two types of problems (find Q or V or D) they are always implicit (Appendix B). The other governing equations are the same as in Sec. 8.14.

Implicit Equations. To solve the many implicit equations that arise when we include minor losses, it is clear we must use iterative procedures, either manually or programmed. Manual iterative procedures (trial and error) for this purpose are generally like those described in Sec. 8.15. But occasionally it will be easier to make trials for D rather than f.

Automated iterative procedures are available in equation solving software packages, and in programmable calculators with an equation solving function, as discussed in Appendixes C and D and Sec. 8.17. These unquestionably are the most convenient, and we can expect them to become more accessible in time.

When using an equation solving software package like Mathcad or Excel (Appendixes C and D), we just need to specify n equations in n unknowns, as described in Sec. 8.17. We merely have to type in with the energy equation the terms added to account for minor losses, and then assign values to any new loss coefficients. Otherwise, the procedure is unchanged.

To use an equation solver in a programmable calculator, we recall from Appendix C.1 and Sec. 8.17 that for the implicit, nonlinear equations we must deal with there can be only one equation in one unknown. Similarly to the development of Eq. (8.56), we can develop a "universal" turbulent flow equation for use in an equation solver, including minor losses, by eliminating h_f and f from Eq. (8.80) with the help of Eqs. (8.13), (8.51), and (4.7). Expressing minor losses $\Sigma h'$ in terms of $\Sigma kV^2/2g$, we obtain

Turbulent flow:

$$\sqrt{\frac{L/D}{\dfrac{\pi^2 g D^4 h_L}{8Q^2} - \Sigma k}} = -2\log\left(\frac{e/D}{3.7} + \frac{2.51\pi\nu D}{4Q}\sqrt{\frac{L/D}{\dfrac{\pi^2 g D^4 h_L}{8Q^2} - \Sigma k}}\right) \tag{8.81}$$

If we express minor losses in terms of an equivalent length of pipe, ND, then we obtain

Turbulent flow:

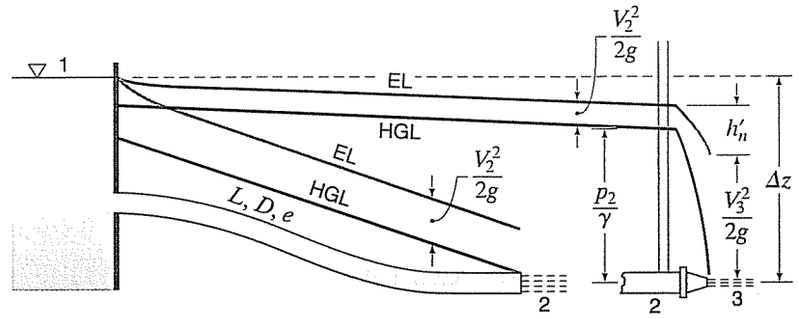
$$\frac{4Q}{\pi D^2}\sqrt{\frac{(L/D)+N}{2gh_L}} = -2\log\left(\frac{e/D}{3.7} + \frac{2.51\nu}{D}\sqrt{\frac{(L/D)+N}{2gh_L}}\right) \tag{8.82}$$

Each of these equations involves eight variables: h_L, Q, D, L, e, ν, g, and Σk or N. We note that the equations are implicit in h_L, Q, and D, but, with an equation solver we may easily solve for any one of these three quantities if the rest of the variables are known. Similar and less complex "universal" equations can also be developed using the Haaland equation (8.52); however, they sometimes yield false results when large diameters cause the argument of the logarithm to be

very close to unity. An important reminder when using these equations is to use Eqs. (4.7) and (8.1) to check the Reynolds number and confirm that the flow is turbulent. If $\mathbf{R} < 2000$, the flow is laminar (Sec. 8.2), and we must instead solve the problem with Eq. (8.28) and Eq. (4.3) if needed.

The following sample problem illustrates the method of solution for flow through a pipeline of uniform diameter with minor losses.

SAMPLE PROBLEM 8.11 Water at 60°F flows through the new 10-in-diameter cast-iron pipe sketched in Fig. S8.11*a*. The pipe is 5000 ft long, its entrance is sharp-cornered but nonprojecting, and $\Delta z = 260$ ft. Find the flow rate using (*a*) only Fig. 8.11 and a basic scientific calculator; (*b*) an equation solver on a programmable calculator; (*c*) equation solving software such as Mathcad.



(*a*) Free discharge (*b*) With nozzle

Figure S8.11
Discharge from a reservoir. As L/D gets larger the EL and HGL approach one another.

Solution
This is a Type 2 problem with minor losses.

Table 8.1 for cast iron: $e = 0.00085$ ft; $e/D = 0.00085/(10/12) = 0.00102$

Sec. 8.21 for the square-edged entrance: $k_e = 0.5$

Energy Eq. (5.28) from the water surface to the free jet at point 2:

$$260 + 0 + 0 - \left(0.5 + f\frac{5000}{10/12}\right)\frac{V_2^2}{2g} = 0 + 0 + \frac{V_2^2}{2g} \tag{1}$$

▦: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

(*a*) Rearranging: $\dfrac{V_2^2}{2g} = \dfrac{260}{1.5 + 6000f}$; $V_2 = \sqrt{\dfrac{2(32.2)260}{1.5 + 6000f}}$

Fig. 8.11, right-hand side, for $e/D = 0.00102$: $f_{min} \approx 0.0196$

$$\text{Try } f = 0.0196: \quad V_2 = \sqrt{16{,}744/[1.5 + 6000(0.0196)]} = 11.86 \text{ fps}$$

$$D''V = 10(11.86) = 118.6$$

Fig. 8.11 for $e/D = 0.00102$ and $D''V = 118.6$: $f = 0.020$. Obtained and assumed f values are different, so we must try again. Tabulating all the trials:

Try f	V, ft/s	$D''V$	Obtained f	
0.0196	11.86	118.6	0.020	Try again
0.020	11.74	117.4	0.020	Converged!

Eq. (4.3): $$Q = \frac{\pi}{4}\left(\frac{10}{12}\right)^2(11.74) = 6.40 \text{ cfs} \qquad \textit{ANS}$$

Alternatively, replace use of the chart by equations as follows:

Table A.1 for water at 60°F: $\nu = 1.217 \times 10^{-5}$ ft^2/sec

Eq. (8.54) for $e/D = 0.00102$: $f_{min} = 0.01973$

Energy equation with assumed f gives V (as before); $\mathbf{R} = DV/\nu$

Insert values of e/D and $\mathbf{R}$ into Haaland equation (8.52) to get a calculated f; compare with assumed f. If the difference is less than about 0.5%, we have convergence; otherwise repeat.

(*b*) Using an equation solver on a programmable calculator (Appendixes C.1 and D.1), follow the procedure of Sample Prob. 8.9*a*, which eliminates f and $\mathbf{R}$ and V, but with an equation that includes minor losses. Equation (8.81) does not appear to be appropriate, because it does not include for the change in velocity head from zero at point 1 to $V_2^2/(2g)$ at point 2. However, from preceding Eq. (1) we see that we *can* include for this by increasing $\sum k$ by one. Assign to the seven known variables the values (without units) $g = 32.2$ ft/sec^2, $\nu = 1.217 \times 10^{-5}$ ft^2/sec, $h_L = \Delta z = 260$ ft, $\sum k = k_e + 1 = 1.5$, $L = 5000$ ft, $e = 0.00085$ ft, and $D = 10/12$ ft. Assign an estimated value of 10 cfs (say) to the unknown Q.

Instruct the solver to solve Eq. (8.81) for Q. The result is $Q = 6.39$ cfs. But we must check the $\mathbf{R}$. Equation (4.7): $V = 4Q/(\pi D^2) = 11.71$ fps, so $\mathbf{R} = DV/\nu = 8.02 \times 10^5$, and the turbulent flow assumption is correct. Therefore

$$Q = 6.39 \text{ cfs} \qquad \textit{ANS}$$

(*c*) Using equation solving software such as Mathcad (Appendixes C.2–C.3 and D.2–D.3), we establish a "solve block" containing four equations as in Sample Prob. 8.9*b*, except that in this case we replace Eq. (8.13) by preceding energy equation (1). Also, we must change the "Find" statement to: Find (f, V, R, Q). We assign the seven known variables the same values as in solution (*b*), and assign estimated values (without units) to the four unknown variables, say $f = 0.030$, $V = 5$ fps, $R = 100{,}000$, and $Q = 10$ cfs.

The program delivers the result:

$$f = 0.0201, \quad V = 11.71 \text{ fps}, \quad R = 801{,}731, \quad Q = 6.39 \text{ cfs} \qquad \textit{ANS}$$

Because this $\mathbf{R}$ is for turbulent flow, the equation (8.51) we used for f is valid, and so therefore is the answer we obtained for Q.

Note the simplicity and clarity of the Mathcad solution (*c*) in comparison to the other two solutions.

If the pipe in this example discharged into a fluid that was at a pressure other than atmospheric, we would have to use the proper value of p_2/γ in the energy equation.

Another example of flow from a reservoir is that of a pipeline (known as a ***penstock***) leading to an impulse turbine (Sec. 16.2). In this case the pipe does not discharge freely, but ends in a nozzle (Fig. S8.11*b*), which has a known or assumed loss coefficient, k_n. The head loss in the nozzle, h_n, is associated with the high issuing velocity head, $V_j^2/2g$, and is therefore not a minor loss (Sec. 8.23). We use the equation of continuity to write all losses in terms of the velocity head in the pipe. This is the logical choice for the "common unknown," because we will again build the trial-and-error solution around the pipe friction loss rather than the nozzle loss.

SAMPLE PROBLEM 8.12 As in Fig. S8.11*b*, suppose that the pipeline of Sample Prob. 8.11 is now fitted with a nozzle, at the end of which discharges a 2.5-in-diameter jet and that has a loss coefficient of $k_n = 0.11$. Find the flow rate using only Fig. 8.11 and a basic scientific calculator.

Solution

Referring to Sample Prob. 8.11, let point 2 now be on the pipe at the base of the nozzle and point 3 be in the jet. The head loss in the nozzle is $0.11\,V_3^2/2g$ (Sec. 8.23). Writing the energy equation (5.28) between 1 and 3,

$$260 + 0 + 0 - \left(0.5\frac{V_2^2}{2g} + 6000f\frac{V_2^2}{2g} + 0.11\frac{V_3^2}{2g}\right) = 0 + 0 + \frac{V_3^2}{2g}$$

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Using continuity Eq. (4.17), $V_3^2/2g = (10/2.5)^4 V_2^2/2g = 256V_2^2/2g$. Thus

$$260 = (0.5 + 1.11 \times 256 + 6000f)\frac{V_2^2}{2g}$$

Select a trial value of f. Let $f = 0.02$ for the first assumption. Then $260 = (285 + 120)V_2^2/2g$, from which

$$\frac{V_2^2}{2g} = \frac{260}{405} = 0.643 \text{ ft}$$

and $V_2 = \sqrt{(2(32.2)0.643} = 6.43$ fps. With $D''V = 10 \times 6.43 = 64.3$ and $e/D = 0.001$, Fig. 8.11 shows $f = 0.02$. In this case we can consider the first solution sufficiently accurate, but in general the value of f we determine from the chart may be quite different from that we assume, and we may need to make one or two more trials.

The rate of discharge is $Q = A_2V_2 = 0.545 \times 6.43 = 3.51$ cfs ***ANS***

Then $$V_3 = 16V_2 = 16 \times 6.43 = 102.9 \text{ fps}$$

In addition, $H_2 = p_2/\gamma + V_2^2/2g = 260 - (0.5 + 0.02 \times 6000)0.643 = 182.58$ ft, and the pressure head $p_2/\gamma = 182.58 - 0.643 = 181.93$ ft.

This example shows that the addition of the nozzle reduces the discharge from 6.40 to 3.51 cfs, but increases the jet velocity from 11.74 to 102.9 fps. The head loss due to pipe friction is 77.1 ft and the head loss through the nozzle is 18.09 ft.

When solving Type 3 sizing problems, in general, the diameter we obtain will not be a standard pipe size, and the size we select will usually be the next largest commercially available size. In planning for the future, we must remember that scale deposits will increase the roughness and reduce the cross-sectional area. For pipes in water service, the absolute roughness e of old pipes (20 years and more) may increase over that of new pipes by three-fold for concrete or cement-lined steel, up to 20-fold for cast iron, and even to 40-fold for tuberculated wrought-iron and steel pipe. Substituting $V = 4Q/(\pi D^2)$ into Eq. (8.13) shows that for a constant value of f, Q varies as $D^{5/2}$. Hence for the case where minor losses are negligible and f is constant, to achieve a 100% increase in flow, we need to increase the diameter by only 32%. This amounts to a 74% increase in cross-sectional area.

EXERCISES

8.27.1 An 8-in-diameter pipeline ($f = 0.028$) 500 ft long discharges a 3-in-diameter water jet into the atmosphere at a point that is 250 ft below the water surface at intake (Fig. X8.27.1). The entrance to the pipe is reentrant, with $k_e = 0.9$, and the nozzle loss coefficient is 0.045. Find the flow rate and the pressure head at the base of (entrance to) the nozzle.

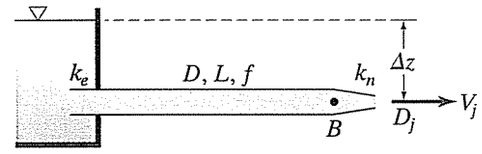


Figure X8.27.1

8.27.2 A 180-mm-diameter pipeline ($f = 0.032$) 150 m long discharges a 60-mm-diameter water jet into the atmosphere at a point that is 80 m below the water surface at intake (Fig. X8.27.1). The entrance to the pipe is reentrant, with $k_e = 0.9$, and the nozzle loss coefficient is 0.055. Find the flow rate and the pressure head at the base of (entrance to) the nozzle.

8.27.3 A horizontal 4-in-diameter pipe ($f = 0.028$) projects into a body of water 2.5 ft below the surface (Fig. X8.27.3). Considering all losses, find the pressure at a point 15 ft from the end of the pipe if the velocity is 12 fps and the flow is (*a*) into the body of water; (*b*) out of the body of water.

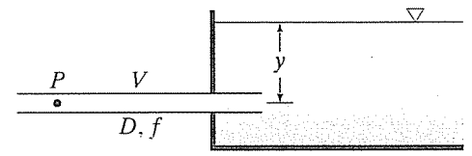


Figure X8.27.3

8.27.4 A horizontal 100-mm-diameter pipe ($f = 0.027$) projects into a body of water 1 m below the surface (Fig. X8.27.3). Considering all losses, find the pressure at a point 5 m from the end of the pipe if the velocity is 4 m/s and the flow is (*a*) into the body of water; (*b*) out of the body of water.

8.27.5 A 450-ft-long pipeline runs between two reservoirs, both ends being under water, and the intake end is square-edged and nonprojecting. The difference between the water surface levels of the two reservoirs is 150 ft. (*a*) What is the discharge if the pipe diameter is 12 in and $f = 0.028$? (*b*) When this same pipe is old, assume that the growth of tubercles has reduced the diameter to 11.25 in and that $f = 0.06$. What will the rate of discharge be then?

8.27.6 Solve Exer. 8.27.5*a* when f is unknown but given that $e = 0.005$ ft and that the water temperature is 60°F. What, then, is the value of f?

8.28 PIPELINE WITH PUMP OR TURBINE

If a pump lifts a fluid from one reservoir to another, as in Fig. 8.24, not only does it do work in lifting the fluid the height Δz, but it also has to overcome the total friction head loss Σh_L (including minor losses) in the suction and discharge pipelines. This head loss is equivalent to some added lift, so that the effect is the

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

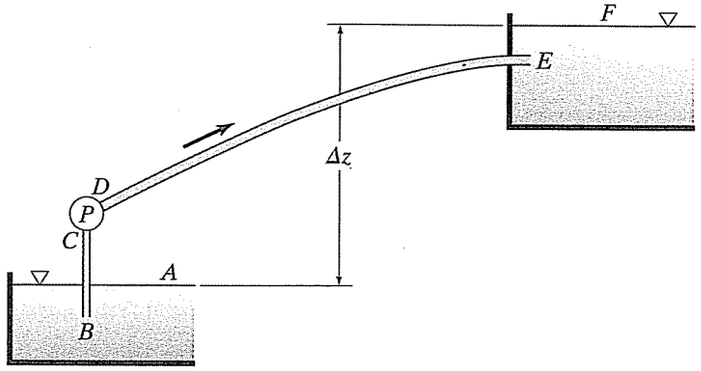


Figure 8.24
Pipeline with pump between two reservoirs.

same as if the pump lifted the fluid a height $\Delta z + \sum h_L$. Hence the power the pump delivers to the liquid is $\gamma Q(\Delta z + \sum h_L)$. The power required to run the pump is greater than this, depending on the efficiency of the pump. The total pumping head h_p for this case is

$$h_p = \Delta z + \sum h_L \tag{8.83}$$

If the pump discharges a stream through a nozzle, as in Fig. 8.25, not only does it lift the liquid a height Δz, but it also imparts a kinetic energy head of

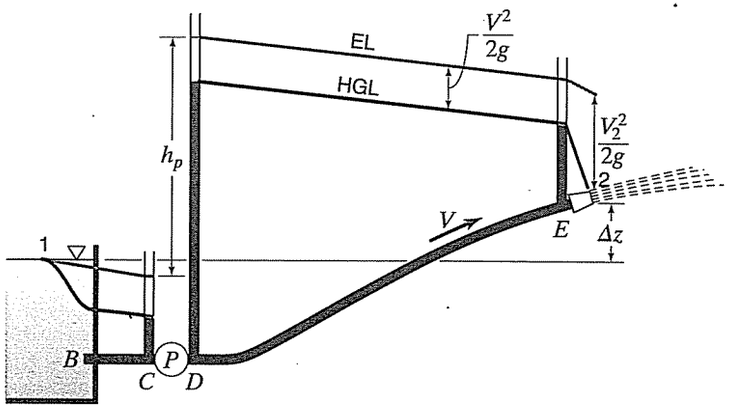


Figure 8.25
Pipeline with pump and nozzle.

$V_2^2/2g$, where V_2 is the velocity of the jet. Thus the total pumping head is now

$$h_p = \Delta z + \frac{V_2^2}{2g} + \sum h_L \tag{8.84}$$

In any case we can determine the total pumping head by writing the energy equation between any point upstream from the pump and any other point downstream, as in Eq. (5.37). For example, if the upstream reservoir were at a higher elevation than the downstream one, then the Δzs in the two foregoing equations would have negative signs.

The machine we use to convert flow energy into mechanical work we call a ***turbine.*** In flowing from the upper tank in Fig. 8.26 to the lower, the fluid loses potential energy head equivalent to Δz. Part of this energy is lost to hydraulic friction in the pipe, and the remainder reaches the turbine. Of that part which reaches the turbine, part is lost in hydraulic friction within the turbine, and the rest converts into mechanical work.

The power that reaches the turbine is the initial power minus the friction losses (pipe plus minor) in the pipeline, or $\gamma Q(\Delta z - \sum h_L)$. The power the *turbine* delivers is less than this, depending on both its hydraulic and mechanical losses. The head under which the turbine operates is

$$h_t = \Delta z - \sum h_L \tag{8.85}$$

Here $\sum h_L$ is the loss of head in the supply line plus the submerged discharge loss (pipe friction and minor losses), but it does *not* include the head loss in the ***draft tube*** (*DE* in Fig. 8.26), since the draft tube is considered an integral part of the turbine. A draft tube has a gradually increasing cross-sectional area, which reduces the velocity at discharge. This enhances the efficiency of the turbine because it reduces the head loss at discharge (Secs. 8.22 and 16.8). Note that the h_t of Eq. (8.85) represents the energy head the turbine removes from the fluid; this, of course, is the same as the energy head the fluid transfers to the turbine.

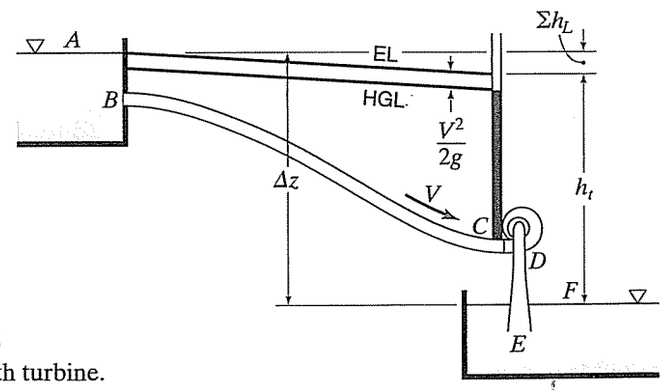


Figure 8.26
Pipeline with turbine.

SAMPLE PROBLEM 8.13 The tanks, pump, and pipelines of Fig. S8.13 have the characteristics noted. The suction line entrance from the pressure tank is flush, and the discharge into the open tank is submerged. If the pump P puts 2.0 hp into the liquid, (*a*) determine the flow rate and (*b*) find the pressure in the pipe on the suction side of the pump.

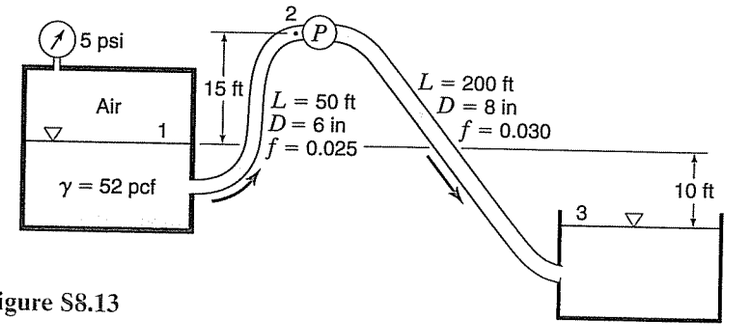


Figure S8.13

Solution

Eq. (5.40): $$P = \frac{\gamma Q h_p}{550} = 2.0 = \frac{52 Q h_p}{550}$$

Thus $$h_p = \frac{21.2}{Q}$$

Figure 8.13 for flush entrance: $k_e = 0.5$

Writing energy Eq. (5.27) from 1 (datum) to 3,

$$\frac{5(144)}{52} + 0 + 0 + h_p - 0.5\frac{V_6^2}{2g} - 0.025\left(\frac{50}{6/12}\right)\frac{V_6^2}{2g} - 0.030\left(\frac{200}{8/12}\right)\frac{V_8^2}{2g} - \frac{V_8^2}{2g} = 0 - 10 + 0$$

Eq. (4.6): $$V_6 = \frac{Q}{A_6} = \frac{Q}{0.1963} \quad \text{and} \quad V_8 = \frac{Q}{0.349}$$

Substituting for h_p, V_6, V_8, and $g = 32.2$, the energy equation reduces to

$$23.8 + \frac{21.2}{Q} - 2.48Q^2 = 0$$

which we can rewrite as the cubic expression

$$2.48Q^3 - 23.8Q - 21.2 = 0$$

▦: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Solving this by trial and error (see Sample Probs. 3.5 and 5.8) or by equation/polynomial solver (Appendix D), $Q = 3.47$ cfs

From Eq. (B.7) the other two roots are $Q = -2.48$ and -0.987 cfs.

So $$Q = 3.47 \text{ cfs} \quad \textit{ANS}$$

(*b*) To obtain the pressure p_2 at the suction side of the pump,

$$V_6 = \frac{3.47}{0.1963} = 17.68 \text{ fps}$$

Writing energy Eq. (5.28) from 1 to 2,

$$\frac{5(144)}{52} + 0 + 0 - 0.5\frac{17.68^2}{2(32.2)} - 0.025\left(\frac{50}{6/12}\right)\frac{17.68^2}{2(32.2)} = \frac{p_2}{\gamma} + 15 + \frac{17.68^2}{2(32.2)}$$

from which $$\frac{p_2}{\gamma} = -20.6 \text{ ft}; \quad p_2 = -20.6\left(\frac{52}{144}\right) = -7.43 \text{ psi} \quad \textit{ANS}$$

From facing inside cover:

This is equivalent to $7.43\left(\frac{29.9}{14.70}\right) = 15.13$ inHg vacuum.

Note: We should check the absolute pressure against the vapor pressure of the liquid (Sec. 5.10) to ensure that vaporization does not occur at point 2.

EXERCISES

8.28.1 An 80-in-diameter pipe ($f = 0.025$) 7252 ft long delivers water to a powerhouse at a point 1500 ft lower in elevation than the water surface at intake. When the flow is 450 cfs, what is the horsepower delivered to the plant?

8.28.2 A 10-in-diameter pipeline ($f = 0.020$) is 3 miles long (Fig. X8.28.2). When pumping 4 cfs of water through it, with a total actual lift of 25 ft, how much power is required? The pump efficiency is 72%.

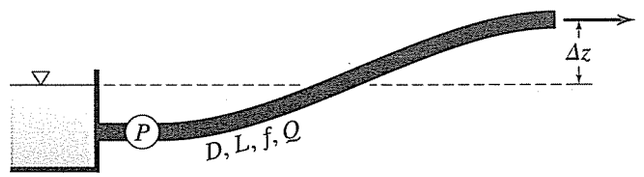


Figure X8.28.2

8.28.3 A 250-mm-diameter pipeline ($f = 0.025$) is 4.7 km long (Fig. X8.28.2). When pumping 100 L/s of water through it, with a total actual lift of 10.5 m, how much power is required? The pump efficiency is 75%.

8.28.4 If in Sample Prob. 8.13 the vapor pressure of the liquid is 1.9 psia and the atmospheric pressure is 14.5 psia, what is the maximum theoretical flow rate?

8.28.5 In Fig. 8.26 assume the pipe diameter is 12 in, $f = 0.021$, $BC = 200$ ft, and $\Delta z = 120$ ft. The entrance to the pipe at the intake is flush with the wall, and discharge losses are negligible. (*a*) If $Q = 8$ cfs of water, what head does it supply to the turbine? (*b*) What power does the turbine deliver if its efficiency is 75%?

8.28.6 In Fig. 8.26 assume the pipe diameter is 300 mm, $f = 0.021$, $BC = 60$ m, and $\Delta z = 36.5$ m. The entrance to the pipe at the intake is flush with the wall, and discharge losses are negligible. (*a*) If $Q = 225$ L/s of water, what head does it supply to the turbine? (*b*) What power does the turbine deliver if its efficiency is 75%?

8.28.7 A 12-in-diameter pipe 9200 ft long ($f = 0.024$) discharges freely into the air at an elevation 18 ft below the surface of the water at intake (Fig. X8.28.7). It is necessary to double the flow by inserting a pump. If the efficiency of the pump is 73%, how much power will be required?

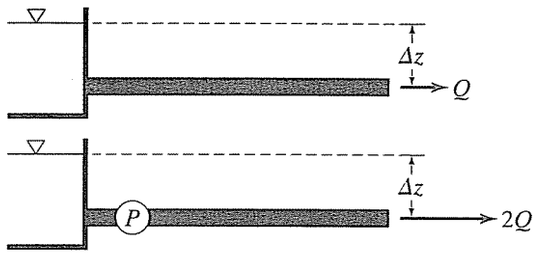


Figure X8.28.7

8.28.8 A 300-mm-diameter pipe 3400 m long ($f = 0.022$) discharges freely into the air at an elevation 5.6 m below the surface of the water at intake (Fig. X8.28.7). It is necessary to double the flow by inserting a pump. If the efficiency of the pump is 76%, how much power will be required?

8.28.9 Refer to Fig. 8.25. When the pump is delivering 1.2 cfs of water, a pressure gage at *D* reads 25 psi, while a vacuum gage at *C* reads 10 inHg. The pressure gage at *D* is 2 ft higher than the vacuum gage at *C*. The pipe diameters are 4 in for the suction pipe and 3 in for the discharge pipe. Find the power delivered to the water.

8.28.10 Refer to Fig. 8.25. When the pump is delivering 35 L/s of water, a pressure gage at *D* reads 175 kPa, while a vacuum gage at *C* reads 250 mmHg. The pressure gage at *D* is 600 mm higher than the vacuum gage at *C*. The pipe diameters are 100 mm for the suction pipe and 75 mm for the discharge pipe. Find the power delivered to the water.

8.29 BRANCHING PIPES

For convenience, let us consider three pipes connected to three reservoirs as in Fig. 8.27 and connected together or branching at the common junction point *J*. Actually, we can consider that any of the pipes is connected to some other destination than a reservoir by simply replacing the reservoir with a piezometer tube in which the water level is the same as the reservoir surface. We shall suppose that all the pipes are sufficiently long that we can neglect minor losses and velocity heads, so $h_L = h_f$, which we shall designate as h.

We name the pipes and flows and corresponding friction losses as in the diagram. The continuity and energy equations require that the flow entering the

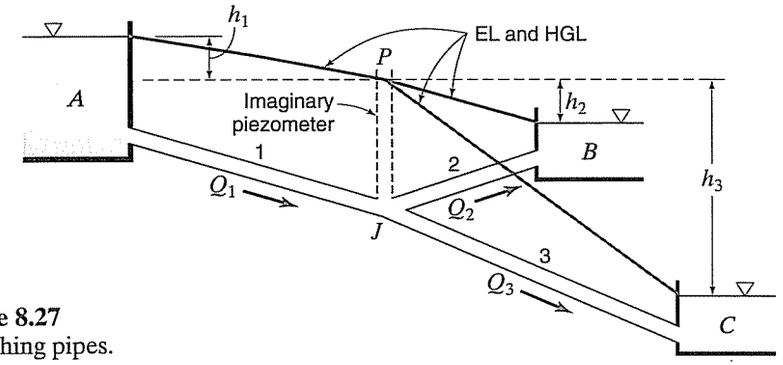


Figure 8.27
Branching pipes.

junction equals the flow leaving it, and that the pressure head at *J* (which we shall represent schematically by the imaginary open piezometer tube shown, with water at elevation *P*) is common to all pipes.

As there are no pumps, the elevation of *P* must lie between the surfaces of reservoirs *A* and *C*. If *P* is level with the surface of reservoir *B* then h_2 and Q_2 are both zero. If *P* is above the surface of reservoir *B* then water must flow into *B* and $Q_1 = Q_2 + Q_3$. If *P* is below the surface of reservoir *B* then the flow must be out of *B* and $Q_1 + Q_2 = Q_3$. So for the situation shown in Fig. 8.27 we have the following governing conditions:

1. $Q_1 = Q_2 + Q_3$.
2. Elevation of *P* is common to all three pipes.

The diagram suggests several different problems or cases, three of which we will discuss below using different methods of solution.

Rigorous Solutions

When we know the pipe wall material, we can estimate its *e* value (Table 8.1), and we know that the friction factor *f* varies with the *e/D* of the pipe and the Reynolds number of the flow. Because we are not considering minor losses, we can use the equations and methods of Secs. 8.12–8.17. In particular, using only a basic scientific calculator, we can solve pipelines for *h* (Type 1 problems) using the Haaland equation (8.52) with Eq. (8.13); we can solve for *V* or *Q* (Type 2 problems) using Eq. (8.56); and more rarely, we can solve for *D* (Type 3 problems) using Eqs. (8.57)–(8.62). We prefer these equations because they avoid trial and error, which can become quite confusing when combined with other trial-and-error techniques needed to solve for branching flow.

The following three cases illustrate some of the "manual" trial-and-error methods used to solve the different types of problems that can occur. In each case all the required pipe data (lengths, diameters, and materials for *e* values) are known.

Case 1. Find the flow to or from two reservoirs, and the surface elevation of one of these, given the other flow and two elevations and all the pipe data.

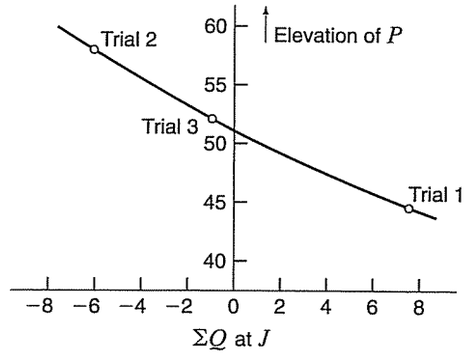


Figure 8.28

We can solve this problem directly, without trial and error. Suppose that Q_1 and the elevations of A and B are given. Then we can directly determine the head loss h_1 (Type 1), using Fig. 8.11 or Eq. (8.52) to find the proper value of f. Knowing h_1 fixes P, so now we can easily obtain h_2. Knowing h_2 enables us to directly determine the flow in pipe 2 using the Type 2 equation (8.56). Condition 1 (continuity at junction J) then determines Q_3, which in turn enables us to directly find h_3 (Type 1), in the same way as for line 1. Finally, P and h_3 define the required surface elevation of C.

Case 2. Find the flow to or from two reservoirs, and the surface elevation of the third, given the other flow and two elevations and all the pipe data.

Let us suppose that Q_2 and the surface elevations of A and C are given. Then we know $h_1 + h_3 = \Delta h_{13}$, say. We may use various solution approaches;[26] we shall discuss a more convenient one. In this, we assume the elevation of P, which yields values for h_1 and h_3, and so Q_1 and Q_3 via Eq. (8.56). If these do not satisfy the discharge relation at J then we must adjust P until they do. To help us converge on the correct elevation of P, we can plot the results of each assumption on a graph like Fig. 8.28. For ΣQ at J, inflows to J are taken as positive and outflows as negative. Two or three points, with one fairly close to the vertical axis, determine a (near-straight) curve that intersects that axis at the equilibrium level of P, where $\Sigma Q = 0$, as required. Last, we can determine h_2 from Q_2 and Eqs. (8.52) and (8.13), and find the required surface elevation of B.

Case 3. Find the flow in each pipe, given the surface elevation of all three reservoirs and all the pipe data.

This is the ***classic three-reservoir problem,*** and it differs from the foregoing cases in that it is not immediately evident whether the flow is *into* or *out of* reservoir B. We can readily determine this direction by first assuming no flow in pipe 2; that is, assume the piezometer level P at the elevation of the surface of B. The head losses h_1 and h_3 then determine the flows Q_1 and Q_3 via Eq. (8.56). If

[26] Other approaches include (a) assuming distributions of the flows Q_1 and Q_3, knowing that $Q_1 - Q_3 = Q_2$, and (b) by substituting for the hs in $h_1 + h_3 = \Delta h_{13}$ using Eq. (8.13) with V_3 written in terms of Q_3, and V_1 written in terms of $Q_2 + Q_3$, and successively solving the resulting quadratic equation for Q_3 while converging on f values.

$Q_1 > Q_3$, then we must raise P to satisfy continuity at J, causing water to flow *into* reservoir B, and we shall have $Q_1 = Q_2 + Q_3$; if $Q_1 < Q_3$, then we must lower P to satisfy continuity at J, causing water to flow *out* of reservoir B, and we shall have $Q_1 + Q_2 = Q_3$. From here on the solution proceeds by adjusting P as for Case 2.

Note: For any of the above three cases, we can avoid manual trial and error by setting up the governing equations and solving them simultaneously using equation solving software like Mathcad or Excel (see Appendixes C.2–C.3 and D.2–D.3). There will be the usual four governing equations for each line, a flow continuity equation for the junction, and, depending on the case addressed, one or two equations relating the various head losses. With so many unknowns to solve for, the success of the procedure becomes more sensitive to the guessed values, and we may have to try different guesses. The great advantage of this approach is that it is so straightforward. It is illustrated in part (b) of the following sample problem.

SAMPLE PROBLEM 8.14 Given that, in Fig. 8.27, pipe 1 is 6000 ft of 15 in diameter, pipe 2 is 1500 ft of 10 in diameter, and pipe 3 is 4500 ft of 8 in diameter, all asphalt-dipped cast iron. The elevations of the water surfaces in reservoirs A and C are 250 ft and 160 ft, respectively, and the discharge Q_2 of 60°F water into reservoir B is 3.3 cfs. Find the surface elevation of reservoir B: (a) using only a basic scientific calculator; (b) using equation solving software.

Solution

This is a Case 2 problem.

Table A.1 for water at 60°F: $\nu = 12.17 \times 10^{-6}$ ft^2/sec.

Pipe:	1	2	3
L, ft	6000	1500	4500
D, ft	1.25	10/12	8/12
e, ft (Table 8.1)	0.0004	0.0004	0.0004
L/D	4800	1800	6750
$A = \pi D^2/4$, ft^2	1.227	0.545	0.349
e/D	0.00032	0.00048	0.0006

(a) Find the elevation of P by trial and error.

Elevation of P lies between 160 and 250 ft. Calculate V from Eq. (8.56) assuming turbulent flow. Trials:

Elev. P	h_1	h_3	V_1	V_3	$\mathbf{R}_1$	$\mathbf{R}_2$	Q_1	Q_3	ΣQ	Move P?
200	50	40	6.444	4.481	662,000	245,000	7.907	1.564	+3.04	Up
230	20	70	4.013	5.984	412,000	328,000	4.925	2.088	−0.463	Down

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Interpolation (Fig. 8.28): (230 − Elev. P)/(230 − 200) = 0.463/(0.463 + 3.04); Elev. P = 226.03.

Elev. P	h_1	h_3	V_1	V_3	$\mathbf{R}_1$	$\mathbf{R}_2$	Q_1	Q_3	ΣQ	Move P?
226	24	66	4.412	5.805	453,000	318,000	5.414	2.026	+0.088	Up

Interpolation (Fig. 8.28): (230 − Elev. P)/(230 − 226) = 0.463/(0.463 + 0.088); Elev. P = 226.64.

Close enough! *Note:* These adjustments are very suitable for making on a spreadsheet (Appendix C.2).

$$V_2 = \frac{Q_2}{A_2} = \frac{3.3}{0.545} = 6.055 \text{ fps}; \quad \mathbf{R}_2 = \frac{D_2 V_2}{\nu} = 416{,}500$$

All three **R** values are turbulent, so the use of Eq. (8.56) and these results are valid.

Eq. (8.52): $f_2 = 0.01761$; Eq. (8.13): $h_2 = 18.05$ ft

$$\text{Elev. } B = \text{Elev. } P - h_2 = 226.64 - 18.05 = 208.59 \text{ ft} \qquad \textit{ANS}$$

(*b*) Using equation solving software such as Excel or Mathcad (Appendixes C.2–C.3), we note that there are 14 governing equations. Two of these are

$$h_1 + h_3 = 250 - 160 = 90, \quad Q_1 = Q_2 + Q_3$$

The remaining 12 equations are the four governing equations (Sec. 8.14) for each of the three pipes.

The complete output for the Excel and Mathcad solutions is provided in Appendixes D.2 and D.3, respectively.

The solver provides values for the 14 unknowns (three values each for f, V, **R**, h, and two values for Q) from which it calculates

$$\text{Elev. } B = 208.53 \text{ ft} \qquad \textit{ANS}$$

Note: Observe how simple and clear the Mathcad solution is.

SAMPLE PROBLEM 8.15 With the sizes, lengths, and material of pipes given in Sample Prob. 8.14, suppose that the surface elevations of reservoirs A, B, and C are 525 ft, 500 ft, and 430 ft, respectively. (*a*) Does water enter or leave reservoir B? (*b*) Find the flow rates of 60°F water in each pipe. Use only a basic scientific calculator.

Solution

This is a Case 3 problem. Find the elevation of P by trial and error.

Table A.1 for water at 60°F: $\nu = 12.17 \times 10^{-6}$ ft²/sec.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

The tabulated pipe data are the same as for Sample Prob. 8.14.

(*a*) *Trial 1.* First, try P at elevation of reservoir surface B = 500 ft:

Pipe:	1	2	3
h, ft	25	0	70
$\sqrt{2gDh/L}$, fps	0.579	0	0.817
V, fps (Eq. 8.56)	4.51	0	5.98
$Q = AV$, cfs	5.53	0	2.09

At J, ΣQ = inflow − outflow = 5.53 − 2.09 = 3.44 cfs. This must be zero, so P must be raised (to reduce Q_1 and increase Q_3); then water will flow *into* reservoir B. ***ANS***

(*b*) *Trial 2.* Raise P. 500 ft < Elev. P < 525 ft. Try P at elevation 510 ft:

h, ft	15	10	80
$\sqrt{2gDh/L}$, fps	0.449	0.598	0.874
V, fps (Eq. 8.56)	3.46	4.49	6.41
Check **R**	355,000	307,000	351,000
$Q = AV$, cfs	4.24	2.42	2.24

At J, $\Sigma Q = 4.24 - 2.42 - 2.24 = -0.42$ cfs. By interpolation (using Fig. 8.28),

$$\frac{510 - \text{Elev. } P}{510 - 500} = \frac{0.42}{0.42 + 3.44}; \quad \text{Elev. } P = 508.91 \text{ ft}$$

Trial 3. Try P at elevation 508.9 ft:

h, ft	16.1	8.9	78.9
$\sqrt{2gDh/L}$, fps	0.465	0.564	0.868
V, fps (Eq. 8.56)	3.59	4.19	6.36
Check **R**	339,000	287,000	348,000
$Q = AV$, cfs	4.40	2.28	2.22

ANS

At J, $\Sigma Q = 4.40 - 2.28 - 2.22 = -0.10$ cfs. Close enough!
Note: These repetitive adjustments are very suitable for making on a spreadsheet (Appendix C.2).

Nonrigorous Solutions

If the value of the friction factor (constant f, C_{HW}, or Manning's n) is *given* for each pipeline, we must use the nonrigorous head-loss equations of Sec. 8.19. Then, to solve the three cases just discussed, we follow exactly the same steps as for the rigorous solutions, but instead of using the equations of Secs. 8.12–8.17,

we simply use Eqs. (8.65)–(8.69). Of course, we must first determine the appropriate K and n value for each pipeline, and select accordingly from Eqs. (8.66)–(8.68). Notice, however, that we can easily solve Case 2 *directly* if $n = 2$ (Darcy-Weisbach or Manning) by writing the known elevation difference $\Delta h_{13} = h_1 + h_3 = K_1 Q_1^2 + K_3 Q_3^2 = K_1(Q_2 + Q_3)^2 + K_3 Q_3^2$, which is a quadratic equation in Q_3, the only unknown.

SAMPLE PROBLEM 8.16 In Fig. 8.27 pipe 1 is 300 mm diameter and 900 m long, pipe 2 is 200 mm diameter and 250 m long, and pipe 3 is 150 mm diameter and 700 m long. The Hazen-Williams coefficient for all pipes is 120. The surface elevations of reservoirs A, B, and C are 160 m, 150 m, and 120 m, respectively. (*a*) Does water enter or leave reservoir B? (*b*) Find the flow rate in each pipe. Use only a basic scientific calculator.

Solution

This is a Case 3 problem. Find the elevation of P by trial and error. Use the Hazen-Williams form (8.67) of Eq. (8.65) in SI units:

$$K = \frac{10.675L}{C_{HW}^{1.852} D^{4.87}}, \qquad n = 1.852$$

Pipe:	1	2	3
L, m	900	250	700
D, m	0.3	0.2	0.15
L/D	3000	1250	4667
C_{HW}	120	120	120
n	1.852	1.852	1.852
K	480	961	10924

(*a*) *Trial 1.* First, try P at elevation of reservoir surface $B = 150$ m:

h, m	10	0	30
Q, m³/s (Eq. 8.69)	0.1236	0	0.0414

At J, ΣQ = inflow − outflow = 0.1236 − 0.0414 = 0.0822 m³/s. This must be zero, so we must raise P (to reduce Q_1 and increase Q_3), then water will flow *into* reservoir B. ***ANS***

(*b*) *Trial 2.* Raise P. 150 m < Elev. P < 160 m, so try P at elevation 155 m:

h, m	5	5	35
Q, m³/s (Eq. 8.69)	0.0850	0.0584	0.0450

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

At J, $\Sigma Q = 0.0850 - 0.0584 - 0.0450 = -0.0184$ m³/s. This must be zero, so we must lower P. By interpolation (Fig. 8.28),

$$\frac{155 - \text{Elev. } P}{155 - 150} = \frac{0.0184}{0.0184 + 0.0822}; \qquad \text{Elev. } P = 154.09 \text{ m}$$

Trial 3. Try P at elevation 154 m:

h, m	6	4	34	
Q, m³/s (Eq. 8.69)	0.0938	0.0518	0.0443	***ANS***
V, m/s (Eq. 4.7)	1.327	1.649	2.51	

At J, $\Sigma Q = -0.0023$ m³/s. This is close enough.

All Vs are < 3 m/s (Sec. 8.18), so these solutions using Eq. (8.67) are valid.

Note: These adjustments are very suitable for making on a spreadsheet (Appendix C.2).

8.30 PIPES IN SERIES

The discussion in Sec. 8.27 addressed the case of a single pipe of constant diameter. If a pipeline is made up of lengths of *different* diameters, as in Fig. 8.29, conditions must satisfy the continuity and energy equations, namely:

$$Q = Q_1 = Q_2 = Q_3 = \cdots \tag{8.86}$$

$$h_L = h_{L1} + h_{L2} + h_{L3} + \cdots \tag{8.87}$$

If we are given the rate of discharge Q, the problem is straightforward. We may find the head loss directly by adding the contributions from the various sections, as in Eq. (8.87); as discussed in Sec. 8.14, $h_L = \Delta Q$. If we are given empirical coefficients or constant f values, we can do the same thing using Eq. (8.65) and the appropriate values of K and n selected from (8.66)–(8.68). If, however, we are given the pipe material or e, the result will be more accurate, because it uses the rigorous Darcy-Weisbach approach. Then we use Eq. (8.13) to find the individual head loss contributions after finding e/D, V, **R**, and f for each pipe.

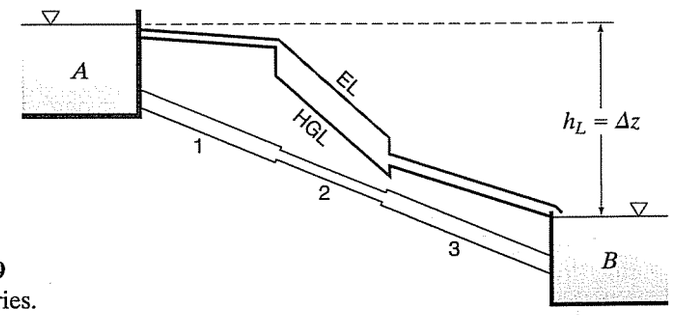


Figure 8.29
Pipes in series.

If we are given the total head loss h_L and want to find the flow, the problem is a little more involved.

Using the nonrigorous equations, we again substitute Eq. (8.65) into Eq. (8.87), to get

$$h_L = h_f = K_1 Q_1^n + K_2 Q_2^n + K_3 Q_3^n + \cdots$$

But since all the Qs are equal from Eq. (8.86), this becomes

$$h_L = (K_1 + K_2 + K_3 + \cdots)Q^n = (\sum K)Q^n \tag{8.88}$$

So, knowing the pipe information and which nonrigorous equation we must use, we can solve for Q. Last, we must check that all velocities are in the required ranges for the empirical equations to be valid (Sec. 8.18).

If we wish to use the more accurate, rigorous Darcy-Weisbach approach to find Q, we must note that in Eq. (8.88) each K has now become a function of a different f. The preferred manual method of solution is similar to that just discussed, and we call it the ***equivalent-velocity-head method.*** Substituting from Eq. (8.13) into Eq. (8.87) and including minor losses if we wish (usually if $L/D < 1000$),

$$h_L = \left(f_1\frac{L_1}{D_1} + \sum k_1\right)\frac{V_1^2}{2g} + \left(f_2\frac{L_2}{D_2} + \sum k_2\right)\frac{V_2^2}{2g} + \cdots$$

Using continuity Eq. (4.17), we know $D_1^2V_1 = D_2^2V_2 = D_3^2V_3$, etc., from which we can express all the velocities in terms of one chosen velocity. So, by assuming reasonable values for each f (e.g., from Eq. (8.54) or Fig. 8.11), for any pipeline, however complex, we can write the total head loss as

$$h_L = K\frac{V^2}{2g} \tag{8.89}$$

where V is the chosen velocity. We can solve this equation for the chosen V, and so can obtain the V and $\mathbf{R}$ and f values for each pipe. For better accuracy, we should replace the assumed values of f by the values just obtained, and obtain an improved solution. When the f values converge V is correct, and we can calculate Q.

SAMPLE PROBLEM 8.17 Suppose in Fig. 8.29 the pipes 1, 2, and 3 are 300 m of 300 mm diameter, 150 m of 200 mm diameter, and 250 m of 250 mm diameter, respectively, of new cast iron and are conveying 15°C water. If $\Delta z = 10$ m, find the rate of flow from A to B using only a basic scientific calculator. Neglect minor losses.

Solution

Table 8.1 for cast-iron pipe: $e = 0.25$ mm $= 0.00025$ m.

Table A.1 for water at 15°C: $\nu = 1.139 \times 10^{-6}$ m^2/s.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Pipe:	1	2	3
L, m	300	150	250
D, m	0.3	0.2	0.25
e/D	0.000833	0.00125	0.00100
f_{min} (Fig. 8.11)	0.019	0.021	0.020

Assuming these friction factor values,

$$\Delta z = h_L = h_f = 10 = 0.019\left(\frac{300}{0.3}\right)\frac{V_1^2}{2g} + 0.021\left(\frac{150}{0.2}\right)\frac{V_2^2}{2g} + 0.020\left(\frac{250}{0.25}\right)\frac{V_3^2}{2g}$$

From continuity, $$\frac{V_2^2}{2g} = \frac{V_1^2}{2g}\left(\frac{D_1}{D_2}\right)^4 = \frac{V_1^2}{2g}\left(\frac{0.3}{0.2}\right)^4 = 5.06\frac{V_1^2}{2g}$$

Similarly, $$\frac{V_3^2}{2g} = 2.07\frac{V_1^2}{2g}$$

and thus $$10 = \frac{V_1^2}{2g}\left(0.019\frac{1000}{1} + 0.021\frac{750}{1}5.06 + 0.020\frac{1000}{1}2.07\right)$$

from which $$\frac{V_1^2}{2g} = 0.0713 \text{ m}$$

So $$V_1 = \sqrt{2(9.81 \text{ m/s}^2)(0.0713 \text{ m})} = 1.183 \text{ m/s}$$

The corresponding values of $\mathbf{R}$ are 0.31×10^6, 0.47×10^6, and 0.37×10^6; the corresponding friction factors are only slightly different from those we originally assumed, since the flow occurs at Reynolds numbers very close to fully-rough-pipe flow. So

$$Q = A_1V_1 = \tfrac{1}{4}\pi(0.30)^2 1.183 = 0.0836 \text{ m}^3\text{/s} \qquad \textbf{\textit{ANS}}$$

Note: We would have obtained greater accuracy if we had adjusted the friction factors to match the pipe-friction chart more closely (further trials) or calculated them by Eq. (8.52), and if we had included minor losses. In that case $Q = 0.0821$ m^3/s.

We can avoid manual iteration for f by solving simultaneous equations using equation solving software like Mathcad or Excel (Appendixes C.2–C.3 and D.2–D.3). There are the usual four equations for each pipe (Sec. 8.14), plus Eq. (8.87); if necessary, we may easily account for minor losses by using head loss equations with the form of Eq. (8.80). For the three pipes of Fig. 8.29, for example, there are therefore 13 simultaneous equations, which we may solve in the usual manner for 13 unknowns (see Sample Prob. 8.14*b* and Appendixes D.2–D.3). The unknowns are either the flow rate or the total head loss, and three values each of h_L, V, $\mathbf{R}$, and f.

8.31 PIPES IN PARALLEL

In the case of flow through two or more parallel pipes, as in Fig. 8.30, conditions must satisfy the following continuity and energy equations:

$$Q = Q_1 + Q_2 + Q_3 \tag{8.90}$$

$$h_L = h_{L1} = h_{L2} = h_{L3} \tag{8.91}$$

because the pressures at A and B are common to all pipes. Problems may be posed in various ways.

If the head loss h_L is given, the problem is straightforward. We can find the total discharge directly by adding the contributions from the various pipes, as in Eq. (8.90). If we are given empirical roughness coefficients or constant f values, we can do the same thing using Eq. (8.69) and the appropriate values of K and n selected from (8.66)–(8.68). If, however, we are given the pipe material or e, the result will be more accurate, because it uses the rigorous Darcy-Weisbach approach. Then we have an independent Type 2 problem for each pipe (see Secs. 8.14–8.17), which we can solve directly by Eq. (8.56) for example.

If we are given the total flow Q and want to find the head loss and individual flows, the problem is a little more involved.

Using the empirical equations, and neglecting minor losses, we again substitute Eq. (8.69) into Eq. (8.90), to get

$$Q = \left(\frac{h_{f1}}{K_1}\right)^{1/n} + \left(\frac{h_{f2}}{K_2}\right)^{1/n} + \left(\frac{h_{f3}}{K_3}\right)^{1/n} + \cdots$$

But since all the h_fs ($= h_L$s) are equal from Eq. (8.91), this becomes

$$Q = (h_f)^{1/n}\left[\left(\frac{1}{K_1}\right)^{1/n} + \left(\frac{1}{K_2}\right)^{1/n} + \left(\frac{1}{K_3}\right)^{1/n} + \cdots\right] = (h_f)^{1/n}\sum\left(\frac{1}{K}\right)^{1/n} \tag{8.92}$$

So, knowing the pipe information and the empirical equation we must use, we can solve for h_f. We can then find the individual flows using Eq. (8.69). Last, we must check that all velocities are in the required ranges for the empirical equations to be valid (Sec. 8.18).

If we wish to use the rigorous Darcy-Weisbach approach to find h_L and the individual Qs, we must note that in Eq. (8.92) each K has now become a

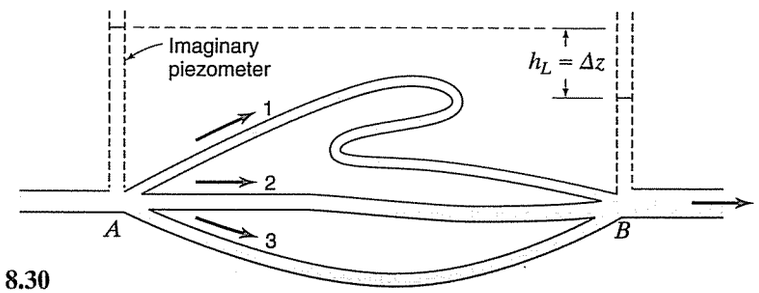


Figure 8.30
Pipes in parallel.

function of a different f. The preferred manual method of solution is similar to the preceding. Writing Eq. (8.13) for each line, including minor losses if we wish,

$$h_L = \left(f\frac{L}{D} + \Sigma k\right)\frac{V^2}{2g}$$

where $\Sigma k \Sigma k$ is the sum of the minor-loss coefficients, which we can usually neglect if the pipe is longer than 1000 diameters. Solving for V and then Q, we obtain the following for pipe 1:

$$Q_1 = A_1V_1 = A_1\sqrt{\frac{2gh_L}{f_1(L_1/D_1) + \Sigma k_1}} = C_1\sqrt{h_L} \tag{8.93}$$

where C_1 is constant for the given pipe, except for the change in f with Reynolds number. We can similarly express the flows in the other pipes, using assumed reasonable values of f from Fig. 8.11 or Eq. (8.54). Finally, Eq. (8.90) becomes

$$Q = C_1\sqrt{h_L} + C_2\sqrt{h_L} + C_3\sqrt{h_L} = (C_1 + C_2 + C_3)\sqrt{h_L} \tag{8.94}$$

This enables us to find a first estimate of h_L and the distribution of flows and velocities in the pipes. Using these, we can next make improvements to the values of f, if indicated, and if necessary repeat them, until we finally obtain a correct determination of h_L and the distribution of flows.

If we use the turbulent-flow equation (8.51) or (8.52) to obtain f, we must remember to confirm that the Reynolds number is in the turbulent range. We can precheck the likelihood of laminar flow occurring in any of the pipes by calculating an "average" flow velocity from the total flow divided by the total area of all the pipes, and using this velocity to obtain an indicator **R** for each pipe.



SAMPLE PROBLEM 8.18 Three pipes A, B, and C are interconnected as in Fig. S8.18. The pipe characteristics are as follows:

Pipe	D (in)	L (ft)	f
A	6	2000	0.020
B	4	1600	0.032
C	8	4000	0.024

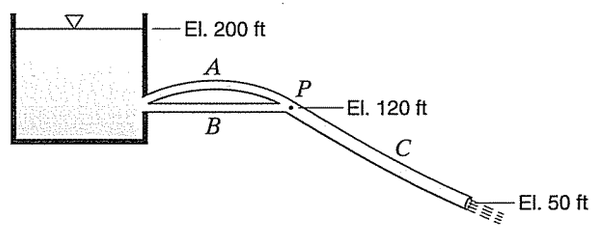


Figure S8.18

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Find the rate at which water will flow in each pipe. Find also the pressure at point P. All pipe lengths are much greater than 1000 diameters, so neglect minor losses.

Solution

Eq. (5.28): $$0 + 200 + 0 - 0.020\frac{2000}{6/12}\frac{V_A^2}{2g} - 0.024\frac{4000}{8/12}\frac{V_C^2}{2g} = 0 + 50 + \frac{V_C^2}{2g}$$

i.e., $$150 = 80\frac{V_A^2}{2g} + 145\frac{V_C^2}{2g} \qquad (1)$$

Continuity: $$6^2V_A + 4^2V_B = 8^2V_C$$

i.e., $$36V_A + 16V_B = 64V_C \qquad (2)$$

Also, $$h_{fA} = h_{fB} = 0.020\frac{2000}{6/12}\frac{V_A^2}{2g} = 0.032\frac{1600}{4/12}\frac{V_B^2}{2g}$$

i.e., $$80V_A^2 = 153.6V_B^2, \quad V_B = 0.722V_A$$

Substituting into (2):

$$36V_A + 16(0.722V_A) = 64V_C$$

$$47.5V_A = 64V_C, \quad V_A = 1.346V_C$$

Substituting into (1):

$$150 = 80\frac{(1.346V_C)^2}{2g} + 145\frac{V_C^2}{2g} = 289.9\frac{V_C^2}{2g}$$

$$V_C^2 = 2(32.2)150/289.9 = 33.3$$

$$V_C = 5.77 \text{ fps}, \quad Q_C = A_CV_C = (0.349)5.77 = 2.01 \text{ cfs} \qquad \textbf{\textit{ANS}}$$

$$V_A = 1.346V_C = 7.77 \text{ fps}, \quad Q_A = (0.1963)7.77 = 1.526 \text{ cfs} \qquad \textbf{\textit{ANS}}$$

$$V_B = 0.722V_A = 5.61 \text{ fps}$$

$$Q_B = A_BV_B = (0.0873)5.61 = 0.489 \text{ cfs} \qquad \textbf{\textit{ANS}}$$

As a check, note that $Q_A + Q_B = Q_C$ is satisfied.

To find the pressure at P:

Eq. (5.28): $$0 + 200 + 0 - 80\frac{V_A^2}{2g} = \frac{p_P}{\gamma} + 120$$

$$\frac{p_P}{\gamma} = 80 - 80\frac{(7.77)^2}{2(32.2)} = 5.01 \text{ ft}$$

Check: $$120 + \frac{p_p}{\gamma} - 144\frac{V_C^2}{2g} = 50 + \frac{V_C^2}{2g}$$

$$\frac{p_P}{\gamma} = 145\frac{(5.77)^2}{2(32.2)} - 70 = 5.01 \text{ ft}$$

So $p_P/\gamma = 5.01$ ft and $p_P = (62.4/144)5.01 = 2.17$ psi. ***ANS***

Note: In this example we were given the values of f for each pipe as known. Actually f depends on $\mathbf{R}$ [Fig. 8.11 or Eq. (8.52)]. Usually we know or assume the absolute roughness e of each pipe, and achieve an accurate solution by trial and error until the fs and $\mathbf{R}$s for each pipe have converged.

We can avoid manual iteration for f by solving simultaneous equations using equation solving software as in Mathcad or Excel (Appendixes C.2–C.3 and D.2–D.3). There are the usual four equations for each pipe (Sec. 8.14), plus Eq. (8.90); if necessary, we may easily account for minor losses by using head loss equations with the form of Eq. (8.80). For the three pipes of Fig. 8.30, for example, there are therefore 13 simultaneous equations, which we may solve in the usual manner for 13 unknowns (see Sample Prob. 8.14*b* and Appendixes D.2–D.3). The unknowns are either the head loss or the total flow rate, and three values each of Q, V, $\mathbf{R}$, and f.

It is instructive to compare the solution methods for pipes in parallel with those for pipes in series. The role of the head loss in one case becomes that of the discharge rate in the other, and vice versa. Students should be already familiar with this situation from the elementary theory of dc circuits. The flow corresponds to the electrical current, the head loss to the voltage drop, and the frictional resistance to the ohmic resistance. The outstanding deficiency in this analogy occurs in the variation of potential drop with flow, which is with the first power in the electrical case ($V = IR$) and with the second power in the hydraulic case ($h_L \propto V^2 \propto Q^2$) for fully developed turbulent flow.

8.32 PIPE NETWORKS

In municipal distribution systems, pipes are frequently interconnected so that the flow to a given outlet may come by several different paths, as in Fig. 8.31. As a result, we often cannot tell by inspection which way the flow travels, as in pipe BE. Nevertheless, the flow in any network, however complicated, must satisfy the basic relations of continuity and energy as follows:

1. The flow into any junction must equal the flow out of it.
2. The flow in each pipe must satisfy the pipe-friction laws for flow in a single pipe.
3. The algebraic sum of the head losses around any closed loop must be zero.

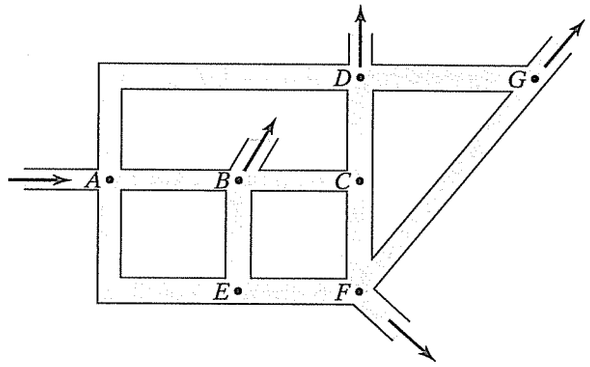


Figure 8.31
Pipe network.

Most pipe networks are too complicated to solve analytically by hand using rigorous (variable f) equations, as was possible in the simpler cases of parallel pipes (Sec. 8.31). Nowadays they are readily solved by specially developed computer programs (Appendix C.5). However, in many cases we cannot predict the capacity requirements of water distribution systems with high precision, and flows in them vary considerably throughout the day, so high accuracy in calculating their flows is not important. As a result, the use of nonrigorous equations (Secs. 8.18–8.19) are very acceptable for this purpose. The method of successive approximations, due to Cross,[27] is such a method that was popular before the advent of computers. We will review it here to help students understand the fluid mechanics of pipe networks and evaluate computer-generated solutions. It consists of the following elements, in order:

Step 1: By careful inspection assume the most reasonable distribution of flows that satisfies condition 1.

Step 2: Write condition 2 for each pipe in the form

$$h_L = KQ^n \tag{8.95}$$

where K and n are constants for each pipe as described in Sec. 8.19. If minor losses are important include them as in Eq. (8.93), which yields $K = 1/C^2$ and $n = 2$ for constant f. We may include minor losses within any pipe or loop, but must neglect them at the junction points.

Step 3: To investigate condition 3, compute the algebraic sum of the head losses around each elementary loop, $\sum h_L = \sum KQ^n$. Consider losses from clockwise flows as positive, counterclockwise negative. Only by good luck will these add up to zero on the first trial.

Step 4: Adjust the flow in each loop by a correction ΔQ to balance the head in that loop and give $\sum KQ^n = 0$. The heart of this method lies in the following determination of ΔQ. For any pipe, we may write

$$Q = Q_0 + \Delta Q$$

where Q is the correct discharge and Q_0 is the assumed discharge. Then, for each pipe,

$$h_L = KQ^n = K(Q_0 + \Delta Q)^n = K(Q_0^n + nQ_0^{n-1}\Delta Q + \cdots)$$

If ΔQ is small compared with Q_0, we may neglect the terms of the binomial series after the second one, so that

$$h_L = KQ_0^n + \Delta QKnQ_0^{n-1}$$

For a loop, $\sum h_L = \sum KQ^n = 0$, so because ΔQ is the same for all pipes in that loop,

$$\sum KQ_0^n + \Delta Q \sum KnQ_0^{n-1} = 0$$

As we must sum the corrections of head loss in all pipes *arithmetically* (treating all terms as positive), we may solve this equation for ΔQ,

$$\Delta Q = \frac{-\sum KQ_0|Q_0^{n-1}|}{n\sum|KQ_0^{n-1}|} = \frac{-\sum h_L}{n\sum|h_L/Q_0|} \tag{8.96}$$

since, from Eq. (8.95), $h_L/Q = KQ^{n-1}$. We emphasize again that we must sum the numerator of Eq. (8.96) *algebraically,* with due account of each sign, while we must sum the denominator *arithmetically.* Note that the $Q_0|Q_0^{n-1}|$ in the numerator gives this quantity the same sign as the head loss. The negative sign in Eq. (8.96) indicates that when there is an excess of head loss around a loop in the clockwise direction, we must subtract the ΔQ from clockwise Q_0 values and add it to counterclockwise ones. The reverse is true if there is a deficiency of head loss around a loop in the clockwise direction.

Step 5: After we have given each loop a first correction, the losses will still not balance, because of the interaction of one loop upon another (pipes which are common to two loops receive two independent corrections, one for each loop). So we repeat the procedure, arriving at a second correction, and so on, until the corrections become negligible.

We may use either form of Eq. (8.96) to find ΔQ. As values of K appear in both the numerator and denominator of the first form, we can use values proportional to the actual K to find the distribution. The second form is more convenient for use with pipe-friction diagrams for water pipes.

An attractive feature of this approximation method is that errors in computation have the same effect as errors in judgment and the process eventually corrects them.

[27] H. Cross, Analysis of Flow in Networks of Conduits or Conductors, *Univ. Ill. Eng. Expt. Sta. Bull.* 286, 1936.

SAMPLE PROBLEM 8.19 If the flow into and out of a two-loop pipe system are as shown in Fig. S8.19, determine the flow in each pipe using only a basic scientific calculator. The K values for each pipe were calculated from the pipe and minor loss characteristics and from an assumed value of f, and $n = 2$.

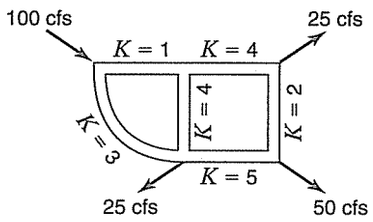


Figure S8.19

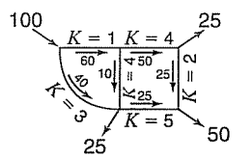
Solution

As a first step, assume a flow in each pipe such that continuity holds at all junctions. Take clockwise flows as positive. Calculate ΔQ for each loop, make corrections to the assumed Qs, and repeat several times until the ΔQs are quite small.

Left loop

$h_L = KQ_0^n$	$n\lvert KQ_0^{n-1}\rvert$
$1 \times 60^2 = 3{,}600$	$1 \times 2 \times 60 = 120$
$4 \times 10^2 = 400$	$4 \times 2 \times 10 = 80$
$3 \times 40^2 = 4{,}800$	$3 \times 2 \times 40 = 240$
800	440

$$\Delta Q_1 = \frac{-(-800)}{440} \approx 2$$

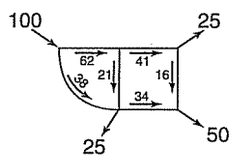


First approximation

Right loop

$h_L = KQ_0^n$	$n\lvert KQ_0^{n-1}\rvert$
$4 \times 50^2 = 10{,}000$	$4 \times 2 \times 50 = 400$
$2 \times 25^2 = 1{,}250$	$2 \times 2 \times 25 = 100$
$4 \times 10^2 = 400$	$4 \times 2 \times 10 = 80$
$5 \times 25^2 = 3{,}125$	$5 \times 2 \times 25 = 250$
$7{,}725$	830

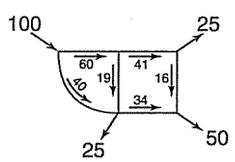
$$\Delta Q_1 = \frac{-(+7725)}{830} \approx 9$$



After first correction

$h_L = KQ_0^n$	$n\lvert KQ_0^{n-1}\rvert$
$1 \times 62^2 = 3{,}844$	$1 \times 2 \times 62 = 124$
$4 \times 21^2 = 1{,}764$	$4 \times 2 \times 21 = 168$
$3 \times 38^2 = 4{,}332$	$3 \times 2 \times 38 = 228$
$1{,}276$	520

$$\Delta Q_2 = \frac{-(+1276)}{520} \approx 2$$



After second correction

$h_L = KQ_0^n$	$n\lvert KQ_0^{n-1}\rvert$
$4 \times 41^2 = 6{,}724$	$4 \times 2 \times 41 = 328$
$2 \times 16^2 = 512$	$2 \times 2 \times 16 = 64$
$4 \times 21^2 = 1{,}764$	$4 \times 2 \times 21 = 168$
$5 \times 34^2 = 5{,}780$	$5 \times 2 \times 34 = 340$
308	900

$$\Delta Q_2 = \frac{-(-308)}{900} \approx 0$$

Further corrections can be made if greater accuracy is desired.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

As noted earlier, varying demand rates usually make high solution accuracy unnecessary with pipe networks. However, if high manual accuracy is required for some reason, we can first solve the problem in a similar manner to the preceding example using the Darcy-Weisbach K in Eq. (8.65) and constant f values. Then we can use the resulting flows to adjust the f and K values, and repeat the process (more than once if necessary) to refine the answers. The value of such refinement is questionable, not only because of uncertainties in the demand flows, but also because of uncertainties in the e values (pipe roughness) we must use (see Sec. 8.15). Usually when we adjust f values they change by only a few percent, but we can see in Fig. 8.11 that for smoother pipe it is possible for them to change by as much as a factor of five.

We can solve simple networks without approximation and manual iteration by solving simultaneous equations using equation solving software like that in Mathcad and Excel (Appendixes C.2–C.3 and D.2–D.3). For networks containing i pipes, $5i$ equations are required if using the Darcy-Weisbach equation with variable f, and $2i$ equations are required if using the simplified Eq. (8.95) with constant friction factors. These required equations include (a) the usual (condition 2) flow equations for each pipe (four or one per pipe, depending on the equations used); (b) flow continuity equations (condition 1) at all but one of the j nodes (as these imply continuity at the last node); (c) equations for the sum of the head losses around $i - j + 1$ loops (condition 3). The unknowns we want to find for each pipe are h_L, Q, V, $\mathbf{R}$, and f using the Darcy-Weisbach equation, or only h_L and Q using Eq. (8.95).

The pipe-network problem lends itself well to solution by use of a digital computer. Programming takes time and care, but once set up, there is great flexibility and it can save many hours of repetitive labor. Many software packages are now available to simulate water distribution networks; see Appendix C.

8.33 FURTHER TOPICS IN PIPE FLOW

The basics of steady incompressible flow in pressure conduits discussed in this chapter are just an introduction to many more advanced subjects involving flow in conduits. Flow through submerged culverts is a special case discussed in Sec. 10.23. Where sewers must dip to flow under obstacles like streams, they flow full and are known as ***inverted siphons*** or ***sag pipes;*** turbulence, and therefore velocities, must remain adequate when flow rates vary in order to prevent suspended solids from accumulating at such low points. Later in this text, we discusss ***unsteady flow*** in pipes (Chap. 12), involving both moderate rates of change and the very rapid changes associated with ***water hammer.*** We often investigate or simulate such flows using ***numerical methods,*** usually on a computer. In Chap. 13 we introduce the analysis of the flow of ***compressible fluids*** (gases), and in Chap. 11 we describe methods of ***flow measurement*** in pipes.

When we inject a different fluid into a pipeline, the main flow transports it (***advection***), and it also mixes and spreads through the main fluid by the processes of ionic and molecular ***diffusion,*** by which dissolved species move relatively slowly from areas of higher concentration to areas of lower concentration,

and by ***dispersion,*** also known as ***eddy diffusion*** or ***turbulent mixing.*** As these latter names suggest, turbulence causes dispersion, and as a result it is usually a far more rapid process than diffusion. Injection of easily detectable fluids into pipelines, such as salts (measured by conductivity) or dyes,[28] have been used to measure flow rates and velocities. If, for example, we continuously inject dye dissolved in water into a pipeline at a known flow rate and concentration, and give it sufficient opportunity to mix completely, the dye concentration downstream will indicate the relative magnitude of the pipe flow rate to the flow rate of the injected dye solution.

Pumping with pipelines, discussed in Sec. 8.28, has many engineering applications. We provide pump stations at low areas such as at underpasses that would otherwise flood during storms. Often, we provide multiple pumps to enable a more efficient response to varying inflows. In sanitary sewer systems pump stations become necessary if the slopes needed for gravity drainage require excessive excavation; sometimes, local topography requires pumping of sewage over a hill, through a ***force main.*** In such cases designers must take precautions to keep solids in suspension, and to prevent accumulation of dangerous gases. Water distribution systems, discussed in the previous section, must often supply areas of uneven topography, requiring their division into subareas or ***pressure zones*** served by separate pump stations which boost the pressures to the appropriate levels. Advanced software exists to analyze such conditions; see Appendix C.

To aid the dilution of sewage effluent, engineers often provide submarine and river ***outfalls*** with a ***diffuser section.*** (We discussed submarine discharge as a single jet in Sec. 5.15.) Diffusers have a number of outlets or ***ports*** along their length, so that the flow in that part of the main line or ***manifold*** decreases with distance. Manifolds also connect to navigation locks, to facilitate filling lock chambers as rapidly as possible while minimizing disturbance to moored vessels. The hydraulics of manifold flow is an advanced subject, particularly when applied to outfalls where port sizes are adjusted to maintain a uniform discharge along the length of the diffuser.

[28] Rhodamine and fluorescein dyes in water are detectable by a fluorometer at concentrations of one in 10^9.

PROBLEMS

8.1 Hydrogen at 30°F and atmospheric pressure has a kinematic viscosity of 0.0011 ft²/sec. Determine the minimum turbulent flow rate in pounds per second through a 2-in-diameter pipe. At this flow rate, what is the average velocity?

8.2 Air at 80°C and a pressure of approximately 1350 kPa abs flows in a 20-mm-diameter tube. What is the minimum turbulent flow rate? Express the answer in liters per second, newtons per second, and kilograms per second. At this flow rate what is the average velocity?

8.3 Two pipes, one circular and one square, have the same cross-sectional area (Fig. P8.3). Which has the larger hydraulic radius, and by what percentage?

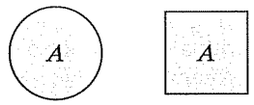


Figure P8.3

8.4 Prove that for a constant rate of discharge and a constant value of f the friction head loss in a pipe varies inversely as the fifth power of the diameter.

8.5 Two long pipes convey water between two reservoirs whose water surfaces are at different elevations. One pipe has a diameter twice that of the other; both pipes have the same length and the same value of f. If minor losses are neglected, what is the ratio of the flow rates through the two pipes?

8.6 Tests were made with 60°F water flowing in a 9-in-diameter pipe. They showed that, when $V = 12$ fps, $f = 0.0165$. Find the unit shear at the pipe wall and at radii of 0, 0.25, 0.4, 0.6, 0.85 times the pipe radius.

8.7 Air at 160°F and standard sea-level atmospheric pressure flows in a 15 in by 21 in rectangular air duct ($e = 0.0007$ in) at the rate of 1.2 lb/min. Find the friction head loss per 100 ft of duct. Express the answer in feet of air flowing and in pounds per square inch.

8.8 Water at 80°F flows in a conduit with a cross section shaped in the form of an equilateral triangle. The cross-sectional area of the duct is 160 in² and $e = 0.0018$ in. If the friction head loss is 3 ft in 150 ft, find the approximate flow rate.

8.9 Water at 20°C flows in a conduit with a cross section shaped in the form of an equilateral triangle. The cross-sectional area of the duct is 0.100 m² and $e = 0.045$ mm. If the friction head loss is 1 m in 50 m, find the approximate flow rate.

8.10 For laminar flow in a circular pipe, find the velocities at $0.1r$, $0.3r$, $0.5r$, $0.7r$, and $0.9r$. Plot the velocity profile.

8.11 Prove that the centerline velocity is twice the average velocity when laminar flow occurs in a circular pipe.

8.12 For laminar flow between two parallel, flat plates a small distance d apart, at what distance from the centerline (in terms of d) will the velocity be equal to the mean velocity?

8.13 Oil with an absolute viscosity of 0.16 N·s/m² and a density of 925 kg/m³ is flowing in a 200-mm-diameter pipe at 0.50 L/s. How much power is lost per meter of pipe length?

8.14 Water at 50°F enters a pipe with a uniform velocity of $U = 14$ fps. (a) What is the distance at which the transition occurs from a laminar to a turbulent boundary layer? (b) If the thickness of this initial laminar boundary layer is given by $4.91\sqrt{\nu x/U}$ (from Eq. 9.10), what is its thickness at the point of transition?

8.15 Water at 10°C enters a pipe with a uniform velocity of 3.4 m/s. (a) What is the distance at which a transition occurs from a laminar to a turbulent boundary layer? (b) If the thickness of this initial laminar boundary layer is given by $4.91\sqrt{\nu x/U}$, what is its thickness at the point of transition?

8.16 In a 36-in-diameter pipe velocities are measured as 18.5 fps at $r = 0$ and 18.0 fps at $r = 4.0$ in. Approximately what is the flow rate?

8.17 Water at 130°F flows in an 0.5-in-diameter copper tube ($e = 0.000005$ ft) at 1.2 gpm. Find the head loss per 100 ft, using Eq. (8.29) or (8.52) to find f. What is the centerline velocity, and what is the value of δ_v?

8.18 Water at 60°C flows in a 15-mm-diameter copper tube ($e = 0.0015$ mm) at 0.06 L/s. Find the head loss per 10 m, using Eq. (8.29) or (8.52) to find f. What is the centerline velocity, and what is the value of δ_v?

8.19 Repeat Prob. 8.17 for flow rates of 0.08 and 18 gpm.

8.20 Repeat Prob. 8.18 for flow rates of 0.004 and 0.9 L/s.

8.21 Oil ($s = 0.85$) with a viscosity of 0.0056 N·s/m² flows at a rate of 80 L/s in a 150-mm-diameter pipe having $e = 0.90$ mm. Find the head loss, using Eq. (8.29) or (8.52) to find f. Determine the shear stress at the pipe wall. Find the velocity 25 mm from the centerline. Under these conditions is the pipe behaving as a fully rough, transitional, or smooth pipe?

8.22 The flow rate in a 10-in-diameter pipe is 7 cfs. The flow is known to be turbulent, and the centerline velocity is 15.2 fps. Plot the velocity profile, and determine the head loss per foot of pipe.

8.23 The flow rate in a 250-mm-diameter pipe is 200 L/s. The flow is known to be turbulent, and the centerline velocity is 4.75 m/s. Plot the velocity profile, and determine the head loss per meter of pipe.

8.24 If the diameter of a pipe is doubled, what effect does this have on the flow rate for a given head loss? Consider (*a*) laminar flow; (*b*) turbulent flow.

8.25 Kerosene ($s = 0.813$) at a temperature of 70°F flows in a 2-in-diameter smooth brass pipeline at a rate of 6.5 gpm. (*a*) Find the friction head loss per foot. (*b*) For the same head loss, what would be the flow rate if the temperature of the kerosene were raised to 100°F?

8.26 Kerosene ($s = 0.813$) at a temperature of 20°C flows in a 75-mm-diameter smooth brass pipeline at a rate of 0.80 L/s. (*a*) Find the friction head loss per meter. (*b*) For the same head loss, what would be the flow rate if the temperature of the kerosene were raised to 40°C?

8.27 Water at 50°C flows in a 150-mm-diameter pipe with $V = 7.5$ m/s. Head loss measurements indicate that $f = 0.020$. Determine the value of e. Find the shear stress at the pipe wall and at $r = 30$ mm. What is the value of du/dy at $r = 30$ mm?

8.28 Oil ($s = 0.85$) with an absolute viscosity of 0.0056 N·s/m² flows in a 150-mm-diameter pipe having $e = 0.90$ mm. (*a*) Above what flow rate will this pipe behave as a fully rough pipe? (*b*) Below what flow rate will it behave as a smooth pipe?

8.29 Air at 70°C and atmospheric pressure flows with a velocity of 22 fps through a 3-in-diameter pipe ($e = 0.00015$ in). Find the friction head loss in 75 ft of pipe.

8.30 Air at 30°C and atmospheric pressure flows with a velocity of 6.5 m/s through a 75-mm-diameter pipe ($e = 0.002$ mm). Find the friction head loss in 30 m of pipe.

8.31 Crude oil ($s = 0.855$) at 50°C flows at 300 L/s through a 450-mm-diameter pipe ($e = 0.054$ mm) 1500 m long. Find the kilowatt loss.

8.32 When water at 50°F flows at 2.5 cfs in a 20-in pipeline, the head loss is 0.0004 ft/ft. What will be the friction head loss when glycerin at 68°F flows through this same pipe at the same rate?

8.33 Air at 90 psia and 70°F flows through a 4.5-in-diameter welded-steel pipe (see Table 8.1) at 60 lb/min. Find the friction head loss and pressure drop in 200 ft of this pipe. Assume the air to be of constant density.

8.34 Air flows at an average velocity of 0.7 m/s through a long 3.8-m-diameter circular tunnel ($e = 1.5$ mm). Find the friction head-loss gradient at a point where the air temperature and pressure are 20°C and 102 kPa abs, respectively. Find also the shear stress at the pipe wall and the thickness δ_v of the viscous sublayer.

8.35 Repeat Prob. 8.34 for the case where the average velocity is 7.0 m/s.

8.36 For **R** ranging from 10^2 to 10^7, make a plot of the values of α and β versus **R** for smooth brass pipes. On the same plot also show values of u_{max}/V.

8.37 Water at 40°F flows in a 42-in-diameter concrete pipe ($e = 0.022$ in). Determine **R**, τ_0, u_{max}/V, δ_v, δ_v/e, and the flow regime (hydraulic smoothness) for flow rates of 250, 25, 2.5, 0.25, and 0.025 cfs.

8.38 A steel pipe ($e = 0.0002$ ft) 13,450 ft long is to convey oil ($\nu = 0.00054$ ft²/sec) at 13 cfs from a reservoir with surface elevation 705 ft to one with surface elevation 390 ft (Fig. P8.38). Theoretically, what pipe size is required?

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

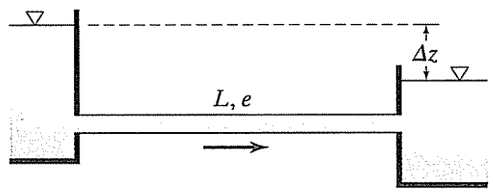


Figure P8.38

8.39 A steel pipe ($e = 0.065$ mm) 4200 m long is to convey oil ($\nu = 5.2 \times 10^{-5}$ m²/s) at 300 L/s from a reservoir with surface elevation 247 m to one with surface elevation 156 m (Fig. P8.38). Theoretically, what pipe size is required?

8.40 Water at 15°C flowing through 25 m of 100-mm-diameter galvanized iron pipe causes a friction head loss of 75 mm. Find the flow rate.

8.41 When gasoline with a kinematic viscosity of 6×10^{-6} ft²/sec flows in a 10-in-diameter smooth pipe, the friction head loss is 0.36 ft per 100 ft. Find the flow rate.

8.42 Oil with a kinematic viscosity of 0.00022 ft²/sec is to flow at 7.4 cfs with a friction head loss of 0.37 ft·lb/lb per 100 ft of pipe length. What size pipe ($e = 0.00013$ ft) is theoretically required?

8.43 Oil with a kinematic viscosity of 2.0×10^{-5} m²/s is to flow at 210 L/s, with a friction head loss of 0.42 N·m/N per 100 m of pipe length. What size pipe ($e = 0.038$ mm) is theoretically required?

8.44 Water at 150°F flows in a straight 0.863-in-diameter iron pipe ($e = 0.00015$ ft) between points *A* and *B* 350 ft apart. At *A* the elevation of the pipe is 112.0 ft and the pressure is 8.35 psi. At *B* the elevation of the pipe is 105.0 ft and the pressure is 8.80 psi. Compute the flow rate as accurately as you can.

8.45 Water at 60°C flows in a straight 20-mm-diameter pipe ($e = 0.060$ mm) between points *A* and *B* 100 m apart. At *A* the elevation of the pipe is 54.1 m and the pressure is 88.7 kPa. At *B* the elevation of the pipe is 52.0 m and the pressure is 91.8 kPa. Compute the flow rate as accurately as you can.

8.46 Using only a basic scientific calculator, solve Prob. 8.40 without trial and error.

8.47 Using only a basic scientific calculator, solve Prob. 8.45 without trial and error.

8.48 Using only a basic scientific calculator, solve Prob. 8.43 without trial and error.

8.49 Using only a basic scientific calculator, solve Prob. 8.39 without trial and error.

8.50 Using an equation solver on a programmable scientific calculator, or in computer software, solve the following without manual trial and error: (*a*) Prob. 8.40; (*b*) Prob. 8.41; (*c*) Prob. 8.44; (*d*) Prob. 8.45.

8.51 Using an equation solver on a programmable scientific calculator, or in computer software, solve the following without manual trial and error: (*a*) Prob. 8.38; (*b*) Prob. 8.39; (*c*) Prob. 8.42; (*d*) Prob. 8.43.

8.52 In a field test of the 16-ft-diameter Colorado River aqueduct flowing full, Manning's n was found to have a value of 0.0132 when 50°F water was flowing at a Reynolds number of 10.5×10^6. Determine the average value of e for this conduit.

8.53 Measurements were taken on pressure flow through a 2.2-m-diameter aqueduct built of concrete. When the water temperature was 10°C and the Reynolds number was 2.0×10^6, Manning's n was determined to be 0.0130. Find the average value of e for this aqueduct.

8.54 A smooth 15-in-diameter pipe is 400 ft long and has a flush entrance and a submerged discharge. It carries 70°F water at a velocity of 8 fps. What is the total head loss?

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

8.55 Oil with a kinematic viscosity of 9.7×10^{-5} m^2/s and a specific gravity of 0.94 flows at 3 m/s through a smooth 300-mm-diameter pipe which is 90 m long with a flush entrance and submerged discharge. What is the head loss in meters of oil and in kPa?

8.56 A smooth pipe consists of 52 ft of 9-in pipe followed by 310 ft of 18-in pipe, with an abrupt change of cross section at the junction (Fig. P8.56). The entrance is flush and the discharge is submerged. If it carries water at 60°F, with a velocity of 19 fps in the smaller pipe, what is the total head loss?

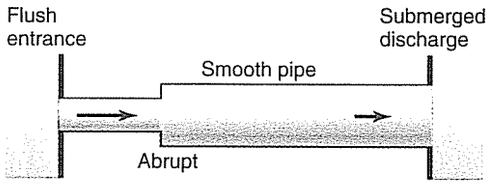


Figure P8.56

8.57 A smooth pipe consists of 12 m of 180-mm-diameter pipe followed by 75 m of 550-mm-diameter pipe, with an abrupt change of cross section at the junction (Fig. P8.56). The entrance is flush and the discharge is submerged. If it carries water at 15°C, with a velocity of 5.7 m/s in the smaller pipe, what is the total head loss?

8.58 Water flows at 12 fps through a vertical 4-in-diameter pipe standing in a body of water with its lower end 5 ft below the surface. Considering all losses and with $f = 0.026$, find the pressure in the pipe at a point 15 ft above the surface of the water when the flow is (*a*) downward; (*b*) upward.

8.59 Water flows at 2.5 m/s through a vertical 100-mm-diameter pipe standing in a body of water with its lower end 1 m below the surface. Considering all losses and with $f = 0.024$, find the pressure in the pipe at a point 4 m above the surface of the water when the flow is (*a*) downward; (*b*) upward.

8.60 A 12-in-diameter pipe ($f = 0.028$) 450 ft long runs from one reservoir to another, both ends of the pipe being under water (Fig. P8.60). The intake is square-edged. The difference between the water surface levels of the two reservoirs is 150 ft. Find (*a*) the flow rate; (*b*) the pressure in the pipe at a point 320 ft from the intake, where the elevation is 135 ft lower than the surface of the water in the upper reservoir.

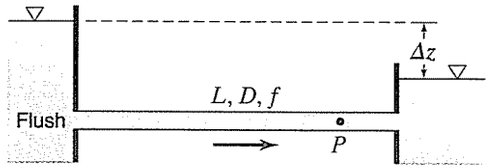


Figure P8.60

8.61 A 320-mm-diameter pipe ($f = 0.025$) 140 m long runs from one reservoir to another, both ends of the pipe being under water (Fig. P8.60). The intake is square-edged. The difference between the water surface levels of the two reservoirs is 36 m. Find (*a*) the flow rate; (*b*) the pressure in the pipe at a point 95 m from the intake, where the elevation is 39 m lower than the surface of the water in the upper reservoir.

8.62 A pump delivers water through 300 ft of 4-in fire hose ($f = 0.025$) to a nozzle that throws a 1-in-diameter jet. The loss coefficient of the nozzle is 0.04. The nozzle is 20 ft higher than the pump, and a jet velocity of 70 fps is required. What must be the pressure in the hose at the pump?

8.63 A pump delivers water through 100 m of 100-mm fire hose ($f = 0.025$) to a nozzle that throws a 25-mm-diameter jet. The loss coefficient of the nozzle is 0.04. The nozzle is 6 m higher than the pump, and a jet velocity of 20 m/s is required. What must be the pressure in the hose at the pump?

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

8.64 A 6-in-diameter horizontal jet of water is discharged into air through a nozzle (loss coefficient of 0.15) at a point 150 ft lower than the water surface above the intake (Fig. P8.64). The 12-in-diameter pipeline ($f = 0.015$) is 600 ft long, with a square-edged nonprojecting entrance. What is (*a*) the jet velocity; (*b*) the pressure at the base of the nozzle?

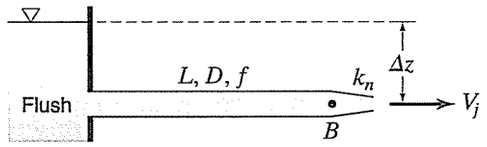


Figure P8.64

8.65 Refer to Fig. S8.11*b*. Suppose $\Delta z = 50$ ft, the line is 600 ft of 8-in-diameter pipe ($f = 0.025$), and the nozzle loss coefficient is 0.05. Find the jet diameter that will result in the greatest jet horsepower.

8.66 Refer to Fig. S8.11*b*. Suppose $\Delta z = 15$ m, the line is 180 m of 200-mm-diameter pipe ($f = 0.025$), and the nozzle loss coefficient is 0.05. Find the jet diameter that will result in the greatest jet power.

8.67 Water at 60°F flows through 800 ft of 12-in-diameter pipe between two reservoirs whose water surface elevation difference is 16 ft. The pipe entrance is flush and square-edged, and there is a half-open gate valve in the line. Using only a basic scientific calculator, find the flow rate (*a*) if $e = 0.0018$ in; (*b*) if e is 20 times larger.

8.68 Solve Prob. 8.67 without manual trial and error, by using an equation solver (*i*) on a programmable calculator; (*ii*) in computer software.

8.69 A 150 m long commercial steel pipe is to convey 30 L/s of oil ($s = 0.9$, $\mu = 0.038$ N·s/m^2) from one tank to another (submerged discharge) where the difference in elevation of the free liquid surfaces is 2 m (Fig. P8.69). The pipe entrance is flush and square-edged, and there is a fully-open gate valve in the line. Using only a basic scientific calculator, find the diameter theoretically required. (*Hint:* It may be easier to try for D than for f in this case.)

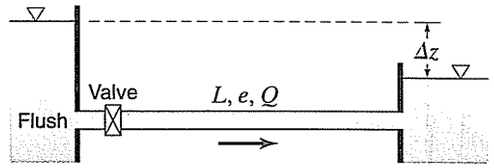


Figure P8.69

8.70 Solve Prob. 8.69 without manual trial and error, by using an equation solver (*a*) on a programmable calculator; (*b*) in computer software.

8.71 In Fig. 8.24, assume a pipe diameter of 350 mm, $f = 0.016$, $BC = 12$ m, $DE = 920$ m, and $\Delta z = 48$ m. Find the maximum theoretical flow rate if 15°C water is being pumped at an altitude of 1000 m above sea level. Point C is 6.0 m above the lower water surface.

8.72 In Fig. 8.24, assume that the pipe diameter is 4 in, $f = 0.035$, $BC = 15$ ft, $DE = 180$ ft, and $\Delta z = 60$ ft. The elevation of C is 10 ft above the lower water surface. (*a*) If the pressure head at C is to be no less than -20 ft, what is the maximum rate at which the water can be pumped? (*b*) If the efficiency of the pump is 65%, what is the horsepower required?

8.73 In a testing laboratory, a certain turbine has been found to discharge 10 cfs under a head h_t of 45 ft. In the field, it is to be installed near the end of a pipe 360 ft long (Fig. P8.73). The supply line (flush entrance) and discharge line (submerged exit) will both have diameters of 10 in with $f = 0.025$. The total fall from the surface of the water at intake to the surface of the tailwater will be 52 ft. What will be the head on the turbine, the rate of discharge, and the power delivered to the flow? Note that for turbines, $Q \propto \sqrt{h_t}$ (from Eq. 16.17).

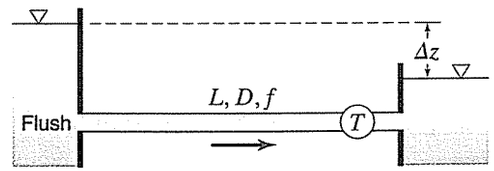


Figure P8.73

8.74 In a testing laboratory, a certain turbine has been found to discharge 285 L/s under a head h_t of 13.5 m. In the field, it is to be installed near the end of a pipe 110 m long (Fig. P8.73). The supply line (flush entrance) and discharge line (submerged exit) will both have diameters of 250 mm with $f = 0.024$. The total fall from the surface of the water at intake to the surface of the tailwater will be 15.5 m. What will be the head on the turbine, the rate of discharge, and the power delivered to the flow? Note that for turbines, $Q \propto \sqrt{h_t}$ (from Eq. 16.17).

8.75 Assume the total fall from the surface of one body of water to another is 130 ft. The water is conveyed by 250 ft of 15-in pipe ($f = 0.020$) that has its entrance flush with the wall (Fig. P8.75). At the end of the pipe is a turbine and draft tube that discharged 6 cfs of water when tested under a head of 63.1 ft in another location. Discharge losses are negligible. What is the rate of discharge through the turbine and the head on it under the present conditions? Note that for turbines, $Q \propto \sqrt{h_t}$ (from Eq. 16.17).

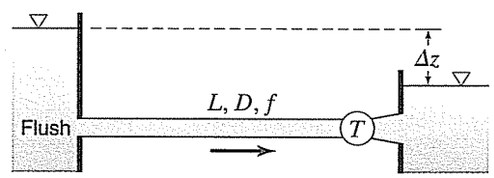


Figure P8.75

8.76 A pump is installed to deliver water from a reservoir of surface elevation zero to another of elevation 200 ft (Fig. P8.76). The 12-in-diameter suction pipe ($f = 0.020$) is 40 ft long, and the 10-in-diameter discharge pipe ($f = 0.032$) is 4500 ft long. The pump head may be defined as $h_p = 300 - 20Q^2$, where the pump head h_p is in feet and Q is in cubic feet per second. Compute the rate at which this pump will deliver the water. Also, what is the horsepower input to the water?

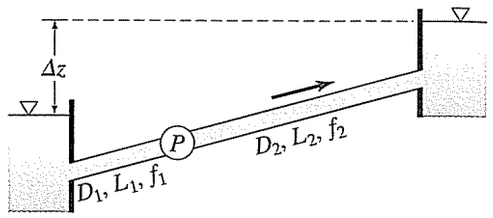


Figure P8.76

8.77 Refer to Fig. 8.25. Suppose that the water surface elevation, elevation of the pump, and elevation of the nozzle tip are 100, 90, and 120 ft, respectively. Pipe BC is 40 ft long, has a diameter of 8 in, with $f = 0.025$; pipe DE is 200 ft long, has a diameter of 10 in, with $f = 0.030$; the jet diameter is 6 in, and the nozzle loss coefficient is 0.04. Assume the pump is 80% efficient under all conditions of operation. Make a plot of flow rate and p_C/γ versus pump horsepower input. At what flow rate will cavitation begin in the pipe at C if the water temperature is 50°F and the atmospheric pressure is 13.9 psia?

8.78 In Fig. 8.24, assume the pipe diameter is 9 in, $f = 0.025$, $BC = 20$ ft, $DE = 2800$ ft, and $\Delta z = 125$ ft. Water is pumped at 5.5 cfs, the pump efficiency is 78%. (a) What horsepower is required? (b) If the elevation of C above the lower water surface is 12 ft, that of D is 15 ft, and that of E is 100 ft, compute the pressure heads at B, C, D, and E. (c) Sketch the energy line and the hydraulic grade line.

8.79 In Fig. 8.27, suppose that pipe 1 is 36-in smooth concrete, 5000 ft long; pipe 2 is 24-in cast iron, 3000 ft long; and pipe 3 is 20-in cast iron, 1300 ft long. The elevations of the water surfaces in reservoirs A and B are 225 and 200 ft, respectively, and the

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

discharge through pipe 1 is 42 cfs. The water temperature is 60°F. Using a basic scientific calculator only, find the elevation of the surface of reservoir C. Neglect minor losses, and assume the energy line and hydraulic grade line are coincident.

8.80 In Fig. 8.27, suppose that pipe 1 is 900-mm smooth concrete, 1500 m long; pipe 2 is 600-mm cast iron, 900 m long; and pipe 3 is 500-mm cast iron, 400 m long. The elevations of the water surfaces in reservoirs A and B are 75 and 67 m, respectively, and the discharge through pipe 1 is 1.2 m³/s. The water temperature is 15°C. Using a basic scientific calculator only, find the elevation of the surface of reservoir C. Neglect minor losses and assume the energy line and hydraulic grade line are coincident.

8.81 Solve Prob. 8.80 by using an equation solver in computer software.

8.82 In Fig. 8.27, suppose that pipe 1 is 36-in smooth concrete, 5000 ft long; pipe 2 is 24-in cast iron, 3000 ft long; and pipe 3 is 20-in cast iron, 1300 ft long. The surface elevations of reservoirs A and C are 250 and 180 ft, respectively, and the discharge through pipe 2 is 10 cfs of water into reservoir B. The water temperature is 60°F. Using a basic scientific calculator only, find the surface elevation of reservoir B.

8.83 Repeat Prob. 8.82, except that the 10 cfs discharge through pipe 2 is now from (not into) reservoir B.

8.84 Solve Prob. 8.82 without manual trial and error, by using an equation solver in computer software.

8.85 In Fig. 8.27, suppose that pipe 1 is 900-mm smooth concrete, 1500 m long; pipe 2 is 600-mm cast iron, 900 m long; and pipe 3 is 500-mm cast iron, 400 m long. The surface elevations of reservoirs A and C are 60 and 38 m, respectively, and the discharge through pipe 2 is 0.3 m³/s of water into reservoir B. The water temperature is 15°C. Using a basic scientific calculator only, find the surface elevation of reservoir B.

8.86 Repeat Prob. 8.85, except that the 0.3 m³/s discharge through pipe 2 is now from (not into) reservoir B.

8.87 Suppose, in Fig. 8.27, that pipes 1, 2, and 3 are 900 m of 600-mm, 300 m of 450-mm, and 1200 m of 400-mm, respectively, of new welded-steel pipe. The surface elevations of reservoirs A, B, and C are 36, 22 and 0 m, respectively. The water temperature is 15°C. Using a basic scientific calculator only, find the flow in all pipes.

8.88 Solve Prob. 8.87 without manual trial and error, by using an equation solver in computer software.

8.89 Suppose, in Fig. 8.27, that pipes 1, 2, and 3 are 3000 ft of 24-in, 1000 ft of 18-in and 4000 ft of 16-in, respectively, of new welded-steel pipe. The surface elevations of reservoirs A, B, and C are 120, 75 and 0 ft, respectively. The water temperature is 60°F. Using a basic scientific calculator only, find the flow in all pipes.

8.90 Solve Prob. 8.89 without manual trial and error, by using an equation solver in computer software.

8.91 Suppose that, in Fig. 8.27, pipe 1 is 1500 ft of 12-in new cast-iron pipe, pipe 2 is 800 ft of 6-in wrought-iron pipe, and pipe 3 is 1200 ft of 8-in wrought-iron pipe. The water surface elevation of reservoir B is 20 ft below that of A, while the junction J is 35 ft lower than the surface of A. In place of reservoir C, pipe 3 leads away to some other destination but its elevation at C is 60 ft below A. (a) Find the flow of 60°F water in each pipe. (b) Find the pressure head at C, when the pressure head at J is 25 ft. Neglect velocity heads.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

8.92 A pipeline 900 ft long discharges freely at a point 200 ft lower than the water surface at intake (Fig. P8.92). The pipe intake projects into the reservoir. The first 600 ft is of 10-in diameter, and the remaining 300 ft is of 6-in diameter. (*a*) Find the rate of discharge, assuming $f = 0.06$. If the junction point C of the two sizes of pipe is 150 ft below the intake water surface level, find the pressure head (*b*) just upstream of C and (*c*) just downstream of C. Assume a sudden contraction at C.

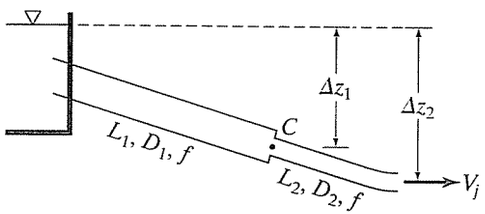


Figure P8.92

8.93 A pipeline 300 m long discharges freely at a point 50 m lower than the water surface at intake (Fig. P8.92). The pipe intake projects into the reservoir. The first 200 m is of 350-mm diameter, and the remaining 100 m is of 250-mm diameter. (*a*) Find the rate of discharge, assuming $f = 0.06$. If the junction point C of the two sizes of pipe is 40 m below the intake water surface level, find the pressure head (*b*) just upstream of C and (*c*) just downstream of C. Assume a sudden contraction at C.

8.94 Repeat Prob. 8.92, neglecting minor losses.

8.95 Repeat Prob. 8.93, neglecting minor losses.

8.96 Suppose that in Fig. 8.29 pipes 1, 2, and 3 are 750 ft of 4-in, 250 ft of 2-in, and 300 ft of 3-in, asphalt-dipped cast-iron pipe. With a total head loss of 25 ft between A and B, find the flow of 60°F water.

8.97 Suppose that in Fig. 8.29 pipes 1, 2, and 3 are 150 m of 80-mm, 60 m of 50-mm, and 120 m of 60-mm wrought-iron pipe. With a total head loss of 6 m between A and B, find the flow of 15°C water.

8.98 Two pipes connected in series are respectively 150 ft of 2-in ($e = 0.000006$ ft) and 450 ft of 8-in ($e = 0.0009$ ft) (Fig. P8.98). With a total head loss of 30 ft, find the flow of 60°F water, using a basic scientific calculator only. Neglect minor losses.

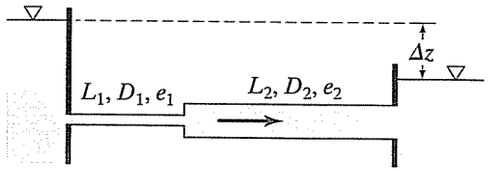


Figure P8.98

8.99 Repeat Prob. 8.96 for the case where the fluid is an oil with $s = 0.92$, $\mu = 0.000096$ lb·sec/ft^2.

8.100 Repeat Prob. 8.97 for the case where the fluid is an oil with $s = 0.94$, $\mu = 0.04$ N·s/m^2.

8.101 Three new cast-iron pipes, having diameters of 24, 21, and 18 in, respectively, each 450 ft long, are connected in series. The 24-in pipe leads from a reservoir (flush entrance), and the 18-in pipe discharges into the air at a point 15 ft lower than the reservoir water surface level. Assuming all changes in section to be abrupt, find the rate of discharge of water at 60°F.

8.102 Suppose that, in Fig. 8.30, pipes 1, 2, and 3 are of smooth brass as follows: 500 ft of 2-in, 350 ft of 3-in, and 750 ft of 4-in, respectively. When the total flow of 70°F crude oil ($s = 0.855$) is 0.7 cfs, find the head loss from A to B and the flow in each pipe. *Note:* This problem can be solved without trial and error, using a basic scientific calculator only.

8.103 Repeat Prob. 8.102 for the case where the total flow rate is 0.07 cfs. Does the "Note" still apply?

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

8.104 Repeat Prob. 8.102 for the case where all the pipes are of galvanized iron. Does the "Note" still apply?

8.105 Suppose that, in Fig. 8.30, pipes 1, 2, and 3 are of smooth plastic tubing as follows: 90 m of 20-mm, 150 m of 40-mm, and 80 m of 60-mm, respectively. When the total flow of 50°C crude oil ($s = 0.855$) is 7 L/s, find the head loss from A to B and the flow in each pipe. *Note:* This problem can be solved without trial and error, using a basic scientific calculator only.

8.106 Repeat Prob. 8.105 for the case where the total flow rate is 0.35 L/s. Does the "Note" still apply?

8.107 Repeat Prob. 8.105 for the case where all the pipes are of galvanized iron. Does the "Note" still apply?

8.108 Referring to Fig. P8.108, A is at elevation 30 ft, and the pipe characteristics are as follows: pipe B is 5000 ft long, of 3 ft diameter, with $f = 0.035$; pipe E is 4500 ft long, of 2 ft diameter, with $f = 0.035$; and pipe C is 5000 ft long, of 3 ft diameter, with $f = 0.025$. When the pump develops 30 ft of head, the velocity of flow in pipe C is 5 fps. Neglecting minor losses, find (*a*) the flow rates in cubic feet per second in pipes B and E under these conditions; (*b*) the elevation of the discharge end of pipe E.

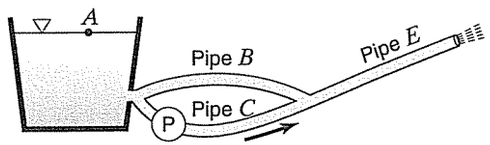


Figure P8.108

8.109 Repeat Prob. 8.108 for the case where the velocity in pipe C is 6 fps, with all other data remaining the same.

8.110 Refer to Fig. P8.108. Assume the water surface in the reservoir is instead at elevation 200 m. Pipes B, C, and E are all 600 m long, and they all have a diameter of 500 mm with $f = 0.030$. When the pump develops 15 m of head, the velocity in pipe C is 5.0 m/s. Neglecting minor losses, find (*a*) the flow rate in all pipes; (*b*) the elevation of the discharge end of pipe B.

8.111 In Fig. P8.111 pipe AB is 1200 ft long, of 8 in diameter, with $f = 0.035$; pipe BC (upper) is 800 ft long, of 6 in diameter, with $f = 0.025$; pipe BC (lower) is 900 ft long, of 4 in diameter, with $f = 0.045$; and pipe CD is 550 ft long, of 6 in diameter, with $f = 0.025$. The elevations are: reservoir water surface = 150 ft, $A = 120$ ft, $B = 70$ ft, $C = 60$ ft, and $D = 30$ ft. There is free discharge to the atmosphere at D. Neglecting velocity heads, (*a*) compute the flow in each pipe; (*b*) determine the pressures at B and C.

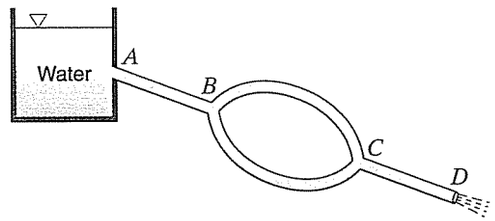


Figure P8.111

8.112 In Fig. P8.111, pipe AB is 600 m long, of 180 mm diameter, with $f = 0.035$; pipe BC (upper) is 500 m long, of 120 mm diameter, with $f = 0.025$; pipe BC (lower) is 400 m long, of 160 mm diameter, with $f = 0.030$; and pipe CD is 900 m long, of 320 mm diameter, with $f = 0.020$. The elevations are: reservoir water surface = 150 m, $A = 100$ m, $B = 60$ m, $C = 50$ m, $D = 20$ m. There is free discharge to the atmosphere at D. Neglecting velocity heads, (*a*) compute the flow in each pipe; (*b*) determine the pressures at B and C. Comment on the practicality of this system.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

8.113 A 10-in cast-iron pipe 1500 ft long forms one link in a pipe network. If the velocities to be encountered are assumed to fall within the range of 4–10 fps, derive an equation for the flow of water at 60°F in this pipe in the form $h_f = KQ^n$. (*Hint:* Using information from Fig. 8.11 and Table 8.1, set up two simultaneous equations corresponding to the ends of the desired velocity range; then solve for the unknowns K and n.)

8.114 A 250-mm cast-iron pipe 400 m long forms one link in a pipe network. If the velocities to be encountered are assumed to fall within the range of 0.75–2.3 m/s, derive an equation for the flow of water at 15°C in this pipe in the form $h_f = KQ^n$. (*Hint:* See Prob. 8.113.)

8.115 The pipes in the system shown in Fig. P8.115 are all galvanized iron. (*a*) With a flow of 15 cfs, find the head loss from A to D. (*b*) What should be the diameter of a single pipe from B to C such that it replaces pipes 2, 3, and 4 without altering the capacity for the same head loss from A to D?

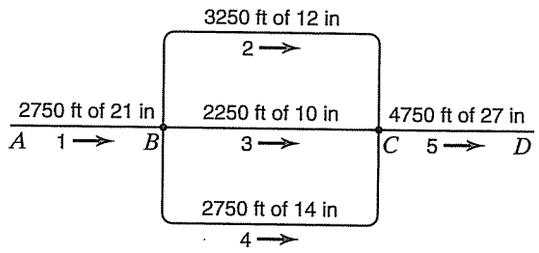


Figure P8.115

8.116 (*a*) With the same pipe lengths, sizes, and connections as in Prob. 8.115, find the flow in each pipe if the head loss from A to D is 150 ft and if all pipes have $f = 0.020$. Also find the head losses from A to B, B to C, and C to D. (*b*) Find the new head loss distributions and the percentage increase in the capacity of the system achieved by adding another 12-in pipe 3250 ft long between B and C.

8.117 Find the magnitude and direction of the flow in network lines *ab* and *bc* (Fig. P8.117) after making two sets of corrections. The numbers on the figure are the K values of each line; take $n = 2.0$. Start by assuming initial flows as follows: 9 cfs in lines *ab* and *cd*, 6 cfs in lines *ac* and *bd*, and 3 cfs in line *bc*.

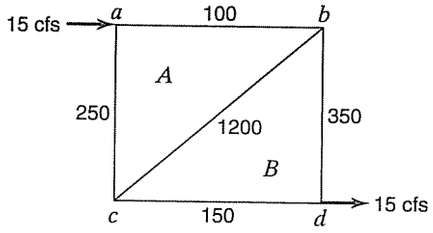


Figure P8.117

8.118 Find the magnitude and direction of the flow in network lines *ab* and *bc* (Fig. P8.118) after making two sets of corrections. The numbers on the figure are the K values of each line; take $n = 2.0$. Start by assuming initial flows as follows: 0.3 m³/s in lines *ab* and *cd*, 0.2 m³/s in lines *ac* and *bd*, and 0.1 m³/s in line *bc*.

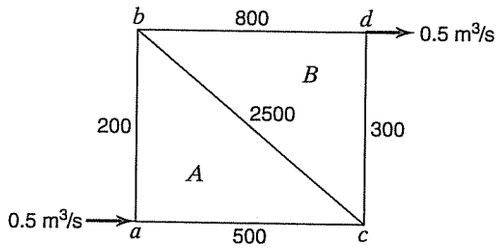


Figure P8.118

8.119 Carry the solution for the pipe network of Fig. P8.119 through four trials, to find the flow in each pipe. For simplicity, take $n = 2.0$ and use the value of f for complete turbulence, as given by Eq. (8.54). All pipes are cast iron, and are at the same elevation. For initial flows, assume only values of 30, 15, and 0 L/s (the zeros in *dg* and *fh*). If the pressure head at a is 40 m, find the pressure head at d (which might represent a fire demand, for example) neglecting velocity heads.

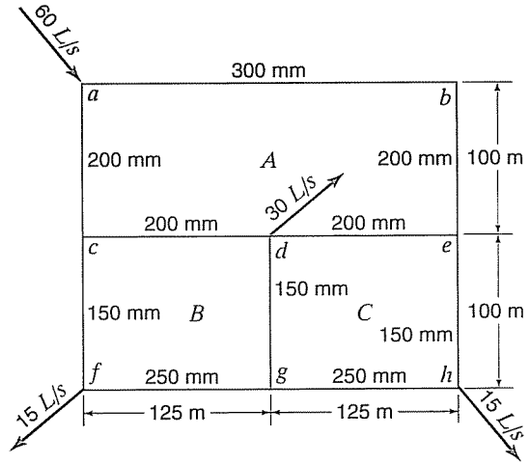


Figure P8.119

8.120 Carry the solution for the pipe network of Fig. P8.120 through five trials, to find the flow in each pipe. The 12-in and 16-in pipes are of average cast iron, while the 18-in and 24-in sizes are of average concrete ($e = 0.003$ ft). Assume $n = 2.0$, and use the values of f from Eq. (8.54) for complete turbulence. If the pressure at h is 80 psi, find the pressure at f.

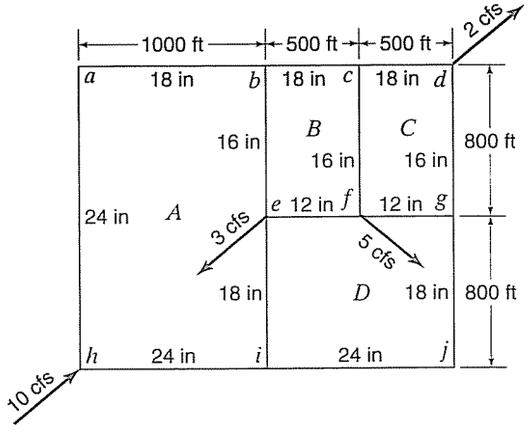


Figure P8.120

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

CHAPTER 9
Forces on Immersed Bodies

In this chapter we shall primarily discuss fluid phenomena encountered in incompressible flow or in low-velocity compressible flow where the effects of compressibility are negligible. Near the end of the chapter in Sec. 9.14 we briefly discuss the effects of compressibility on drag and lift. These become important at Mach numbers above 0.7.

9.1 INTRODUCTION

When a flowing fluid completely surrounds bodies, such as airplanes, birds, automobiles, raindrops, submarines, and fish, the flows are known as ***external flows.*** Such a body, wholly immersed in a homogeneous fluid, may be subject to two kinds of forces arising from relative motion between the body and the fluid. These forces are termed the ***drag*** and the ***lift,*** depending on whether the force is parallel to the motion or perpendicular to it, respectively. Fluid mechanics draws no distinction between two cases of relative motion, namely, when a body moves rectilinearly at constant speed through a stationary fluid or when a fluid travels at constant velocity past a stationary body. So we can test airplane models in wind tunnels and torpedo models in water tunnels and predict with confidence the behavior of their prototypes when moving through still fluid. When studying the subject we find it somewhat simpler to fix our ideas on the stationary body in the moving fluid, while the practical result desired is more frequently associated with a body moving through still fluid.

In this chapter we shall first consider the drag, or resistance, forces. As we shall not be concerned with wave action at a free surface, gravity does not enter the problem, and the forces involved are those due to inertia and viscosity.[1] We can view the drag forces on a submerged body as having two components: a

[1] Actually, without viscosity there could be no drag force at all. The flow of a frictionless fluid about any body, as constructed mathematically or by the flow-net techniques of Chap. 4, produces opposing stagnation points at the nose and tail of the body. The pressure distribution, as computed from the Bernoulli theorem and integrated over the entire body, always adds up to zero in the direction of the flow. We call this situation ***d'Alembert's paradox.***

pressure drag F_p and a ***friction drag*** (or ***surface drag***) F_f. The pressure drag, which we often call the ***form drag*** because it depends largely on the form or shape of the body, is equal to the integration of the components in the direction of motion of all the pressure forces exerted on the surface of the body. We can express[2] it as the dynamic component of the stagnation pressure (from Eq. 5.17) acting on the projected area A of the body *normal to the flow* times a coefficient C_p that is dependent on the geometric form of the body and generally determined by experiment. So,

$$F_p = C_p \rho \frac{V^2}{2} A \tag{9.1}$$

The friction drag along a body surface is equal to the integral of the components of the shear stress along the surface in the direction of motion. For convenience, we commonly express the friction drag in the same general form as Eq. (9.1). So,

$$F_f = C_f \rho \frac{V^2}{2} BL \tag{9.2}$$

where C_f = friction-drag coefficient, dependent on viscosity among other factors
L = length of surface parallel to flow
B = transverse width, conveniently approximated for irregular shapes by dividing total surface area by L

It is important to note that for a body such as a plate with both sides immersed in the fluid, Eq. (9.2) gives the drag for *one side* only.

From our experience with pipe flow, we should expect that the friction drag would be more amenable to a theoretical approach than pressure drag. It turns out that this is not necessarily the case. In Chap. 8 we described the ***boundary layer*** as a very thin layer of fluid adjacent to a surface, in which viscosity is important, while we can consider the fluid outside this layer as frictionless or ideal. This concept, originated by Ludwig Prandtl in 1904 (see Sec. 8.8), is one of the important advances in modern fluid mechanics. It means that we can actually use the mathematical theory of ideal fluid flow, including the flow-net method we discussed in Chap. 4, to determine the streamlines in the real fluid at a short distance from a solid boundary. We can then use the Bernoulli theorem to determine the normal pressures on the surface, for such pressures are practically the same as those outside this thin layer.

[2] Note that Eq. (9.1) is of the same general form as the expression for the drag on a sphere that we developed by dimensional analysis in Sec. 7.7. The comparison indicates that $C_p = \phi(\mathbf{R})$. If we had considered the effects of compressibility in Sec. 7.7, the comparison would have shown that $C_p = \phi(\mathbf{R}, \mathbf{M})$. Similarly, we can compare Eq. (9.1) with Eq. (7.13).

The boundary layer may be entirely laminar, or it may be primarily turbulent with a viscous sublayer, as in Fig. 8.7. We usually define the thickness δ of the boundary layer as the distance from the boundary to the point where the velocity is 99% of the undisturbed velocity, i.e., to where $u = 0.99U$. This thickness increases with distance from the leading edge of a surface, as shown in both Figs. 8.4 and 8.7. As in Figs. 9.1 and 8.7 for a plate and a reentrant pipe, a stagnation point (Sec. 4.10) is located at the leading tip, and because the flow approaching this point must decelerate, the boundary layer begins a short distance upstream. At higher approach velocities the boundary layer becomes thinner in accordance with Eq. (8.38), and this upstream distance becomes very small. Engineers commonly neglect it, and assume that the boundary layer originates from the leading edge, as we shall assume in the following sections.

There is an important difference between flow around immersed bodies and pipe flow. In pipe flow the boundary layers from the opposite walls of the pipe merge together after a certain distance and the flow becomes "all boundary layer," while with airplanes, submarines, trains, etc., the boundary layer, even though it may reach a thickness of several inches, is still small compared with the dimensions of the "ideal fluid" outside the boundary layer.

9.2 FRICTION DRAG OF BOUNDARY LAYER—INCOMPRESSIBLE FLOW

Figure 9.1 illustrates the growth of a boundary layer along one side of a smooth plate in steady flow of an incompressible fluid. Let us consider the control volume $ABCD$ shown in Fig. 9.2, which extends a distance δ (delta) from the plate, where δ is the thickness of the boundary layer at a distance x along the plate. For convenience, in this analysis we will assume that the velocity $u = U$ at the edge of the boundary layer, instead of the usual $0.99U$. The undisturbed velocity U exists along control surface AB. The pressure forces around the periphery of the control volume will cancel one another out, because the undisturbed flow field pressure must exist along AB and DA, and the distance BC $(=\delta)$ is so small it will have a negligible effect on pressure variations.

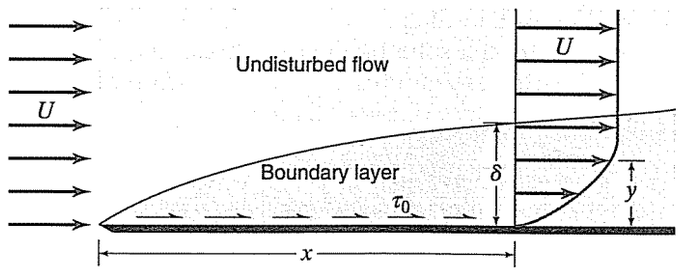


Figure 9.1
Growth of a boundary layer along a smooth plate (vertical scale exaggerated).

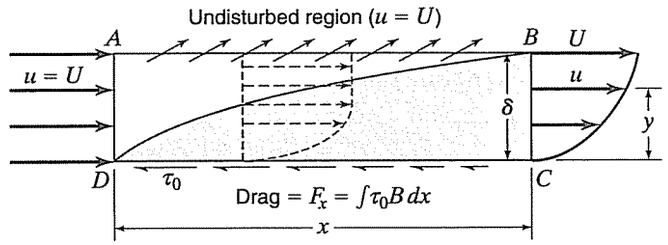


Figure 9.2
Control volume $ABCD$ for flow over one side of a flat plate.

Applying Eq. (6.7*a*), we get

$$-F_x = -\text{drag} = \begin{array}{l}\text{rate of momentum in } x \text{ direction leaving through } BC \\ + \text{ rate of momentum in } x \text{ direction leaving through } AB \\ - \text{ rate of momentum in } x \text{ direction entering through } DA\end{array} \tag{9.3}$$

Because the flow $Q_{BC} < Q_{DA}$, there is flow out of the control volume across control surface AB and $Q_{AB} = Q_{DA} - Q_{BC}$.

If the width of the plate is B, neglecting edge effects, we can express the flows and momentums across the control surfaces as:

Control surface	Flow rate	Rate of momentum in x direction
DA	$UB\delta$	$\rho(UB\delta)U$
BC	$B\int_0^\delta u\,dy$	$\rho B\int_0^\delta u^2\,dy$
AB	$UB\delta - B\int_0^\delta u\,dy$	$\rho\left(UB\delta - B\int_0^\delta u\,dy\right)U$

Substituting these momentum values into Eq. (9.3) gives

$$F_x = \rho B\int_0^\delta u(U-u)\,dy \tag{9.4}$$

where F_x is the total friction drag of the plate on the fluid *from the leading edge up to* x directed to the left, as shown on Fig. 9.2. Equal and opposite to this is the drag of the fluid on the plate.

We will now assume that the velocity profiles within the boundary layer at various distances along the plate are similar to one another, i.e.,

$$\frac{u}{U} = f\left(\frac{y}{\delta}\right) = f(\eta)$$

where η (eta) $= y/\delta$ is the proportional distance. Experimental evidence tells us that this assumption is valid if there is no pressure gradient along the surface and

if the boundary layer does not change from laminar to turbulent within the region considered. Then, substituting $u = Uf(\eta)$, $y = \delta\eta$, and $dy = \delta\, d\eta$ into Eq. (9.4) and noting that the limits of integration become 0 to 1, we get

$$F_x = \rho B U^2 \delta \int_0^1 f(\eta)[1 - f(\eta)]\, d\eta$$

which, for convenience, we can write as

$$F_x = \rho B U^2 \delta \alpha \tag{9.5}$$

where α is the indicated integral (not to be confused with the kinetic-energy correction factor α discussed in Chap. 5), which depends only on the boundary-layer velocity distribution, $u/U = f(\eta)$.

Next we shall investigate the local wall shear stress τ_0 at distance x from the leading edge. From the definition of surface resistance, $dF_x = \tau_0 B\, dx$, or

$$\tau_0 = \frac{1}{B}\frac{dF_x}{dx} = \frac{1}{B}\frac{d}{dx}(\rho B U^2 \delta \alpha)$$

and as all terms in the expression for F_x are constant except δ,

$$\tau_0 = \rho U^2 \alpha \frac{d\delta}{dx} \tag{9.6}$$

This expression for the shear stress is valid for either laminar or turbulent flow in the boundary layer, but in this form it is not useful until we have evaluated the quantities α and $d\delta/dx$.

9.3 Laminar Boundary Layer for Incompressible Flow Along a Smooth Flat Plate

As in the case of laminar flow in pipes, we can examine the shear stress at the plate wall with the aid of the velocity gradient and the definition of viscosity.

$$\tau_0 = \mu\left(\frac{du}{dy}\right)_{y=0} = \frac{\mu}{\delta}\left(\frac{du}{d\eta}\right)_{\eta=0} = \frac{\mu U}{\delta}\left[\frac{df(\eta)}{d\eta}\right]_{\eta=0}$$

which can be abbreviated to

$$\tau_0 = \frac{\mu U \beta}{\delta} \tag{9.7}$$

where β, like α, is a dimensionless function of the velocity-distribution curve and is given by the expression in square brackets.

Equations (9.6) and (9.7) are two independent expressions for τ_0. Equating them to one another results in a simple differential equation,

$$\delta\, d\delta = \frac{\mu\beta}{\rho U \alpha} dx$$

with the solution

$$\frac{\delta^2}{2} = \frac{\mu\beta}{\rho U \alpha} x + C$$

where $C = 0$, since $\delta \approx 0$ at $x = 0$ for large U. Therefore

$$\delta = \sqrt{\frac{2\mu\beta x}{\rho U \alpha}} = \sqrt{\frac{2\beta}{\alpha}}\frac{x}{\sqrt{\mathbf{R}_x}} \tag{9.8}$$

where we can call

$$\mathbf{R}_x = \frac{xU\rho}{\mu} = \frac{xU}{\nu}$$

the ***local Reynolds number.*** Note that $\mathbf{R}_x$ increases linearly in the downstream direction. By examining the first expression of Eq. (9.8) we see that the thickness of the laminar boundary layer increases with distance from the leading edge; thus the shear stress (Eq. 9.7) decreases as the layer grows along the plate.

To evaluate Eq. (9.8), we must know or assume the velocity profile in the laminar boundary layer. The velocity distribution can be closely represented by a parabola, as shown in Fig. 9.3. In dimensionless terms, this curve becomes

$$\frac{u}{U} = f(\eta) = 2\eta - \eta^2 \tag{9.9}$$

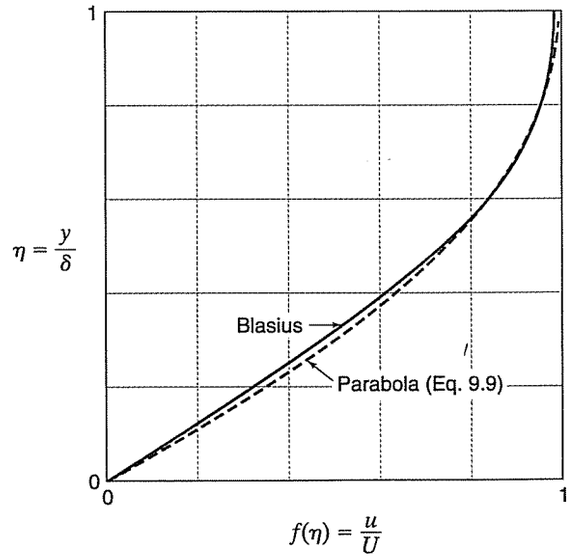


Figure 9.3
Velocity distributions in the laminar boundary layer on a flat plate.

Blasius derived the other velocity profile in Fig. 9.3 from the fundamental equations of viscous flow, with all factors considered, and the result has been closely checked by experiment. This curve is based on defining the thickness δ as that for which $u = 0.99U$.

As Prob. 9.2 can demonstrate, the parabolic distribution will give numerical values for α and β of 0.133 and 2.0, respectively. The Blasius curve yields $\alpha = 0.135$ and $\beta = 1.63$, where the principal difference lies in the milder slope of the velocity gradient at the wall. When we substitute the Blasius values in Eq. (9.8) we obtain

$$\frac{\delta}{x} = \sqrt{\frac{2 \times 1.63}{0.135}} \frac{1}{\sqrt{\mathbf{R}_x}} = \frac{4.91}{\sqrt{\mathbf{R}_x}} \tag{9.10}$$

If we substitute the value of δ from Eq. (9.10) into Eq. (9.7) with $\beta = 1.63$, we get for the shear stress

$$\tau_0 = 0.332 \frac{\mu U}{x} \sqrt{\mathbf{R}_x} \tag{9.11}$$

But we have another expression for shear stress, given in Eq. (8.10), $\tau_0 = c_f \rho U^2/2$.[3] Setting this equal to Eq. (9.11) enables us to determine the *local* friction coefficient

$$c_f = \frac{0.332 \mu U \sqrt{\mathbf{R}_x}}{\rho x U^2/2} = \frac{0.664}{\sqrt{\mathbf{R}_x}} \tag{9.12}$$

If the boundary layer remains laminar over a length L of the plate, the total friction drag on one side of the plate is given by integrating Eq. (9.11):

$$F_f = B \int_0^L \tau_0 dx = 0.332 B \sqrt{\rho \mu U^3} \int_0^L x^{-1/2} dx = 0.664 B \sqrt{\rho \mu L U^3} \tag{9.13}$$

Comparing Eq. (9.13) with a standard friction-drag equation (9.2), and substituting U for the more general velocity V, we can see that for a laminar boundary layer,

Laminar layer:
$$C_f = 1.328 \sqrt{\frac{\mu}{\rho L U}} = \frac{1.328}{\sqrt{\mathbf{R}}} \tag{9.14}$$

[3] Note the apparent inconsistency between the notation used here and that used in Chap. 8. In pipe flow, the significant reference velocity is the mean velocity V in the pipe, while in flow over a plate, it is the *uniform* velocity U of the undisturbed fluid. Likewise, in Chap. 8 we used C_f to denote a friction coefficient for the fully developed boundary layer in a pipe, while here we use c_f to denote the *local* friction coefficient of the growing layer.

where $\mathbf{R}$ is based on the characteristic length of the *whole* plate. The laminar boundary layer will remain laminar if undisturbed, up to a value of $\mathbf{R}_x$ of about 500,000. In this region the layer becomes turbulent, increasing noticeably in thickness and displaying a marked change in velocity distribution.

SAMPLE PROBLEM 9.1 (*a*) Find the friction drag on one side of a smooth flat plate 6 in (150 mm) wide and 18 in (500 mm) long, placed longitudinally in a stream of crude oil ($s = 0.925$) at 60°F (20°C) flowing with undisturbed velocity of 2 fps (600 mm/s) (Fig. S9.1). (*b*) Find the thickness of the boundary layer and the shear stress at the trailing edge of the plate.

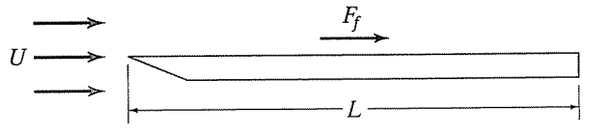


Figure S9.1

Solution (BG units)

(*a*) Fig. A.2 for crude oil ($s = 0.925$) at 60°F: $\nu = 0.00105$ ft^2/sec

Table A.1 for water at 60°F: $\rho = 1.938$ slug/ft^3

Then, at $x = L$: $$\mathbf{R} = \frac{LU}{\nu} = \frac{(18/12)2}{0.00105} = 2860$$

which is well within the laminar range; that is, $\mathbf{R} < 500{,}000$.

Eq. (9.14): $$C_f = \frac{1.328}{\sqrt{\mathbf{R}}} = \frac{1.328}{\sqrt{2860}} = 0.0248$$

Eq. (9.2): $$F_f = 0.0248(0.925 \times 1.938)\frac{2^2}{2}\frac{6 \times 18}{144} = 0.0668 \text{ lb} \quad \textit{ANS}$$

(*b*) Eq. (9.10): $$\frac{\delta}{x} = \frac{4.91}{\sqrt{2860}} = 0.0919$$

$$\delta = 0.0919(18/12) = 0.1378 \text{ ft} = 1.653 \text{ in} \quad \textit{ANS}$$

Eq. (9.11), at $x = L$:

$$\tau_0 = 0.332 \frac{\mu U}{L} \sqrt{\mathbf{R}} = 0.332 \frac{\rho \nu U}{L} \sqrt{\mathbf{R}}$$

$$\tau_0 = 0.332 \frac{0.00105(0.925 \times 1.938)2}{18/12} \sqrt{2860} = 0.0445 \text{ lb/ft}^2 \quad \textit{ANS}$$

Solution (SI units)

(*a*) Fig. A.2 for crude oil ($s = 0.925$) at 20°C: $\nu = 0.73 \times 10^{-4}$ m²/s

Table A.1 for water at 20°C: $\rho = 998.2$ kg/m³

Then, at $x = L$: $$\mathbf{R} = \frac{LU}{\nu} = \frac{(0.50\text{ m})(0.60\text{ m/s})}{0.73 \times 10^{-4}} = 4110$$

which is well within the laminar range; that is, $\mathbf{R} < 500000$.

Eq. (9.14): $$C_f = \frac{1.328}{\sqrt{\mathbf{R}}} = 0.0207$$

Eq. (9.2): $$F_f = 0.0207(0.925 \times 998.2)\frac{\text{kg}}{\text{m}^3}\frac{(0.6\text{ m/s})^2}{2}(0.15 \times 0.50)\text{ m}^2$$

$$F_f = 0.258\frac{\text{kg·m}}{\text{s}^2} \times \frac{\text{N}}{\text{kg·m/s}^2} = 0.258\text{ N} \qquad \textit{ANS}$$

(*b*) Eq. (9.10):

$$\frac{\delta}{x} = \frac{4.91}{\sqrt{4110}} = 0.0766, \quad \delta = 0.0766(500\text{ mm}) = 38.3\text{ mm} \qquad \textit{ANS}$$

Eq. (9.11), at $x = L$:

$$\tau_0 = 0.332\frac{\mu U}{L}\sqrt{\mathbf{R}} = 0.332\frac{\rho\nu U}{L}\sqrt{\mathbf{R}}$$

$$\tau_0 = 0.332\frac{(0.925 \times 998.2)(0.73 \times 10^{-4})0.6}{0.50}\sqrt{4110} = 1.721\text{ N/m}^2 \qquad \textit{ANS}$$

EXERCISES

9.3.1 For the critical Reynolds number of 500,000 for transition from laminar to turbulent flow in the boundary layer, find the corresponding critical Reynolds number for flow in a circular pipe. How does this compare with the value given in Chap. 8? (*Hint:* Let the boundary-layer thickness correspond to the radius of the pipe in laminar flow, and let the undisturbed velocity U of the boundary-layer theory represent the centerline velocity u_{max} of the pipe flow.)

9.3.2 Determine the shear stress 9 in and 18 in back from the leading edge of the plate in Sample Prob. 9.1.

9.3.3 Find the shear stress and the thickness of the boundary layer (*a*) at the center and (*b*) at the trailing edge of a smooth, flat plate 3.0 m wide and 0.6 m long parallel to the flow, immersed in 15°C water flowing at an undisturbed velocity of 0.9 m/s. Assume a laminar boundary layer over the whole plate. Also, (*c*) find the total friction drag on one side of the plate.

9.4 TURBULENT BOUNDARY LAYER FOR INCOMPRESSIBLE FLOW ALONG A SMOOTH FLAT PLATE

Comparing the laminar and turbulent boundary layers in Fig. 9.4, the velocity distribution in the turbulent boundary layer shows a much steeper gradient near the wall and a flatter gradient throughout the remainder of the layer. As we would expect, then, the wall shear stress is greater in the turbulent boundary layer than in the laminar layer at the same Reynolds number. In this case, however, it is not practical to proceed along the lines of Eqs. (9.6) and (9.7), determining the shear stress from the velocity gradient at the wall. As an approximation, we turn instead to turbulent flow in a circular pipe because of the wealth of experimental information on that subject compared with that of flow in a turbulent boundary layer along a smooth flat plate. We learned in Eq. (8.19) that the shear stress at the wall of a pipe is given by

$$\tau_0 = f\rho\frac{V^2}{8} \tag{9.15}$$

where V again denotes the average velocity in the pipe. Now we shall assume that the turbulent boundary layer occupies all the region between the wall and the centerline of the pipe, as in Fig. 9.5. The radius of the pipe then becomes the thickness of the boundary layer, and, by analogy, the velocity at the center of the pipe, here denoted by U, corresponds to the undisturbed velocity at the outer edge of the boundary layer. We can obtain an approximate relation between V and U by using the pipe factor equation (8.43). Taking a middle value of $f = 0.028$ and allowing for the 1% difference in velocity between the edge of the boundary layer and the free stream, we have

$$U = 1.235V \tag{9.16}$$

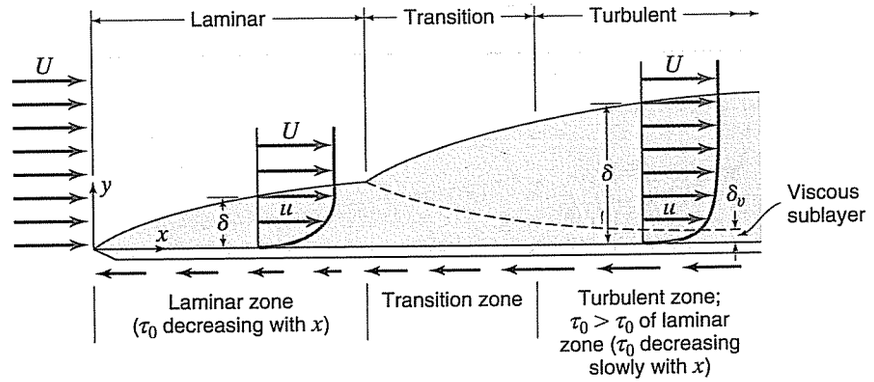


Figure 9.4
Laminar and turbulent boundary layers along a smooth flat plate (vertical scale greatly exaggerated).

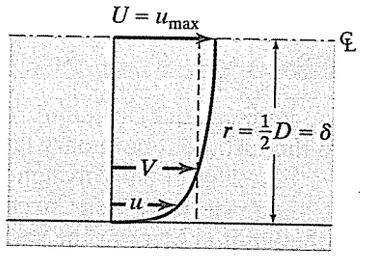


Figure 9.5
Flow in a pipe as a turbulent boundary layer.

To proceed further, we need a simple relation between f and $\mathbf{R}$ for turbulent pipe flow in a smooth pipe. The Blasius equation (8.48) provides a useful relationship. Substituting Eqs. (8.48) and (9.16) into (9.15) gives

$$\tau_0 = \frac{0.316}{(DV/\nu)^{1/4}}\frac{\rho V^2}{8} = \frac{0.316}{[(2\delta/\nu)(U/1.235)]^{1/4}}\frac{\rho}{8}\left(\frac{U}{1.235}\right)^2 = \frac{0.0230\rho U^2}{(\delta U/\nu)^{1/4}} \tag{9.17}$$

If we now equate the two expressions for τ_0 (Eqs. 9.6 and 9.17), we obtain

$$\rho U^2 \alpha \frac{d\delta}{dx} = \frac{0.023\rho U^2}{(\delta U/\nu)^{1/4}} \tag{9.18}$$

Separating variables and integrating this expression (with the condition $\delta = 0$ at $x = 0$), yields

$$\delta = \left(\frac{0.0287}{\alpha}\right)^{4/5}\left(\frac{\nu}{Ux}\right)^{1/5} x \tag{9.19}$$

In using Eq. (9.6) we assume that we know the velocity distribution in the turbulent boundary layer. Of the many formulas for this distribution that have been proposed, the most convenient for our purpose is Eq. (8.49), the seventh-root law, which we can express as

$$u = Uf(\eta) = U(\eta)^{1/7} = U(y/\delta)^{1/7} \tag{9.20}$$

For the case of such a velocity distribution, $\alpha = 0.0972$. This is found by evaluating the integral definition of α (Sec. 9.2):

$$\alpha = \int_0^1 f(\eta)[1 - f(\eta)]\,d\eta$$

where $\eta = y/\delta$
$d\eta = dy/\delta$
$f(\eta) = (y/\delta)^{1/7}$

Substituting this value for α into Eq. (9.19) gives

$$\frac{\delta}{x} = 0.377\left(\frac{\nu}{Ux}\right)^{1/5} = \frac{0.377}{\mathbf{R}_x^{1/5}} \tag{9.21}$$

and substituting this value of δ into Eq. (9.17) yields

$$\tau_0 = 0.0587\rho\frac{U^2/2}{\mathbf{R}_x^{1/5}} \tag{9.22}$$

And because (Sec. 9.3) $\tau_0 = c_f\rho U^2/2$,

$$c_f = \frac{0.0587}{\mathbf{R}_x^{1/5}} \tag{9.23}$$

With this value for c_f (the ***local friction coefficient***), we can express the total friction drag on one side of the plate as

$$F_f = B\int_0^L \tau_0 dx = 0.0735\rho\frac{U^2}{2}\left(\frac{\nu}{UL}\right)^{1/5} BL \tag{9.24}$$

or in Eq. (9.2) the friction-drag coefficient for a turbulent boundary layer is

Turbulent layer: $$C_f = \frac{0.0735}{\mathbf{R}^{1/5}} \quad \text{for } 500{,}000 < \mathbf{R} < 10^7 \tag{9.25}$$

where we again note that the characteristic length in $\mathbf{R}$ is L, the total length of the plate parallel to the flow. For Reynolds numbers above 10^7, Schlichting has proposed a modification of Eq. (9.25) that agrees better with experimental results:[4]

Turbulent layer: $$C_f = \frac{0.455}{(\log \mathbf{R})^{2.58}} \quad \text{for } \mathbf{R} > 10^7 \tag{9.26}$$

For Reynolds numbers less than 10^7, this equation gives values for C_f that are very close to those given by Eq. (9.25), and consequently engineers commonly use Eq. (9.26) over the entire range of Reynolds numbers above 500,000.

SAMPLE PROBLEM 9.2 (*a*) Find the frictional drag on the top and sides of a box-shaped moving van 8 ft wide, 10 ft high, and 35 ft long, traveling at 60 mph through air at 50°F (Fig. S9.2). Assume that the vehicle has a rounded nose so that the flow does not separate from the top and sides (see Fig. 9.12*b*). Assume also that even though the top and sides of the van are relatively smooth, there is enough roughness that, for all practical purposes, a turbulent boundary layer starts immediately at the leading edge. (*b*) Find the thickness of the boundary layer and the shear stress at the trailing edge.

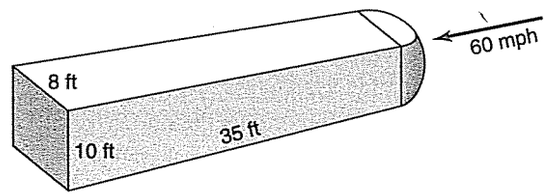


Figure S9.2

[4] H. Schlichting, Boundary Layer Theory, Part II, *NACA Tech. Memo.* 1218, p. 39, 1949.

Solution
Table A.2 for air at 50°F: $\nu = 0.000152$ ft²/sec, $\rho = 0.00242$ slug/ft³. 60 mph = 88 fps.

$$(a) \qquad \mathbf{R} = \frac{LU}{\nu} = \frac{35 \times 88}{0.000152} = 20{,}260{,}000, \quad \log \mathbf{R} = 7.31$$

As $\mathbf{R} > 10^7$, use Eq. (9.26),

$$C_f = \frac{0.455}{(7.31)^{2.58}} = 0.00269$$

Then, by Eq. (9.2),

$$F_f = 0.00269(0.00242)\frac{(88)^2}{2}(10 + 8 + 10)35 = 24.7 \text{ lb} \qquad \textit{ANS}$$

(*b*) Applying Eq. (9.21), at $x = 35$ ft,

$$\delta_{35} = \frac{36 \times 0.377}{(20{,}260{,}000)^{1/5}} = 0.456 \text{ ft} \qquad \textit{ANS}$$

From Eq. (9.22), with $x = 35$ ft,

$$(\tau_0)_{35} = 0.0587(0.00242)\frac{88^2}{2}\left(\frac{0.000152}{88 \times 35}\right)^{1/5} = 0.01901 \text{ lb/ft}^2 \qquad \textit{ANS}$$

EXERCISES

9.4.1 A lifeguard determines the wind velocity 6 ft above the beach to be 25 fps. If one wishes to get out of the wind by lying down, what would be the velocity at (*a*) 0.5 ft, and (*b*) at 1.0 ft above beach level?

9.4.2 Compute C_f for $\mathbf{R} = 10^7$ using Eqs. (9.25) and (9.26), and compare the two values.

9.4.3 Find the shear stress on the sides of the van in Sample Prob. 9.2 at (*a*) 2 ft, (*b*) 12 ft, and (*c*) 22 ft back from the leading edge of the sides.

9.4.4 Assume that the boundary layer of Exer. 9.3.3 is disturbed near the leading edge. Compute the corresponding quantities for the turbulent boundary layer covering the entire plate, and compare the results.

9.4.5 A 280-ft-long streamlined train has 8.5-ft-high sides and an 8-ft-wide top. Compute the power required to overcome the skin-friction drag when the train is traveling at 90 mph through the ICAO standard atmosphere at sea level, assuming the drag on the sides and top to be equal to that on one side of a flat plate 25 ft wide and 280 ft long.

9.5 FRICTION DRAG FOR INCOMPRESSIBLE FLOW ALONG A SMOOTH FLAT PLATE WITH A TRANSITION REGIME

In Secs. 9.3 and 9.4 we have treated separately the resistance due to laminar and turbulent boundary layers on a smooth flat plate. We now wish to determine how to compute the total friction drag when there is a transition from the laminar to the turbulent boundary layer part way along the plate surface.

Let x_c in Fig. 9.6 be the distance from the leading edge to the point where the boundary layer becomes turbulent, which will normally occur at a value of $\mathbf{R}_x$ of about 500,000. The drag of the turbulent portion of the boundary layer may be approximated as the drag that would occur if a turbulent boundary layer extended along the whole plate, minus the drag of a fictitious turbulent layer from the leading edge to x_c. Thus

$$F_{\text{turb}} \approx F_{\text{total turb}} - F_{\text{turb to } x_c}$$

When we add this to the drag from the laminar boundary layer up to x_c, we have, from Eqs. (9.2), (9.14), (9.25), and (9.26), for the total drag, assuming the plate is long enough[5] that $\mathbf{R} > 10^7$,

$$F_f = \rho\frac{U^2}{2}B\left[\frac{1.328x_c}{\sqrt{\mathbf{R}_c}} + \frac{0.455L}{(\log \mathbf{R})^{2.58}} - \frac{0.0735x_c}{\mathbf{R}_c^{1/5}}\right]$$

where $\mathbf{R}_c$ is based on the length x_c to the point of transition, while $\mathbf{R}$ is based on the total length L of the plate, as before. Next we observe that

$$\frac{\mathbf{R}_c}{\mathbf{R}} = \frac{x_c}{L}, \quad \text{or} \quad x_c = \frac{\mathbf{R}_c}{\mathbf{R}}L$$

and thus $$F_f = \rho\frac{U^2}{2}BL\left[1.328\frac{\sqrt{\mathbf{R}_c}}{\mathbf{R}} + \frac{0.455}{(\log \mathbf{R})^{2.58}} - \frac{0.0735\mathbf{R}_c^{4/5}}{\mathbf{R}}\right]$$

and, because the quantity in brackets is the friction-drag coefficient C_f in Eq. (9.2), we get for $\mathbf{R}_c = 500{,}000$,

With transition at $\mathbf{R}_c = 500{,}000$:
$$C_f = \frac{0.455}{(\log \mathbf{R})^{2.58}} - \frac{1700}{\mathbf{R}} \tag{9.27}$$

[5] This expression for F_f is also quite accurate in the undisturbed transition regime for a shorter plate with $\mathbf{R} < 10^7$ because Eqs. (9.25) and (9.26) give almost identical values for C_f when $\mathbf{R} < 10^7$.

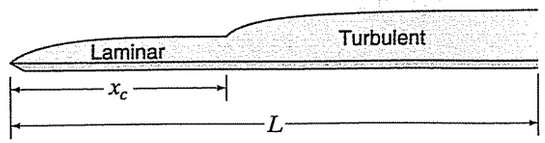


Figure 9.6
Boundary layers along a smooth flat plate of finite length.

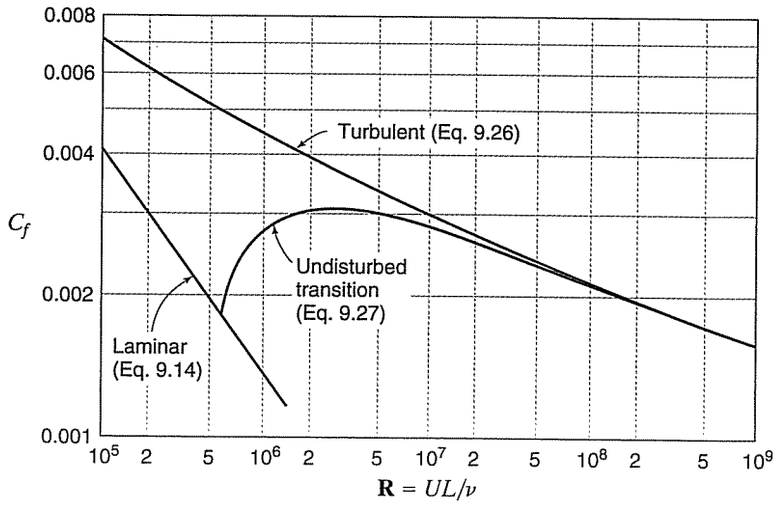


Figure 9.7
Drag coefficients for a smooth flat plate.

Equations (9.14), (9.26), and (9.27) are plotted in Fig. 9.7, for comparison.

All our treatment of laminar and turbulent boundary layers has so far been based on the surface of the immersed body being smooth. However, a local region of excess roughness in the laminar zone can "trip" the laminar layer into becoming a turbulent layer at Reynolds numbers less than 500,000. We call the height of the roughness that will cause this tripping the ***critical roughness,*** which is given approximately by[6]

$$e_c = \frac{15\nu}{\sqrt{\tau_0/\rho}} = 26.0\frac{\nu}{U}\mathbf{R}_x^{1/4} \tag{9.28}$$

where τ_0 is determined by the *laminar* equation (9.11), not by the turbulent equation (9.22).

We see that the height of the critical roughness depends on its distance from the leading edge. As the laminar boundary layer grows along the plate, the roughness must be greater in order to upset the stability of the layer. Recall that when $\mathbf{R}_x$ reaches a value in the neighborhood of 500,000, the laminar layer of itself becomes unstable, however smooth the surface, and changes to a turbulent boundary layer with a thin viscous sublayer. As in the case of flow in pipes, we consider the surface hydraulically smooth if the effect of the roughness projections does not extend through this sublayer.

In the turbulent zone the thickness of the viscous sublayer is not a clearly determinable quantity, but engineers appear agreed that the thickness of this

[6] I. Tani, J. Hama, and S. Mituisi, On the Permissible Roughness in the Laminar Boundary Layer, *Aeronaut. Res. Inst., Tokyo Imp. Univ., Rept.* 15, p. 419, 1940.

predominantly laminar film is given approximately (see Sec. 8.10 for pipe flow) by

$$\delta_v = \frac{5\nu}{u_*} \tag{9.29}$$

while the transition layer extends out to

$$\delta_t = \frac{70\nu}{u_*} \tag{9.30}$$

where the shear-stress velocity u_* was first defined in Sec. 8.10, and τ_0 is given by the *turbulent* equation (9.22), so that

$$u_* = \sqrt{\frac{\tau_0}{\rho}} = \frac{0.1713U}{\mathbf{R}_x^{0.1}} \tag{9.31}$$

If the roughness height is only of the order of δ_v, the surface may still be considered smooth, but if the roughness height is greater than δ_t, the surface is truly rough and the drag is materially increased.

Finally we can note that a plate or wing that is to incur minimum drag must be very smooth near the leading edge, where the laminar layer or sublayer is thinnest, while greater roughness may be tolerated near the trailing edge. Because the wall shear is so much greater in a turbulent than in a laminar boundary layer, anything that we can do to delay the breakdown of the laminar boundary layer will greatly reduce the frictional drag force on a body. The ***laminar flow wing*** for aircraft is one for which suction slots along the leading edge of the wing together with a smooth leading edge and a properly shaped wing profile help to maintain a favorable pressure gradient (Sec. 9.6) along the upper surface of the wing. This delays the breakdown of the laminar boundary layer, and thus such wings have much less drag than conventional ones.

SAMPLE PROBLEM 9.3 A small submarine, which we can suppose approximates a cylinder 10 ft in diameter and 50 ft long, travels submerged at 3 knots (5.06 fps) in seawater at 40°F (Fig. S9.3). (*a*) Find the friction drag assuming no separation from the sides. (*b*) Find the value of the critical roughness for a point 1 ft from the nose of the submarine. (*c*) Find the height of roughness at the midsection of the submarine that would class the surface as truly rough.

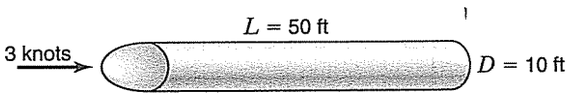


Figure S9.3

Solution
Viscosity of seawater ≈ viscosity of fresh water; specific weight ≈ 64 lb/ft^3. Table A.1 at 40°F: $\nu = 0.00001664$ ft^2/sec.

(*a*)
$$\mathbf{R} = \frac{50 \times 5.06}{0.00001664} = 1.525 \times 10^7$$

Eq. (9.27) or Fig. 9.7: $C_f = 0.00270$

$$F_f = 0.00270\left(\frac{64}{32.2}\right)\frac{(5.06)^2}{2}(\pi \times 10)50 = 108.0 \text{ lb} \qquad \textit{ANS}$$

(*b*) At $x = 1$ ft: $\mathbf{R}_x = \dfrac{5.06 \times 1}{0.00001664} = 305{,}000$

Eq. (9.28): $e_c = \dfrac{26 \times 0.00001664}{5.06}(305{,}000)^{1/4} = 0.00200 \text{ ft}$ ***ANS***

(*c*) At $x = 25$ ft: $\mathbf{R}_x = \dfrac{5.06 \times 25}{0.00001664} = 7.63 \times 10^6$

Eq. (9.31): $u_* = \dfrac{0.1713 \times 5.06}{(7.63 \times 10^6)^{0.1}} = 0.1778 \text{ fps}$

Eq. (9.30): $\delta_t = \dfrac{70 \times 0.00001664}{0.1778} = 0.00653 \text{ ft}; \quad e = 0.00653 \text{ ft}$ ***ANS***

EXERCISES

9.5.1 A 7.5-ft by 1.5-ft smooth, thin, flat plate with sharpened edges is submerged in 60°F water moving with a velocity of 1.4 fps in the direction of the 7.5 ft length. What is the total drag?

9.5.2 An 1-in-diameter harpoon 6 ft long, with a sharp tip, is launched at 20 fps into 60°F water. Find (*a*) the friction drag; (*b*) the maximum thickness of the boundary layer.

9.5.3 An airplane wing with a chord length of 3 m parallel to the flow moves through standard atmospheric air at an altitude of 6 km with a speed of 350 km/h. Find (*a*) the critical roughness for a point one-tenth the chord length back from the leading edge; (*b*) the surface drag on an 8-m-span section of this wing.

9.6 BOUNDARY-LAYER SEPARATION AND PRESSURE DRAG

The motion of a thin stratum of fluid lying wholly inside the boundary layer is determined by three forces:

1. The forward pull of the outer free-moving fluid, transmitted through the laminar boundary layer by viscous shear and through the turbulent boundary layer by momentum transfer (Sec. 8.9);

2. The viscous retarding effect of the solid boundary, which must, by definition, hold the fluid stratum immediately adjacent to it at rest;
3. The pressure gradient along the boundary: the stratum is accelerated by a pressure gradient whose pressure decreases in the direction of flow and is retarded by an adverse gradient.

In Secs. 9.2–9.5 we have restricted the treatment of fluid resistance to the drag of the boundary layer along a smooth flat plate located in an unconfined fluid, that is to say, in the *absence of a pressure gradient.* In the presence of a favorable pressure gradient, the boundary layer is "held" in place. This is what occurs in the accelerated flow around the ***forebody,*** or upstream portion, of a cylinder, sphere, or other object, such as that of Fig. 4.12. If a particle enters the boundary layer near the forward stagnation point with a low velocity and high pressure, its velocity will increase as it flows into the lower-pressure region along the side of the body. But there will be some retardation from wall friction (force 2 above), so that its total useful energy will be reduced by a corresponding conversion into thermal energy.

What happens next can best be explained by referring to Fig. 9.8. Let A represent a point in the region of accelerated flow with a normal velocity distribution in the boundary layer (either laminar or turbulent), while B is the point where the velocity outside the boundary layer reaches a maximum. Then C, D, and E are points downstream where the velocity outside the boundary layer decreases, resulting in an increase in pressure in accordance with ideal-flow theory. Thus the velocity of the layer close to the wall is reduced at C and finally brought to a stop at D. Now the increasing pressure calls for further retardation; but this is impossible, and so the boundary layer actually ***separates*** from the wall. At E there is a ***backflow*** next to the wall, driven in the direction of decreasing pressure—upstream in this case—and feeding fluid into the boundary layer that has left the wall at D.

Downstream from the point of separation, the flow is characterized by irregular turbulent eddies (Sec. 4.2), formed as the separated boundary layer

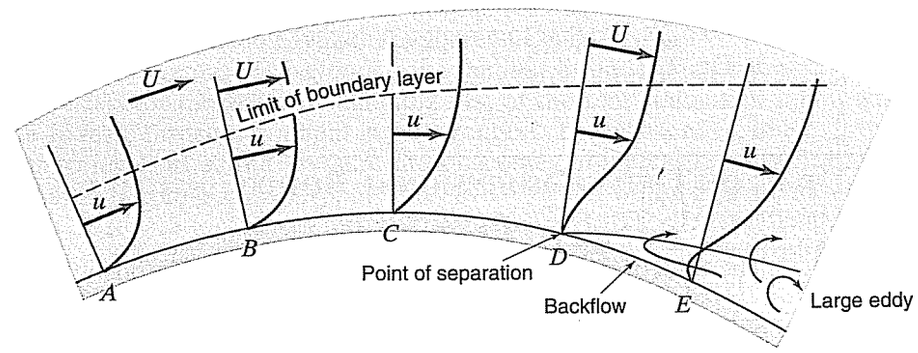


Figure 9.8
Growth and separation of boundary layer owing to increasing pressure gradient. Note that U has its maximum value at B and then gets smaller.

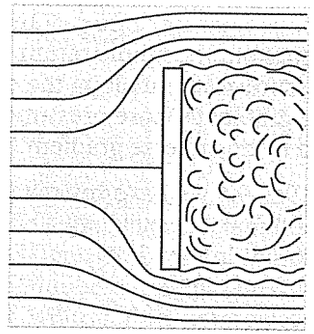


Figure 9.9
Turbulent wake behind a flat plate held normal to the flow.

becomes rolled up in the reversed flow. This condition generally extends for some distance downstream until the eddies are worn away by viscous attrition. The whole disturbed region is called the ***turbulent wake*** of the body (Fig. 9.9).

Because the eddies cannot convert their kinetic energy of rotation into an increased pressure, as the ideal-fluid theory would dictate, the pressure within the wake remains close to that at the separation point. Because this is always less than the pressure at the forward stagnation point, a net pressure difference results, tending to move the body with the flow, and this force is the pressure drag.

Although the laminar and turbulent boundary layers behave in essentially the same manner at a point of separation, the *location* of the separation point on a given curved surface will be very different for the two cases. In the laminar layer the transfer of momentum from the rapidly moving outer strata through the viscous-shear process to the inner strata is slow and ineffective. Consequently, the laminar boundary layer is "weak" and cannot stick long to the wall against an adverse pressure gradient. The transition to a turbulent boundary layer, on the other hand, brings a violent mixing of the faster-moving outer strata into the slower-moving inner strata, and vice versa. The mean velocity close to the boundary is greatly increased, as shown in Fig. 9.4. This added energy enables the boundary layer to better withstand the adverse pressure gradient, with the result that *with a turbulent boundary layer the point of separation is moved downstream* to a region of higher pressure. An example of this is shown in Fig. 9.11.

9.7 DRAG ON THREE-DIMENSIONAL BODIES (INCOMPRESSIBLE FLOW)

The total drag on a body is the sum of the friction drag and the pressure drag:

$$F_D = F_f + F_p$$

In the case of a well-streamlined body, such as an airplane wing or the hull of a submarine, the friction drag is the major part of the total drag, and we can

estimate it by the methods of Secs. 9.2–9.5 on the boundary layer. Only rarely is it desired to compute the pressure drag separately from the friction drag. Usually, when the wake resistance becomes significant, we are interested only in the total drag. Indeed, it is customary to employ a single equation that gives the total drag[7]

$$F_D = C_D \rho \frac{V^2}{2} A \tag{9.32}$$

in terms of an overall drag coefficient C_D, with the other quantities the same as in Eq. (9.1), except that in the case of the lifting vane (like an airplane wing) the area A is defined as the product of the span and the mean chord (Figs. 9.15 and 9.21). In such a case the area is neither strictly parallel to nor normal to the flow.

In the case of a body with sharp corners, such as the plate of Fig. 9.9, set normal to the flow, separation always occurs at the same point, and the wake extends across the full projected width of the body. This results in a relatively constant value of C_D, as we can see from the plot for the flat disk in Fig. 9.10. If the body has curved sides, however, we determine the location of the separation point by whether the boundary layer is laminar or turbulent. This location in turn determines the size of the wake and the amount of the pressure drag.

The foregoing principles are vividly illustrated in the case of the flow around a sphere. For very low Reynolds numbers ($DV/\nu < 1$, in which D is the diameter of the sphere), the flow about the sphere is completely viscous, and the friction drag is given by ***Stokes' law,***

$$F_D = 3\pi\mu VD \tag{9.33}$$

Equating this equation to Eq. (9.32), where A is defined as $\pi D^2/4$, the frontal area of the projected sphere, gives the result that $C_D = 24/\mathbf{R}$. The similarity between this case and the value of the friction factor for laminar flow in pipes is at once apparent. This regime of the flow about a sphere is shown as the straight line at the left of the log–log plot of C_D versus $\mathbf{R}$ in Fig. 9.10.

As we increase $\mathbf{R}$ beyond 1, the laminar boundary layer separates from the surface of the sphere, beginning first at the rear stagnation point, where the adverse pressure gradient is the strongest. The curve of C_D in Fig. 9.10 begins to level off as the pressure drag becomes of increasing importance, and the drag becomes more proportional to V^2. When we further increase $\mathbf{R}$, the point of separation moves forward on the sphere, until at $\mathbf{R} \approx 1000$ the point of separation becomes fairly stable at about 80° from the forward stagnation point.

[7] In the equations for total drag on submerged bodies, we revert to the use of V to designate a general reference velocity.

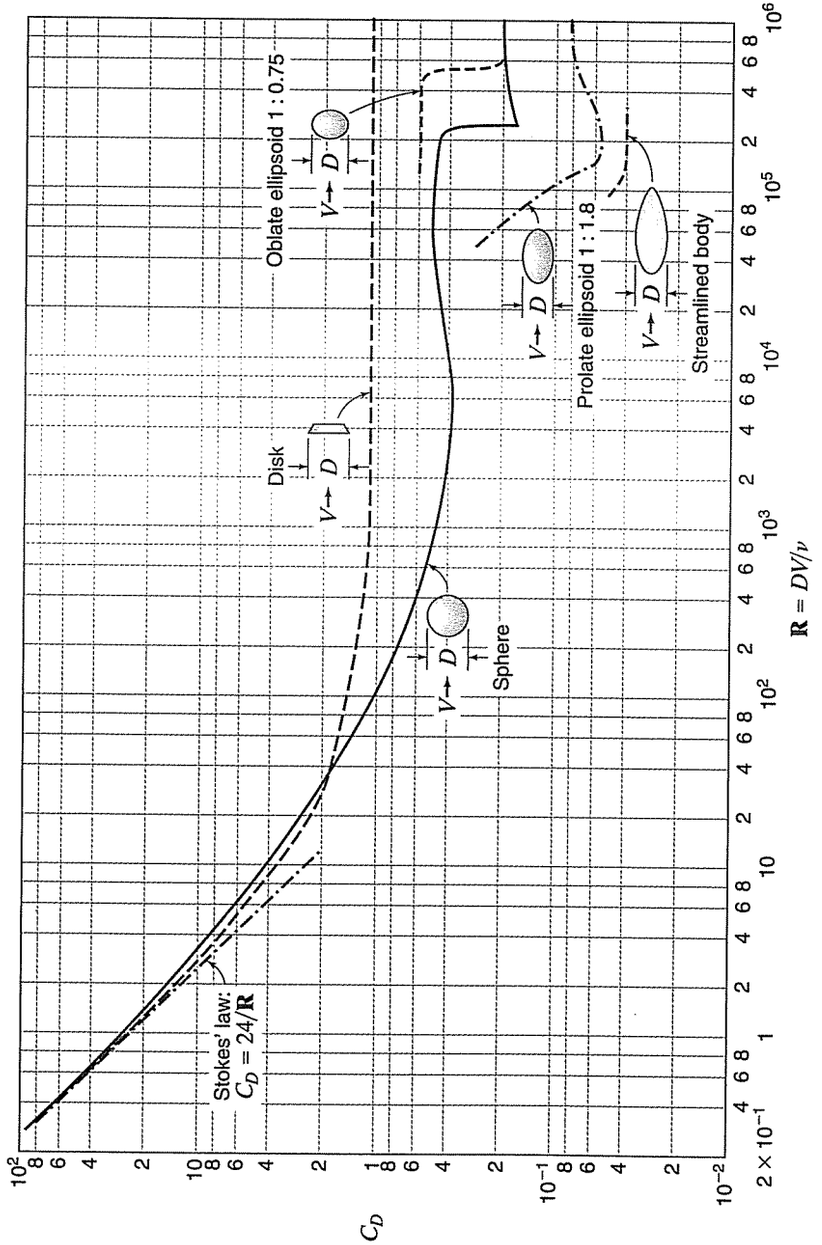
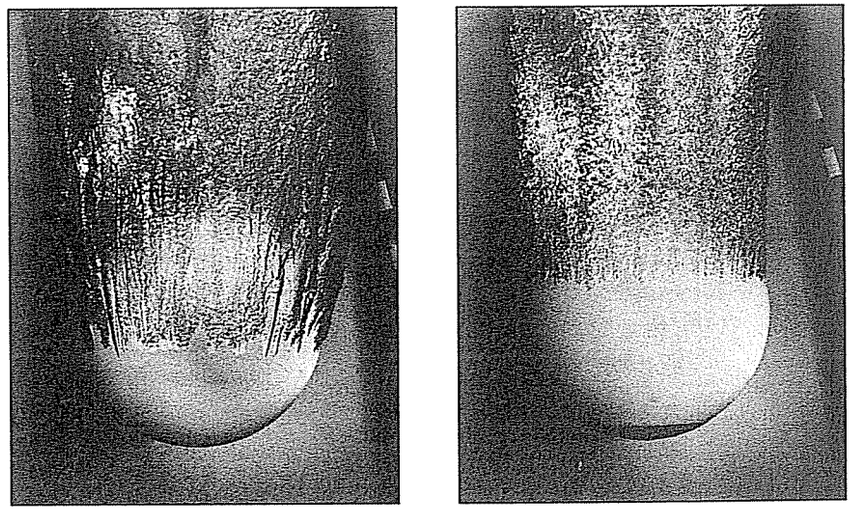


Figure 9.10
Drag coefficients for axisymmetric bodies.

For a considerable range of Reynolds numbers, conditions remain fairly stable, the laminar boundary layer separating from the forward half of the sphere and C_D remaining fairly constant at about 0.45. At a value of **R** of about 250,000 for the smooth sphere, however, the drag coefficient is suddenly reduced by about 50%, as we can see in Fig. 9.10. The reason for this lies in a change from a laminar to a turbulent boundary layer on the sphere. The point of separation moves back to something like 115°, from the stagnation point, with a consequent decrease in the size of the wake and the pressure drag.

If the level of turbulence in the free stream is high, the transition from laminar to turbulent boundary layer will take place at lower Reynolds numbers. Because this phenomenon of shift in separation point is so well defined, the sphere is often used as a turbulence indicator. The Reynolds number producing a value of C_D of 0.3—which lies in the middle of the rapid-drop range—becomes an accurate measure of the turbulence.

As we already noted, the transition from a laminar to a turbulent boundary layer can also be prematurely induced by artificially roughening the surface over a local region. The two pictures of Fig. 9.11 clearly show the effectiveness of this procedure. When the nose of the sphere is roughened, the boundary layer becomes turbulent and the separation point moves back. The added roughness and turbulent boundary layer cause an increase in friction drag, to be sure, but this is of secondary importance compared with the marked decrease in the size and effect of the wake. This is the main reason why the surface of a golf ball is



(a) $C_D \approx 0.4$ (b) $C_D \approx 0.2$

Figure 9.11
Shift in point of separation on 8.5-in-diameter sphere (bowling ball) at a velocity of approximately 25 fps in water. (a) Smooth sphere—laminar boundary layer. (b) Sphere with 4-in-diameter patch of sand grains cemented to nose—turbulent boundary layer. Reynolds numbers are the same. (U.S. Naval Ordnance Test Station, Pasadena Annex)

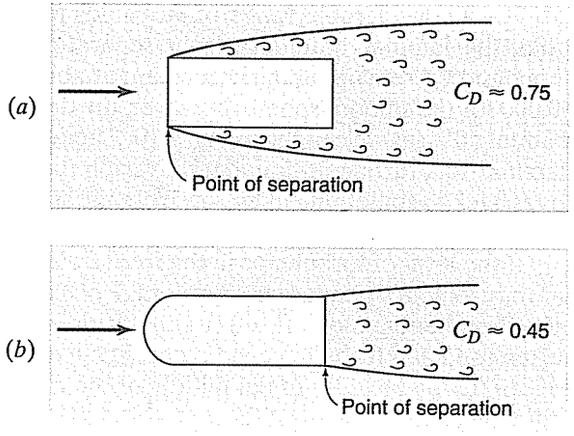


Figure 9.12
Plan view of flow about a motor vehicle (delivery van). (*a*) Blunt nose with separated flow along the entire side wall and a large drag coefficient $C_D = 0.75$. (*b*) Round nose with separation at the rear of the vehicle and smaller drag coefficient $C_D = 0.45$. (Adapted from H. Schlichting, *Boundary Layer Theory,* 4th ed., p. 34, McGraw-Hill, New York, 1960.)

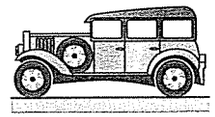
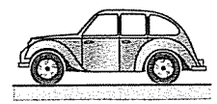
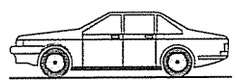
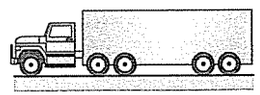
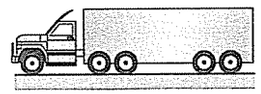
dimpled. A smooth-surfaced ball would have greater overall drag and would travel only about 60% as far when driven.

Plots of C_D versus $\mathbf{R}$ for various other three-dimensional shapes are also shown in Fig. 9.10. Note here that the object of ***streamlining*** a body is to move the point of separation as far back as possible and thus to produce the minimum size of turbulent wake. This decreases the pressure drag, but by making the body longer so as to promote a gradual increase in pressure, we increase the friction drag. The optimum amount of streamlining, then, is that for which the sum of the friction and pressure drag is a minimum. Quite evidently, from what we have learned, attention in streamlining must be given to the rear end, or downstream part, of a body as well as to the front. The shape of the forebody is important principally to the extent that it governs the location of the separation point(s) on the afterbody. A rounded nose produces the least disturbance in the streamlines, and is therefore the best for incompressible or compressible flow at subsonic velocities.[8] This is illustrated in Fig. 9.12, where we compare flow about a blunt-nosed motor vehicle with that about a round-nosed vehicle.

Efforts to reduce the drag force on automobiles and trucks, and thereby improve their fuel efficiency, have strongly influenced body design. Examples of historical body shapes and the resulting drag coefficients are given in Table 9.1.

[8] For supersonic flow, a sharp-nosed body has less drag than a round-nosed body (Fig. 9.27).

TABLE 9.1 **The effects of automobile body shapes on aerodynamic drag**

	Body shape	C_D based on frontal area
1920		0.80
1940–45		0.54–0.58
1968–69		0.36–0.38
1990–92		0.29–0.30
Tractor-trailer		0.75–0.95
With rounded cab, fairing		0.55–0.75

SAMPLE PROBLEM 9.4 Using the data of Sample Prob. 9.2 determine the total drag exerted by the air on the van. Assume that $C_D \approx 0.45$ (see Fig. 9.12).

Solution

$$F_D = C_D \rho \frac{V^2}{2} A = 0.45(0.00242)\frac{(88)^2}{2}(8 \times 10) = 337 \text{ lb} \qquad \textbf{\textit{ANS}}$$

Thus the pressure drag = 337 − 25 = 312 lb; in this case the pressure drag is responsible for about 93% of the total drag, while the friction drag comprises only 7% of the total.

SAMPLE PROBLEM 9.5 Find the "free-fall" velocity of an 8.5-in-diameter sphere (bowling ball) weighing 16 lb falling through these fluids under the action of gravity: (*a*) the standard atmosphere at sea level; (*b*) the standard atmosphere at 10,000 ft elevation; (*c*) water at 60°F; (*d*) crude oil ($s = 0.925$) at 60°F.

Solution

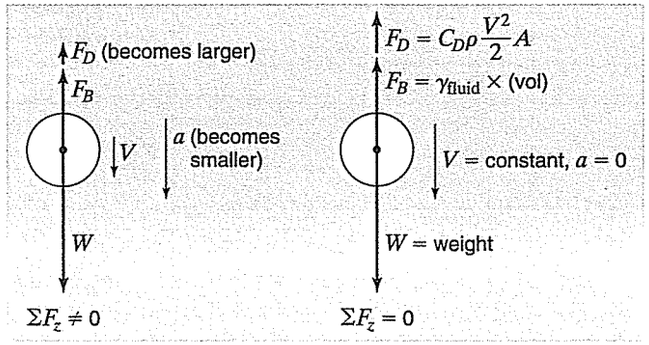


Figure S9.5

When first released, the sphere will accelerate (Fig. S9.5*a*), because the forces acting on it are out of balance. This acceleration results in an increase in velocity, which causes an increase in the drag force. After a while, the drag force will increase to the point where the forces acting on the sphere are in balance, as indicated in Fig. S9.5*b*. When that point is reached, the sphere will attain a constant or terminal (free-fall) velocity. Thus for free-fall conditions,

$$\sum F_x = W - F_B - F_D = \text{mass} \times \text{acceleration} = 0$$

where W is the weight, F_B is the buoyant force, and F_D is the drag force. The buoyant force is equal to the unit weight of the fluid multiplied by the volume ($\pi D^3/6 = 0.186\ \text{ft}^3$) of the sphere. The basic data are approximately:

Fluid	γ, lb/ft^3	ρ, slug/ft^3	$\nu = \mu/\rho$, ft^2/sec	F_B, lb
Air (sea level)	0.0765	0.00238	1.57×10^{-4}	0.0142
Air (10,000 ft)	0.0564	0.00176	2.01×10^{-4}	0.0105
Water, 60°F	62.4	1.94	1.22×10^{-5}	11.6
Oil, 60°F	57.7	1.79	0.001	10.7

The detailed analysis for the sphere falling through the standard sea-level atmosphere is

$$16 - 0.0142 - C_D\rho\frac{V^2}{2}A = 0$$

where $\rho = 0.00238\ \text{slugs/ft}^3$, $A = \frac{1}{4}\pi(8.5/12)^2 = 0.394\ \text{ft}^2$

or $$15.986 = C_D(0.00238)\frac{V^2}{2}(0.394) = 0.00047C_DV^2$$

A trial-and-error solution is required. After some trials, let $C_D = 0.2$; then $V = 412$ fps, and

$$\mathbf{R} = \frac{DV}{\nu} = \frac{(8.5/12)412}{1.57 \times 10^{-4}} = 1.86 \times 10^6$$

The values of C_D and **R** check (Fig. 9.10); hence

$$C_D = 0.2 \quad \text{and} \quad V = 412\ \text{fps} \qquad \textit{ANS}$$

Following a similar procedure for the other three cases gives these free-fall velocities:

Fluid	C_D	**R**	V_{fall}	
Standard atmosphere at 10,000 ft	0.20	1.69×10^6	480 fps	*ANS*
Water at 60°F	0.19	453,000	7.4 fps*	*ANS*
Crude oil ($s = 0.925$) at 60°F	0.39	4,390	6.2 fps	*ANS*

*In this instance the Reynolds number is 453,000, which, for the case of a sphere, generally indicates a turbulent boundary layer (Fig. 9.10). This is very close to the point where the boundary layer changes from laminar to turbulent. If the water had been at a somewhat lower temperature, and hence more viscous, a laminar boundary layer might have developed, in which case the free-fall velocity would have been only about 5.2 fps.

EXERCISES

9.7.1 An 0.3-in-diameter steel sphere ($s = 8.0$) is released in a large tank of oil ($s = 0.80$). The terminal velocity of the sphere is determined to be 2.5 fpm. Calculate the viscosity of the oil.

9.7.2 A 5-mm-diameter steel sphere ($s = 8.0$) is released in a large tank of oil ($s = 0.80$). The terminal velocity of the sphere is determined to be 0.7 m/min. Calculate the viscosity of the oil.

9.7.3 For a 380-mm-diameter sphere, compute the drag from wind under sea-level conditions in a standard ICAO atmosphere. Plot drag versus velocity for 0, 10, 20, and 30 m/s.

9.7.4 A poorly streamlined automobile has a body form corresponding roughly to the 1:0.75 oblate ellipsoid of Fig. 9.10, while a well-streamlined car has a body approximating the streamlined body in the same figure, each with a diameter of 5 ft. If the velocity is 50 mph through standard air at sea level (Appendix A, Table A.3), find the horsepower required to overcome air resistance in each case.

9.7.5 For the streamlined train in Exer. 9.4.5, calculate the pressure drag. As a rough approximation, assume that the nose and tail of the train are of the shape of the two halves of the prolate ellipsoid of Fig. 9.10, of 8.5 ft diameter. Find the drag on the ellipsoid (pressure drag on the train), and compare this with the skin-friction drag on the train determined earlier.

9.7.6 A hemispherical shell with its concave side upstream has a drag coefficient of approximately 1.33 if $\mathbf{R} > 10^3$. If the total load is 250 lb, find the diameter of a hemispherical parachute needed to provide a fall velocity no greater than that caused by jumping from a height of 10 ft. Assume standard air at sea level.

9.7.7 A hemispherical shell with its concave side upstream has a drag coefficient of approximately 1.33 if $\mathbf{R} > 10^3$. If the total load is 1000 N, find the diameter of a hemispherical parachute needed to provide fall velocity no greater than that caused by jumping from a height of 3 m. Assume standard air at sea level.

9.7.8 Assuming $C_D = 1.25$ and a speed of 30 m/s, find the drag force exerted by a 3.5-m-diameter braking parachute at sea level? At what speed will the same braking force be exerted by this parachute at an elevation of 2000 m. Assume C_D remains constant.

9.8 DRAG ON TWO-DIMENSIONAL BODIES (INCOMPRESSIBLE FLOW)

Two-dimensional bodies are also subject to friction and pressure drag. However, the flow about a two-dimensional body exhibits some peculiar properties that are not ordinarily found in the three-dimensional case of flow around a sphere. For example, with Reynolds numbers less than 1, the flow around a cylinder is completely viscous, and the drag coefficient is given by the straight-line part of the curve at the left of Fig. 9.13. As the Reynolds number increases from 2 to about 30, the boundary layer separates symmetrically from the two sides of the cylinder and two weak, symmetrical, standing large eddies are formed. The equilibrium of the standing eddies is maintained by the flow from the separated boundary layer, and if the cylinder is of finite length, then the eddies increase in length with increase in velocity in order to dissipate their rotational energy to the free-streaming fluid.

At some limiting Reynolds number, usually about 60, depending on the shape of the cylinder (not necessarily circular), the width of the confining channel, and the turbulence in the stream, the eddies break off, having become too long to hang on, and wash downstream. This gives rise to the beginning of the so-called ***Kármán vortex street.*** Above this critical **R**, and visibly up to a value of about 120, the vortices are shed first from one side of the cylinder and then from the other. The result is a staggered double row of vortices in the wake of the object spaced approximately as shown in Fig. 9.14. This alternating shedding of vortices and the accompanying forces give rise to the phenomenon of aerodynamic instability, of much importance in the design of tall smoke stacks and suspension bridges. We also understand it to account for the "singing" of wind

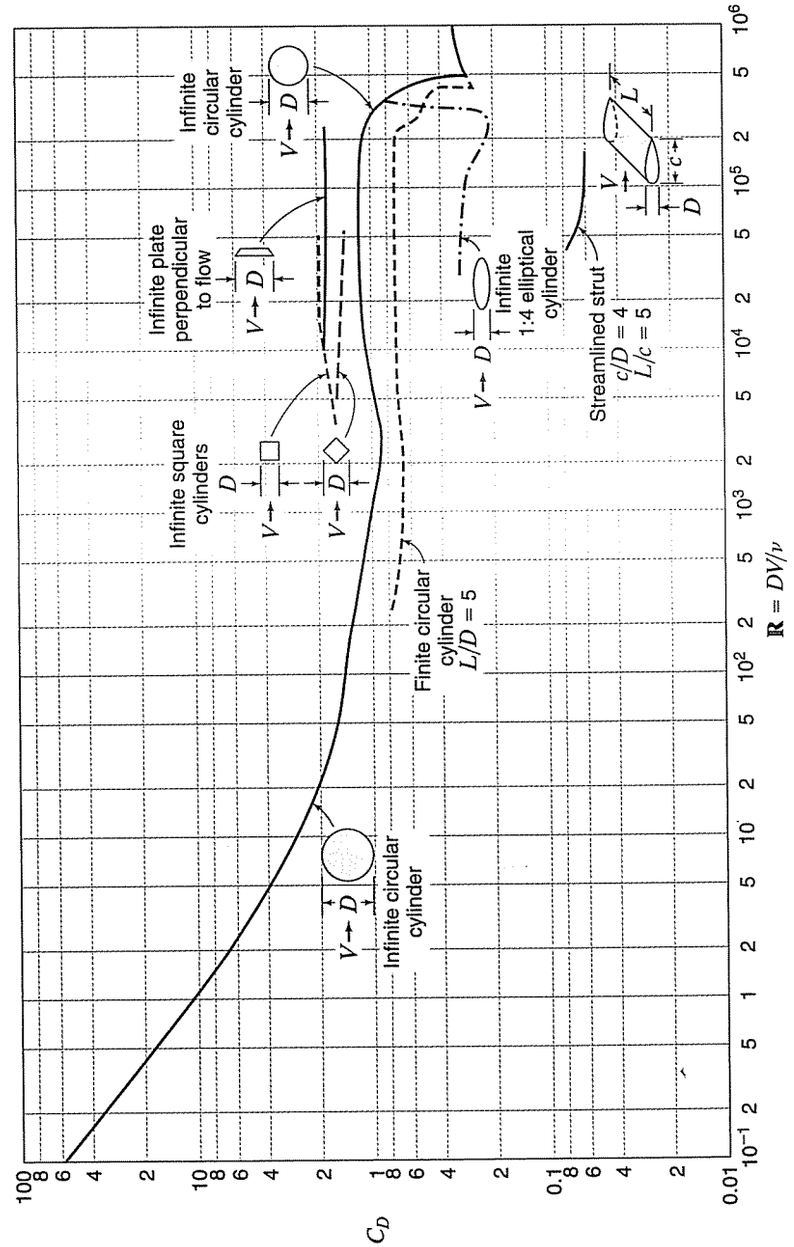


Figure 9.13
Drag coefficients for two-dimensional bodies.

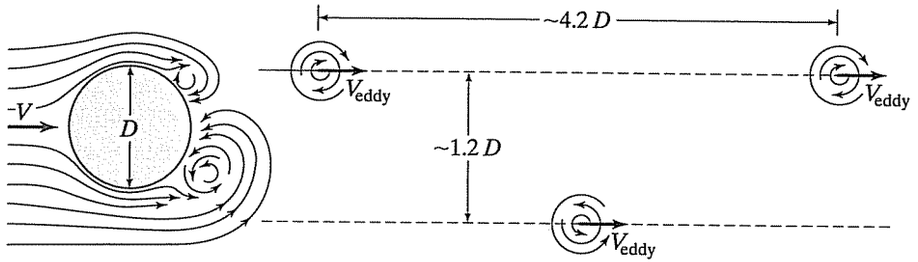


Figure 9.14
The Kármán vortex street following a cylinder. Velocity of eddies $V_{eddy} < V$, velocity of the fluid.

blowing across wires. The frequency at which the vortices are shed was given by G. I. Taylor and substantiated by Lord Rayleigh to be about

$$\text{Frequency} \approx 0.20 \frac{V}{D}\left(1 - \frac{20}{\mathbf{R}}\right) \tag{9.34}$$

A serious condition can result if the natural frequency of vibration of the structure is equal or close to the frequency of the shedding of the vortices.

For Reynolds numbers above 120 or so it is difficult to perceive the vortex street, but the eddies continue to be shed alternately, from each side up to a value of $\mathbf{R}$ of about 10,000. Beyond this, the viscous forces become negligible, and we can not say how the eddies form and leave the cylinder. As in the case of the sphere, the boundary layer for a circular cylinder becomes turbulent at a value of $\mathbf{R}$ of about 350,000. We can see the corresponding sharp drop in C_D in Fig. 9.13.

Values of C_D for various other two-dimensional shapes are given in Fig. 9.13. From the curve for the finite cylinder, we can see that the resistance is decreased if three-dimensional flow can take place around the ends. This decrease in C_D occurs because the vortices can extend laterally into the flow field and permit dissipation of energy over a larger region.

SAMPLE PROBLEM 9.6 What frequency of oscillation is produced by a 15 m/s wind at −20°C blowing across a 2-mm-diameter telephone wire at sea level?

Solution

Table A.2 at −20°C: $\nu_{air} = 1.15 \times 10^{-5}\ \text{m}^2/\text{s}$

$$\mathbf{R} = \frac{DV}{\nu} = \frac{2 \times 10^{-3}(15)}{1.15 \times 10^{-5}} = 2610$$

Eq. (9.34): $$\text{frequency} = 0.20 \frac{15}{2 \times 10^{-3}}\left(1 - \frac{20}{2610}\right) = 1489\ \text{Hz} \qquad \textit{ANS}$$

EXERCISES

9.8.1 What is the bending moment at the base of a vertical 0.3-in-diameter cylindrical radio antenna 6 ft tall on an automobile traveling at 80 mph through standard air at sea level?

9.8.2 A 70-mph wind blows across an 0.1-in-diameter wire. Find the frequency of oscillation that results in standard atmosphere: (*a*) at sea level; (*b*) at 10,000 ft elevation.

9.9 LIFT AND CIRCULATION

At the start of this chapter, we briefly mentioned the lift as a force that acts on an immersed body normal to the relative motion between the fluid and the body. The most commonly observed example of lift is that of the airplane wing supported in the air by this force. The elementary explanation for such a lift force is that the air velocity over the top of the wing is faster than the mean velocity, while that along the underside is slower than the mean (Fig. 9.15*a*). The Bernoulli theorem (Secs. 5.2, 5.6) then shows a lower pressure on the top and a higher pressure on the bottom (Fig. 9.15*b*), resulting in a net upward lift.

We can explain the increased velocity over the top of the wing of Fig. 9.15*a* and the decreased velocity around the bottom of the wing by noting that a ***circulation*** (Sec. 14.3) is induced as the wing moves relative to the flow field. The strength of the circulation depends, in the real case, on the shape of the wing and its velocity and orientation with respect to the flow field. We present a schematic diagram of the situation in Fig. 9.16.

For years many investigators have studied exhaustively the relationship between lift and circulation. An understanding of this relationship is essential to the analysis of various aerodynamic and hydrodynamic problems. To understand the theory of lift, we shall consider the flow of an ideal fluid past a cylinder and assume that a circulation about the cylinder is imposed on the flow. First, though, let us consider the velocity field surrounding a free vortex (Fig. 9.17).

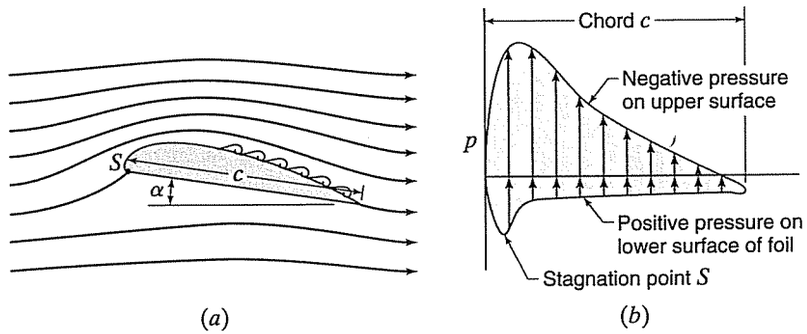


Figure 9.15
(*a*) Streamlines and (*b*) pressure distribution about a cambered airfoil, at an angle of attack of $\alpha = 8.6°$.

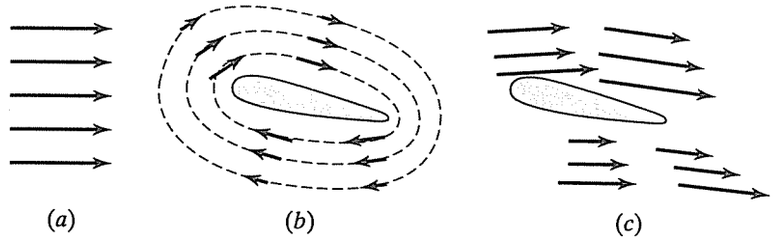


Figure 9.16
Schematic superposition of circulation on a uniform rectilinear flow field. (*a*) Uniform rectilinear flow field. (*b*) Circulation. (*c*) Net effect (sums of velocity vectors).

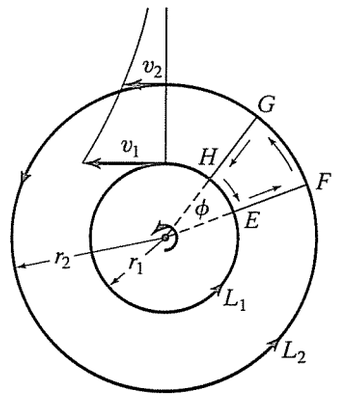


Figure 9.17
Circulation about a free vortex center.

The equation for this field was given in Sec. 5.18 as $vr = C$, a constant. The circulation Γ (gamma) can be readily computed by application of Eq. (14.9) if we choose the closed path as the circular streamline L_1 concentric with the center of the vortex. The velocity v_1 is constant around this path and tangent to it ($\cos\beta = 1$), and the line integral of dL is simply the circumference of the circle. Applying the same treatment to another concentric circle L_2, we get

$$\Gamma = v\oint_L dL = v_1(2\pi r_1) = v_2(2\pi r_2)$$

But, from the free-vortex velocity field, $v_1 r_1 = v_2 r_2 = C$, and hence

$$\Gamma = 2\pi C = 2\pi v r \tag{9.35}$$

which demonstrates that the circulation around two different curves, each completely enclosing the vortex center, is the same. We can more rigorously prove that the circulation around *any path enclosing the vortex center* is given by $\Gamma = 2\pi C$. We see that the circulation depends only on the vortex constant C, which we call the ***strength*** of the vortex.

A corollary states that the circulation around a path *not enclosing* the vortex center is zero. Let us take the path *EFGH* of Fig. 9.17. Along the two radial lines *EF* and *GH*, $\cos\beta = 0$, while along the two circular arc segments, $\Gamma_{FG} = v_2\phi r_2$ and $\Gamma_{HE} = -v_1\phi r_1$, resulting in a net circulation of $\phi(v_2 r_2 - v_1 r_1) = 0$.

EXERCISE

9.9.1 Estimate the lift per unit length of span if the mean velocity along the top of a wing having a 10-ft chord is 100 mph and that along the bottom of the wing is 80 mph when the wing moves at 85 mph through still air ($\gamma = 0.072$ lb/ft^3).

9.10 IDEAL FLOW ABOUT A CYLINDER

Let us first consider uniform flow of an ideal fluid about a cylinder that is infinitely long. From classical hydrodynamics,[9] it has been shown that with steady flow of uniform velocity U (Fig. 9.18*a*), the clockwise velocity v_{t_1} at the periphery ($r = R$) of the cylinder is given by

$$v_{t_1} = 2U\sin\theta \tag{9.36}$$

The pressure distribution on the cylinder may be computed by writing the Bernoulli theorem between a point at infinity in the free-streaming fluid and a point on the cylinder wall. Since the pressure distribution is completely symmetrical about the cylinder, there is no net lift or drag for this ideal case.

[9] We can obtain Eq. (9.36) by noting that $v_t = -\partial\psi/\partial r$ (by definition, Sec. 14.4) with v_t positive counterclockwise, where ψ is given by Eq. (14.19), in which $y = r\sin\theta$ and $m = 2\pi UR^2$ (Sec. 14.5).

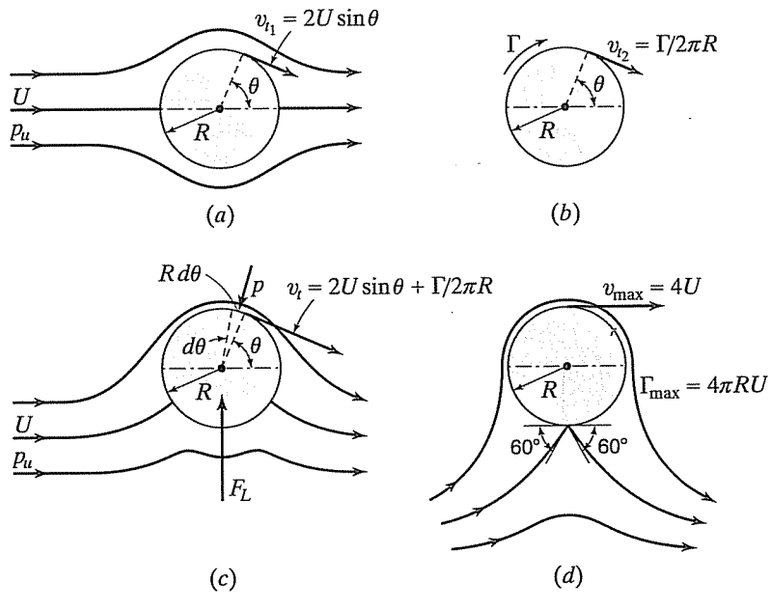


Figure 9.18
Circulation and lift from unsymmetrical flow about a cylinder.

Putting this uniform flow aside for the moment, we next suppose a circulatory flow about the cylinder (Fig. 9.18*b*). Adopting the positive clockwise direction of circulation Γ, from Eq. (9.35), the peripheral clockwise velocity v_{t_2} on the surface of the cylinder due to circulation may be expressed as

$$v_{t_2} = \frac{\Gamma}{2\pi R} \tag{9.37}$$

where R is the radius of the cylinder. Thus in the flow field outside the cylinder, where $r > R$, $v_{t_2} = \Gamma/2\pi r$. This velocity distribution produces a pressure variation that is radially symmetrical, in accordance with the free-vortex theory. We see that the solid cylinder has replaced the vortex center in the circulation theory. If the reader wishes an explanation for the existence of the circulation, it may be supposed to arise from rotating the cylinder, which indeed may be demonstrated in a real fluid.

Next let us ***superpose*** the circulatory flow onto the uniform motion, to form the unsymmetrical flow of Fig. 9.18*c*. The clockwise velocity at the periphery is the sum of the two contributions, or

$$v_t = 2U\sin\theta + \frac{\Gamma}{2\pi R} \tag{9.38}$$

The general equation for the pressure p at any point on the circumference of the cylinder is obtained from the energy Eq. (5.29) as follows:

$$\frac{p_u}{\gamma} + \frac{U^2}{2g} = \frac{p}{\gamma} + \frac{v_t^2}{2g}$$

where p_u is the pressure at some distance away where the velocity is uniform. From these two equations,

$$p - p_u = \frac{\rho}{2}\left[U^2 - \left(2U\sin\theta + \frac{\Gamma}{2\pi R}\right)^2\right] \tag{9.39}$$

Because the elementary area per unit of length of the cylinder is $R\,d\theta$ and the lift dF_L on this elementary area is normal to the direction of U and equal to $R\,d\theta\sin\theta(p_u - p)$, the resulting total lift is

$$F_L = -B\int_0^{2\pi} (p - p_u)R\sin\theta\,d\theta$$

Substituting the expression for $p - p_u$ from Eq. (9.39) and integrating, this reduces to

$$F_L = \rho BU\Gamma \tag{9.40}$$

where F_L is the lift force and B is the length of the cylinder.

The existence of this transverse force on a rotating cylinder is known as the ***Magnus effect,*** after Heinrich G. Magnus (1802–1870), the German scientist who first observed it in 1852. Equation (9.40) is known as the ***Kutta-Joukowski theorem,*** after W. M. Kutta and N. E. Joukowski, who independently pioneered

the quantitative investigation of the lift force in 1902 and 1906, respectively. The great importance of this theorem is that it applies not only to the circular cylinder but to a cylinder of any shape, including in particular the lifting vane, or ***airfoil,*** as shown in Fig. 9.15.

It is clear from Fig. 9.18*c* that the stagnation points have shifted downward from the horizontal axis, but they are still symmetrical about the vertical axis. At the point of stagnation on the cylinder, v_t in Eq. (9.38) will be zero. Thus we have at the stagnation point

$$-2U\sin\theta = \frac{\Gamma}{2\pi R}$$

This shows that if we can measure the angle to the stagnation point and know the free-stream velocity, we can obtain the circulation from

$$\Gamma = -4\pi RU\sin\theta_0 \tag{9.41}$$

where θ_0 is the angle between the horizontal diameter and the stagnation point in Fig. 9.18*c*. Figure 9.18*c* illustrates a case where $\Gamma < 4\pi RU$, that is, where $|\sin\theta| < 1$. For the case of $\Gamma < 4\pi RU$, $\sin\theta = -1$, and the two stagnation points meet together at the bottom of the cylinder as shown in Fig. 9.18*d*. The two streamlines make angles of 60° with the tangent to the cylinder, and the maximum velocity in the flow for this case occurs at the top of the cylinder and is equal to

$$v_{\max} = 2U + \frac{4\pi RU}{2\pi R} = 2U + 2U = 4U$$

Thus, according to ideal-flow theory,[10] if the cylinder is rotated so that $v_t = 2U$ (that is, with $\omega = v_t/R = 2U/R$), the circulation thus produced will cause the stagnation point to occur at the bottom of the cylinder, as in Fig. 9.18*d*. If the cylinder is rotated at still greater speed, the stagnation point is removed entirely from the cylinder surface, and a ring of fluid is dragged around with the cylinder.

SAMPLE PROBLEM 9.7 A cylinder 4 ft in diameter and 25 ft long rotates at 90 rpm with its axis perpendicular to an airstream with a wind velocity of 120 fps (Fig. S9.7). The specific weight of the air is 0.0765 lb/ft^3. Assuming no slip between the cylinder and the circulatory flow, find (*a*) the value of the circulation: (*b*) the transverse or lift force; (*c*) the position of the stagnation points.

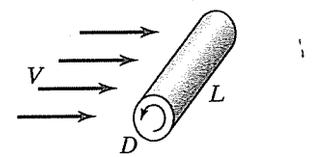


Figure S9.7

[10] In the case of a real fluid, because of viscosity, the required velocity to bring the stagnation point to the bottom of the cylinder is about twice that indicated by ideal-flow theory.

Solution

(*a*) Peripheral velocity: $v_t = \dfrac{2\pi Rn}{60} = 2\pi(2)\dfrac{90}{60} = 18.85$ fps

Eq. (9.35): $\Gamma = 2\pi R v_t = 2\pi(2)18.85 = 237$ ft²/sec ***ANS***

(*b*) Eq. (9.40): $F_L = \rho BU\Gamma = \dfrac{0.0765}{32.2}(25)120(237) = 1688$ lb ***ANS***

(*c*) Eq. (9.41): $\sin\theta_0 = -\dfrac{\Gamma}{4\pi RU} = -\dfrac{237}{4\pi 2 \times 120} = -0.0785$

Therefore $\theta_0 = 184.5°, 355.5°$ ***ANS***

Note: The real circulation actually produced by surface drag of the rotating cylinder would be only about one-half of that obtained above for the no-slip assumption.

EXERCISES

9.10.1 Imagine that a circulation of 25 ft²/sec is superimposed around a 1-ft-diameter cylinder immersed in 60°F water flowing at 10 fps perpendicular to the cylinder axis. Find the location of the stagnation points, and the lift on a 20-ft length of cylinder.

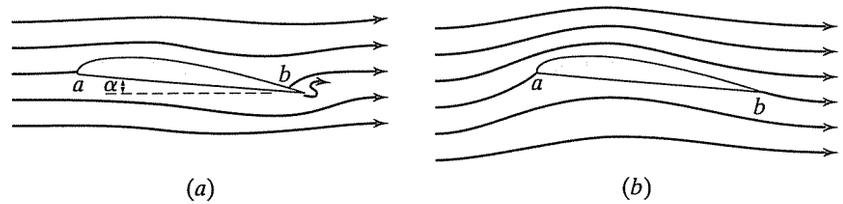
9.10.2 Imagine that a circulation of 2.4 m²/s is superimposed around a 300-mm-diameter cylinder immersed in 15°C water flowing at 3.5 m/s perpendicular to the cylinder axis. Find the location of the stagnation points, and the lift on a 10-m length of cylinder.

9.11 LIFT OF AN AIRFOIL

We might ask why so much attention is given to the flow about a cylinder when it is obvious that there are few practical applications of the lift on a cylinder.[11] The answer is that one of the most remarkable applications of mathematics to engineering is ***conformal transformation,***[12] by which the flow about one body can be mapped into the flow about a body of different (though mathematically

[11] In the early 1920s A. Flettner in Germany developed the "rotorship," which substituted motor-driven cylindrical rotors for sails. The ship was then driven by the Magnus effect but still required wind. A few trans-Atlantic crossings were made, but the rotorship was ultimately found to be uneconomical. See A. Flettner, *The Story of the Rotor,* Willhoft, New York, 1926.

[12] For an excellent discussion of conformal transformation, see H. R. Vallentine, *Applied Hydrodynamics,* 2d ed., Butterworth, London, and Plenum, New York, 1967.



(*a*) (*b*)

Figure 9.19
Adjustment of stagnation points to avoid infinite velocity at trailing edge.

related) shape. Certain quantities, notably the circulation and relative position of the stagnation points, remain unchanged under the mapping. The importance of the circular cylinder, then, is that it can be mapped into a perfectly workable airfoil by the so-called ***Joukowski transformation.*** We determine the position of the stagnation points from the physical requirements of the flow about the airfoil, and these stagnation points, mapped back onto the cylinder, determine the circulation, by Eq. (9.41), and the lift, by Eq. (9.40).[13]

Let us examine the airfoil of Fig. 9.19. As fluid flows past the foil, there will be a tendency for stagnation points to form at the points of the foil, corresponding to the 0° and 180° points of the corresponding cylinder (Fig. 9.18*a*). Just where these points occur on the foil depends on the ***angle of attack*** α, or the attitude of the foil with respect to the oncoming flow, as shown in the figure. We shall assume a positive angle of attack in Fig. 9.19*a*, with corresponding initial stagnation points *a* and *b*. While the location of these stagnation points involves no difficulty in the case of the ideal fluid, we see at once that the condition at the trailing edge—with the air from the underside trying to flow around the sharp cusp of the foil—becomes a point of violent separation in real fluid flow.

This condition lasts no more than an instant, however, for stagnation point *b* is soon swept back to the trailing edge of the foil (Fig. 9.19*b*), where it stays. This stable position of the point *b* is necessary, according to the so-called ***stagnation hypothesis*** of N. Joukowski, in order to avoid an infinite velocity around the sharp cusp of the foil. Now this shift in the rear stagnation point of the foil corresponds to a shift in the rear stagnation point of the related cylinder to a negative angle, somewhat as shown in Fig. 9.18*c*. Vertical symmetry of the flow about the cylinder requires that the forward stagnation point move downward by the same angle. This in turn maps a new location of the forward stagnation point on the airfoil, and such a shift also takes place in the real flow. We see, then, that a circulation has become established about the airfoil, the magnitude of which is determined by the location of the stagnation points on the corresponding cylinder. We can then determine the lift analytically by Eq. (9.40).

[13] Note that although the Joukowski profile is a workable lifting vane, the modern airfoil has undergone many modifications to improve its performance for various special purposes. The mapping theory described here applies exactly to the Joukowski foil and in principle to any lifting vane.

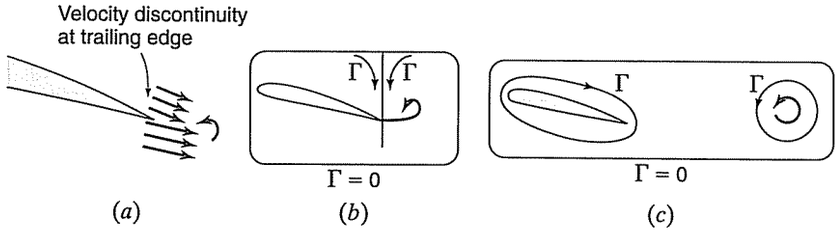


Figure 9.20
Life history of the starting vortex.

Although the Joukowski hypothesis appears perfectly reasonable, we must investigate whether or not nature will actually perform this adjustment of the stagnation point to the cusp of the airfoil profile. Our acceptance of this hypothesis is complicated by the perfectly valid theorem of the Scottish physicist and mathematician W. Thomson (Baron Kelvin) 1824–1907, which states that "the circulation around a closed curve in the fluid does not change with time if one moves with the fluid." How is a circulation created around the airfoil where none existed before? The answer was first suggested by Prandtl, and has been well substantiated with photographs. He showed that the initial separation point at the cusp caused a ***starting vortex*** to form, as shown in Fig. 9.20*a*. In order to satisfy Thomson's theorem, an equal and opposite circulation must automatically be generated around the foil (Fig. 9.20*b*). After this circulation has been established, the starting vortex breaks off and is left behind as the airplane moves forward, but just to satisfy Thomson's theorem, the starting vortex keeps whirling around (Fig. 9.20*c*) until it dies out from viscous effects. The net circulation around a curve including the profile and this vortex is still zero. When the airfoil comes to a stop or changes its angle of attack, new vortices are formed to effect the necessary change in circulation.

9.12 INDUCED DRAG ON AIRFOIL OF FINITE LENGTH

We have limited the discussion of lift so far to strictly two-dimensional flow. When the foil or lifting vane is of finite length in a free fluid, however, there are end conditions that affect both the lift and the drag. Because the pressure on the underside of the vane is greater than that on the upper side, fluid will escape around the ends of the vane, and there will be a general flow outward from the center to the ends along the bottom of the vane and inward from the ends to the center along the top. The movement of the fluid upward around the ends of the vane results in small ***tip vortices,*** which are cast off from the wing tips. In theory, Thomson's theorem still holds, for the tip vortices are of equal and opposite magnitude. If the circulation is computed about a closed path passing along the axis of the airfoil and along the axes of the tip and starting vortices, as shown in Fig. 9.21, it will still add up to zero. Practically, of course, the circulation about the foil continues to exist, but the tip and starting vortices soon die out from viscous resistance.

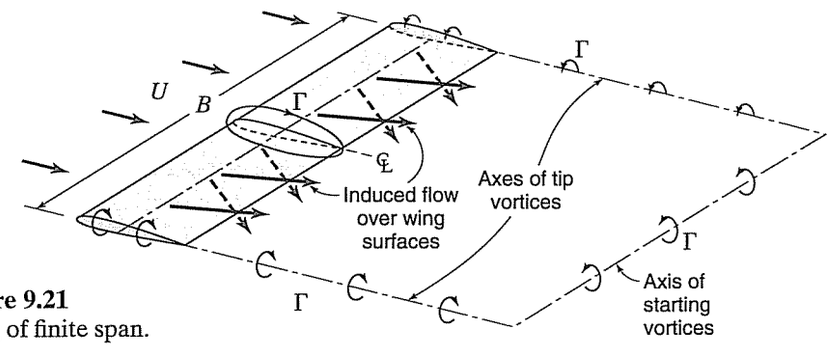


Figure 9.21
Wing of finite span.

The closed path consisting of the finite wing, the tip vortices, and the starting vortices of Fig. 9.21 constitutes a large vortex ring, inside of which there is a downward velocity induced by the vortices. Prandtl showed this induced, or ***downwash,*** velocity U_i to be a constant if the wing is so constructed as to produce an elliptical distribution of lift over a given span. The downwash changes the direction of the flow in the vicinity of the foil from U to U_0, thus *decreasing the effective angle of attack* from α to α_0. The decrease in the effective angle of attack $\alpha_i = \alpha - \alpha_0 = \arctan(U_i/U)$, as shown in Fig. 9.22.

We can analyze the wing on the basis of a foil of infinite length set in a stream of uniform velocity U_0, at angle of attack α_0. The lift F_{L0} generated from the circulation about the infinite foil must be normal to U_0. We see that this force resolves into two components: the true lift F_L normal to U and a component parallel to U called the ***induced drag*** F_{Di}. In conformity with our other drag terms, we represent the induced drag in the standard form

$$F_{Di} = C_{Di}\rho\frac{V^2}{2}A \tag{9.42}$$

We now need to distinguish between the two- and three-dimensional cases of drag. We will lump the skin friction and pressure drag discussed earlier in this

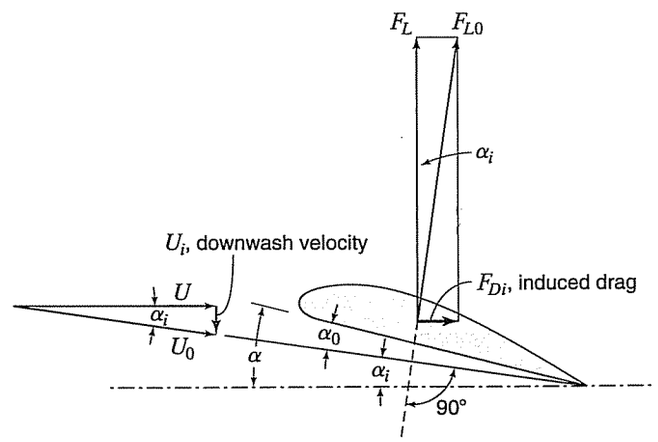


Figure 9.22
Definition sketch for induced drag.

chapter into the ***profile drag*** F_{D0}, which includes all drag forces acting on the profile of infinite length. The total drag on the finite span is then the sum of the profile and induced drags:

$$F_D = F_{D0} + F_{Di} \tag{9.43}$$

Because the angle α_i is small,

$$U_0 \approx U, \quad F_{L0} \approx F_L, \quad F_{Di} \approx \alpha_i F_L$$

Note, at this point, that in addition to expressing the lift force by Eq. (9.40), it is convenient to express it as

$$F_L = C_L \rho \frac{V^2}{2} A \tag{9.44}$$

where C_L is the lift coefficient, whose value depends primarily on the angle of attack and the cross-sectional shape of the airfoil, and A is the projected area of the airfoil or body normal to the lift vector.

The computations for the elliptical distribution of lift are too complex to include here, but they result in the simple relation

$$\frac{U_i}{U} = \alpha_i \text{ (radians)} = \frac{C_L}{\pi B^2/A} \tag{9.45}$$

where B is the span of the airfoil and A is its plane area. The quantity $B^2/A = B/c$, where c is the mean chord length, is referred to as the ***aspect ratio.***

From Eqs. (9.42)–(9.44), together with the above expression for F_{Di}, we have

$$C_{Di} = \frac{C_L^2}{\pi B^2/A} \tag{9.46}$$

Dividing Eq. (9.43) by $\rho V^2 A/2$ and substituting equations similar to Eq. (9.42) and making use of Eq. (9.46) gives for the coefficient of total drag on a foil of finite length,

$$C_D = C_{D0} + C_{Di} = C_{D0} + \frac{C_L^2}{\pi B^2/A} \tag{9.47}$$

As we would expect, we see that C_{Di} depends on the lift coefficient, i.e., the angle of attack α_0 and the aspect ratio B^2/A. For zero lift or infinite aspect ratio, the induced drag would be zero. These equations are important in comparing data for an airfoil tested at one aspect ratio with data for another foil at a different aspect ratio.

The explanation of how the induced drag, occurring as it does in the ideal-fluid theory, fits into d'Alembert's paradox—which states that there is no drag on a body in ideal flow—is that the work done against the flow by the induced drag is conserved in the kinetic energy of the tip vortices cast from the ends of the foil. In a real fluid, we can frequently see evidence of the tip vortices in the form of ***vapor trails*** extending for miles across the sky. The decreased temperature caused by the decreased pressure at the center of the vortex causes condensation of the moisture in the air.

EXERCISE

9.12.1 A wing with a 15-m span and a 45-m^2 "planform" area moves horizontally through the standard atmosphere at 6000 m with a velocity of 750 km/hr. If the wing supports 180 kN, find (*a*) the required value of the lift coefficient; (*b*) the downwash velocity, assuming an elliptical distribution of lift over the span; (*c*) the induced drag.

9.13 LIFT AND DRAG DIAGRAMS

Experimenters have obtained a wealth of data on the lift and drag of various airfoils from wind tunnel tests. The results of such tests can be presented graphically as plots of the lift and drag coefficients versus the angle of attack. Because the efficiency of the airfoil is measured by the ratio of lift to drag, the value of C_L/C_D is generally plotted also. These three curves can be combined neatly into a single curve, suggested by Prandtl, called a ***polar diagram*** (Fig. 9.23).

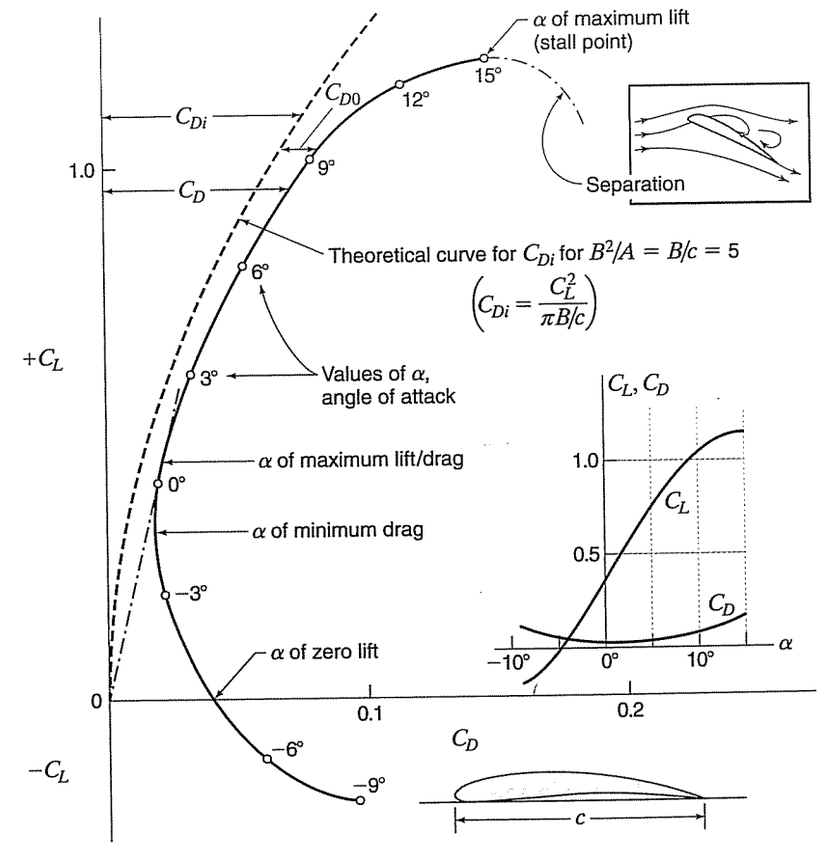


Figure 9.23
Polar diagram for wing of aspect ratio 5. (Curve from L. Prandtl and O. G. Tietjens, *Applied Hydro- and Aeromechanics,* p. 152, McGraw-Hill, New York, 1934.)

The coordinates of the polar diagram are the lift and drag coefficients, while the angles of attack are represented by different points along the curve. The ratio of lift to drag is the slope of the line from the origin to the curve at any point. Clearly, the maximum value of the ratio occurs when this line is tangent to the curve. The lift increases with the angle of attack up to the ***point of stall.*** Beyond this point the boundary layer along the upper surfaces of the foil separates and creates a deep turbulent wake.

The polar diagram is notably instructive with regard to the drag coefficient, consisting of the coefficients of profile and induced drag as shown in Eq. (9.47). The dashed line in Fig. 9.23 is the parabola of Eq. (9.46). For an aspect ratio of 5, as shown, the induced drag is a major part of the total drag. For larger aspect ratios the parabola remains closer to the vertical axis, and the total drag is correspondingly decreased.

The polar diagram of a Clark Y airfoil, rectangular in plan, 6-ft chord by 36-ft span, is shown in Fig. 9.24. Notice that we read the angle of attack from a geometric reference that has little meaning by itself. The important reference

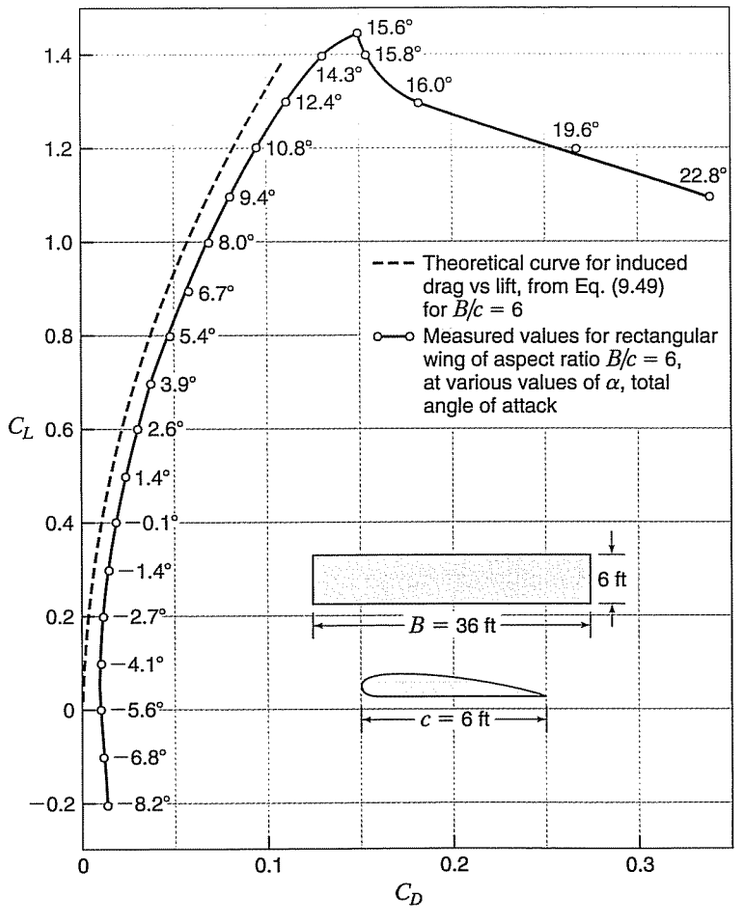


Figure 9.24
Polar diagram for rectangular Clark Y airfoil of 6-ft chord by 36-ft span.

angle is the angle of attack for zero lift, in this case −5.6°. In general, this is also the angle for minimum drag. We can show the lift coefficient to be theoretically given by

$$C_L = 2\pi\eta\alpha_0' \tag{9.48}$$

where α_0' is the angle of attack (for the airfoil of infinite span) measured in radians from the attitude of no lift, and η is a correction factor for frictional effects, having a value of about 0.9 for modern airfoil sections.

We recall from Sec. 9.12 that the induced-drag theory assumed an elliptical distribution of lift over the span of the finite airfoil. Such a distribution of lift is only an approximation, and for the rectangular airfoil the expressions for induced angle of attack and induced-drag coefficient given in Eqs. (9.45) and (9.46) must be corrected as

$$\alpha_i \text{ (radians)} = \frac{C_L}{\pi B/c}(1 + \tau) \tag{9.49}$$

and

$$C_{Di} = \frac{C_L^2}{\pi B/c}(1 + \sigma) \tag{9.50}$$

where τ and σ are correction factors given in Fig. 9.25. We can find information on airfoils of shapes other than rectangular and lift and drag on the fuselage, stabilizers, and control surfaces of aircraft[14] in the literature.

[14] H. Schlichting and E. Truckenbrodt, *Aerodynamics of the Airplane,* translated by Heinrich J. Ramm, McGraw-Hill International Book Co., New York, 1979.

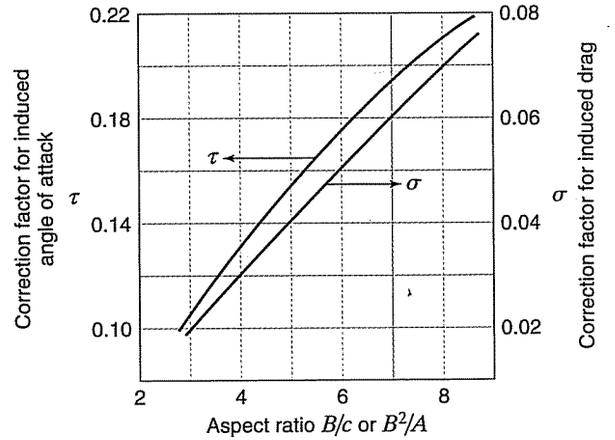


Figure 9.25
Correction factors for transforming rectangular airfoils from finite to infinite aspect ratio.

SAMPLE PROBLEM 9.8 For a rectangular Clark Y airfoil of 6-ft chord by 36-ft span, find (*a*) the value of the friction correction factor η if the angle of attack $\alpha = 5.4°$ when the wing is moving at 300 fps through standard atmosphere at altitude 10,000 ft. Find (*b*) the weight which the wing will carry and (*c*) the horsepower required to drive it.

Solution

(*a*) Fig. 9.24, with $\alpha = 5.4°$: $C_L = 0.8$ and $C_D = 0.047$. From Fig. 9.25, for $B/c = 36/6 = 6$: $\tau = 0.175$. From Eq. (9.49):

$$\alpha_i = \frac{0.8}{\pi(36/6)}(1 + 0.175) = 0.0499 \text{ rad} = 2.86°$$

From Fig. 9.22: $\alpha_0 = \alpha - \alpha_i = 5.40 - 2.86 = 2.54°$. Since the angle of zero lift is $-5.6°$,

$$\alpha_0' = 2.54 + 5.6 = 8.14° = 0.1421 \text{ rad}$$

From Eq. (9.48): $$\eta = \frac{C_L}{2\pi\alpha_0'} = \frac{0.8}{2\pi \times 0.1421} = 0.896 \qquad \textit{ANS}$$

(*b*) Table A.3 at 10,000 ft: $\rho = 0.001756$ slug/ft^3.

The wing will support a weight equal to the lift force:

Eq. (9.44): $$W = F_L = 0.8(0.001756)\frac{(300)^2}{2}(36)6 = 13{,}650 \text{ lb} \qquad \textit{ANS}$$

(*c*) From Eqs. (9.32) and (9.44): $$F_D = \left(\frac{C_D}{C_L}\right)F_L = \frac{0.047}{0.8} \times 13{,}650 = 802 \text{ lb}$$

$$\text{Horsepower required} = \frac{FV}{550} = \frac{802 \times 300}{550} = 438 \text{ hp} \qquad \textit{ANS}$$

EXERCISES

9.13.1 A sailplane including its load weighs 300 lb. It has a Clark Y section wing with a 3.5-ft chord by 21-ft span. Given that it has the same characteristics as the larger wing of the same aspect ratio shown in Fig. 9.24, find the angle of glide through standard air at 2000 ft that will produce the greatest horizontal distance range. Neglect air forces on the fuselage and the tail. (*Note:* The aspect ratio of 6 is chosen here to facilitate working the problem with the available data. Actually, the sailplane may be constructed with an aspect ratio of about twice this, in order to reduce the drag to a minimum.)

9.13.2 Solve Exer. 9.13.1 for a sailplane with a 1-m chord by 6-m span wing, flying at 600 m altitude with a total weight of 1800 N.

UNIVERSIDAD POLITECNICA SALESIANA BIBLIOTECA

9.13.3 A 1400-lb airplane has a Clark Y airfoil wing of 6-ft chord by 36-ft span, with polar diagram given in Fig. 9.24. Find (*a*) the speed required to get the plane off the ground; (*b*) the horsepower required; (*c*) the circulation about the wing; (*d*) the strength of the starting vortex. Assume standard air at sea level and the angle of attack for maximum ratio of lift to drag. Neglect aerodynamic forces on the fuselage and tail.

9.14 EFFECTS OF COMPRESSIBILITY ON DRAG AND LIFT

In Sec. 13.11 we shall see that when a body moves through a fluid at supersonic velocity a shock wave is formed. The wave pattern (Fig. 13.9) is the same whether the body is moving through the fluid or whether the fluid is moving past the body. Unlike subsonic flow (Fig. 4.12), in supersonic flow the streamlines in front of the body are unaffected, because the body is moving faster than the disturbance can be transmitted ahead. This is demonstrated in Fig. 13.10.

With most bodies, the drag coefficient tends to increase drastically at a Mach number of about 0.70. This happens because the body is encountering ***transonic flow*** phenomena, which means that at some place in the flow field supersonic flow is occurring. With a streamlined body, the highest velocity in the flow field occurs at some point such as *b* in Fig. 9.26 near the body and away from its nose. The local Mach number at *b* will reach unity when the free-stream Mach number at *a* has a value of perhaps only 0.7 or 0.8. Thus a shock wave will form at *b*. Through the shock wave, there is a sudden jump in pressure, which causes an adverse pressure gradient in the boundary layer, resulting in separation and an increase in drag. Drag coefficients for several bodies as a function of the free-stream Mach number are given in Fig. 9.27. The increased drag is caused not only by the separation effects; a substantial amount of energy is dissipated in the shock wave. Skin friction also contributes to the drag, and at Mach numbers above 2 or 3 heating in the boundary layer from skin friction may be an important phenomenon. As the value of the Mach number increases beyond about 2, for most bodies there is a drop in the value of the drag coefficient because of a shift in the point of separation.

Earlier we noted that, to streamline for subsonic flow, a rounded nose and a long, tapered afterbody generally result in minimum drag. *In supersonic flow the best nose form is a sharp point.* This tends to minimize the effect of the shock wave.

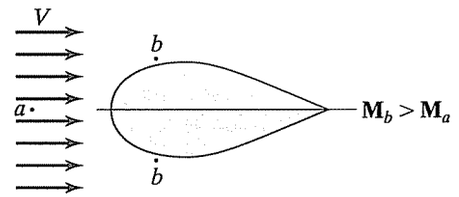


Figure 9.26
Local Mach number greater than free-stream Mach number. When $\mathbf{M}_b = 1.0$, $\mathbf{M}_a \approx 0.7$.

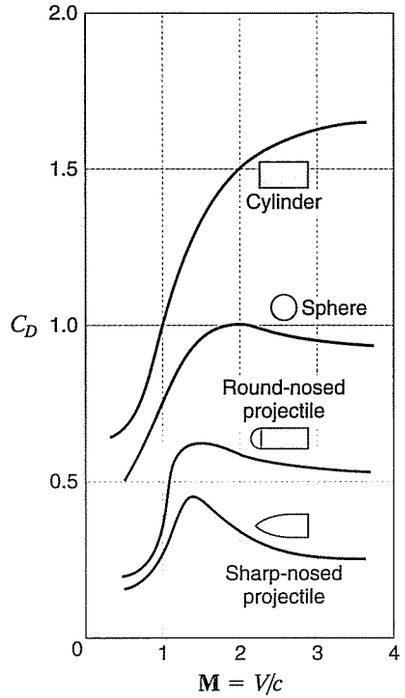


Figure 9.27
Drag coefficients as a function of Mach number.

With respect to lift, researchers have found that for Mach numbers less than about 0.7, the lift coefficient for compressible fluids can be estimated by dividing the lift coefficient for incompressible flow by $\sqrt{1 - \mathbf{M}^2}$, where $\mathbf{M}$ is the Mach number of the free stream.[15] The reduced pressures on the top of the airfoil are responsible for this trend.

At $\mathbf{M} \approx 0.8$, there is a rather abrupt drop in the lift coefficient, because the shock wave induced by the local supersonic flow creates high pressures on the top side of the airfoil which may result in ***shock stall.*** At somewhat higher values of the Mach number, a shock wave forms on the bottom of the airfoil, which tends to compensate for the preceding action, and there is an increase in the value of the lift coefficient. Although there have been tremendous advances in theory, the best way to predict the drag and lift of a particular airfoil is by conducting model tests in a wind tunnel.

EXERCISES

9.14.1 The blunt-nosed projectile (cylinder) of Fig. 9.27 has a diameter of 16 in and weighs 500 lb. Find its rate of deceleration when moving through standard sea-level atmosphere (*a*) horizontally at 1200 mph; (*b*) upward at an angle of 40° with the horizontal at a velocity of 1200 mph.

[15] This is called the ***Prandtl-Glauert rule.***

9.14.2 The blunt-nosed projectile (cylinder) of Fig. 9.27 has a diameter of 0.45 m and weighs 2500 N. Find its rate of deceleration when moving through standard sea-level atmosphere (*a*) horizontally at 450 m/s; (*b*) upward at an angle of 45° with the horizontal at a velocity of 450 m/s.

9.15 CONCLUDING REMARKS

We ought to mention some other aspects of drag. One of these is drag under conditions of ***supercavitation.*** This occurs when bodies move at high speeds through liquids. It is particularly common with blunt bodies. A large cavity forms behind the body, which alters the pressure distribution because the limiting minimum absolute pressure in the cavity is the vapor pressure. Such problems require special treatment; experimental data provide the best information.

Another interesting drag problem is that of an object moving at the interface of two fluids of different density. A good example of this is a ship or boat moving through water. In such cases energy is also expended in the generation of waves. The drag of the ship is caused primarily by both skin friction and wave action; hence in a model test both the Reynolds and Froude criteria ought to be satisfied, but this is usually not practical (see Sec. 7.4). The modeling procedure usually employed involves determining the total drag of the model by testing it at the prototype Froude number. The frictional drag of the model is then estimated by using boundary-layer theory, as presented in Secs. 9.3–9.5, and subtracted from the total drag to get an estimate of the drag on the model caused by wave action. This is then translated by model laws to an estimate of the wave-action drag on the prototype, to which is added the prototype friction drag as estimated by boundary-layer theory to give the total prototype drag.

SAMPLE PROBLEM 9.9 A 200-mm-diameter round-nosed projectile whose drag coefficient is shown in Fig. 9.27 travels at 600 m/s through the standard atmosphere at an altitude of 6000 m. Find the drag.

Solution
Table A.3 at 6000 m:

$$\rho = 0.660 \text{ kg/m}^3, \text{ and temperature} = -24°\text{C or } 273 - 24 = 249 \text{ K}$$

Table A.5 for air: $k = 1.40$ and $R \approx 287 \text{ m}^2/(\text{s}^2 \cdot \text{K})$.

Eq. (13.15) per Sec. 7.4: $c = \sqrt{kRT} = \sqrt{1.40(287)249} = 316 \text{ m/s}$

Eq. (7.10): $$\mathbf{M} = \frac{600}{316} = 1.897$$

Fig. 9.27 with $\mathbf{M} = 1.897$: $$C_D = 0.62$$

Eq. (9.32): $$F_D = 0.62(0.660)\frac{(600)^2}{2}\frac{\pi(0.20)^2}{4} = 2310 \text{ N}$$ ***ANS***

PROBLEMS

9.1 Given the general equation for a parabola, $u = ay^2 + by + c$, derive the dimensionless velocity distribution of Eq. (9.9).

9.2 Given the parabolic velocity distribution of Eq. (9.9), prove that $\alpha = 0.1333$ (in Eq. 9.5) and $\beta = 2.0$ (in Eq. 9.7).

9.3 Determine the shear stress at 150 mm and 300 mm back from the leading edge of the plate in Sample Prob. 9.1.

9.4 Find the shear stress and the thickness of the boundary layer (*a*) at the center and (*b*) at the trailing edge of a smooth, flat plate 10 ft wide and 2 ft long parallel to the flow, immersed in 60°F water flowing at an undisturbed velocity of 3 fps. Assume a laminar boundary layer over the whole plate. Also, (*c*) find the total friction drag on one side of the plate.

9.5 Derive Eq. (9.19) along the lines suggested in the text.

9.6 For the turbulent boundary layer, derive the value of $\alpha = 0.0972$ from the seventh-root law given in Eq. (9.20).

9.7 From the information given, derive Eq. (9.22).

9.8 Assume that the boundary layer of Prob. 9.4 is disturbed near the leading edge. Compute the corresponding quantities for the turbulent boundary layer covering the entire plate, and compare the results.

9.9 An 85-m-long streamlined train has 2.5-m-high sides and a 2.5-m-wide top. Compute the power required to overcome the skin-friction drag when the train is traveling at 40 m/s through the ICAO standard atmosphere at sea level, assuming the drag on the sides and top to be equal to that on one side of a flat plate 7.5 m wide and 85 m long.

9.10 A lifeguard determines the wind velocity 2 m above the beach to be 8 m/s. If one wishes to get out of the wind by lying down, what would be the velocity at (*a*) 0.15 m; (*b*) 0.3 m above beach level?

9.11 Verify that the two expressions of Eq. (9.28) are equal.

9.12 A 20-mm-diameter harpoon 1.8 m long, with a sharp tip, is launched at 6 m/s into 15°C water. Find (*a*) the friction drag; (*b*) the maximum thickness of the boundary layer.

9.13 An airplane wing with a chord length of 7 ft parallel to the flow moves through standard atmospheric air at an altitude of 15,000 ft with a speed of 300 mph (Fig. P9.13). Find (*a*) the critical roughness for a point one-tenth the chord length back from the leading edge; (*b*) the surface drag on a 25-ft-span section of this wing.

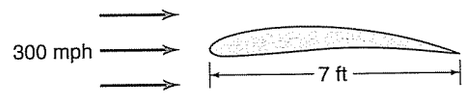


Figure P9.13

9.14 A flat plate 25 ft long and 1.5 ft wide is towed at 8 fps through a liquid (γ = 50 lb/ft^3, $\mu = 0.00026$ lb·sec/ft^2). Determine the total drag on the plate. Plot the thickness of the boundary layer and the local shear stress τ_0 as functions of x along the plate. Determine the area under the stress curve, and compare it with the previously computed value of the drag. Assume that the boundary layer changes from laminar to turbulent at a Reynolds number of 300,000.

9.15 For velocities of 0, 10, 20, 30, 40, 50, and 60 fps, make the necessary calculations to plot drag vs velocity for the plate and data of Prob. 9.14.

9.16 Determine the drag on the harpoon of Exer. 9.5.2 for velocities of 0, 10, 30, and 50 fps, through (*a*) 60°F water; (*b*) 60°F air at standard sea-level pressure. Calculate the length x_c of the laminar zone, and plot curves of drag versus velocity for each case.

9.17 Determine the drag on the harpoon of Prob. 9.12 for velocities of 0, 3, 9, and 15 m/s, through (*a*) 15°C water; (*b*) 15°C air at standard sea-level pressure. Calculate the length x_c of the laminar zone, and plot curves of drag versus velocity for each case.

9.18 What will be the terminal velocity of the sphere of Exer. 9.7.1 in 120°F water?

9.19 For an 18-in-diameter sphere, compute the drag from wind under sea-level conditions in a standard ICAO atmosphere. Plot drag versus velocity for 0, 25, 50, and 100 fps.

9.20 Repeat Prob. 9.19 for wind at a 10,000-ft elevation in a standard ICAO atmosphere.

9.21 A 1.5-ft-diameter metal ball weighing 150 lb is dropped into the ocean from a boat. Determine the maximum velocity of the ball as it falls through seawater with ρ = 2.0 slugs/ft^3 and $\mu = 3.3 \times 10^{-5}$ lb·sec/ft^2.

9.22 A 400-mm-diameter metal ball weighing 500 N is dropped into the ocean from a boat. Determine the maximum velocity of the ball as it falls through seawater with $\rho = 1030$ kg/m^3 and $\mu = 0.0016$ N·s/m^2.

9.23 A poorly-streamlined automobile has a body form corresponding roughly to the 1:0.75 oblate ellipsoid of Fig. 9.10, while a well-streamlined car has a body approximating the streamlined body in the same figure, each with a diameter of 1.5 m. If the velocity is 30 m/s through standard air at sea level (Appendix A, Table A.3), find the power in kW required to overcome air resistance in each case.

9.24 For the streamlined train in Prob. 9.9, calculate the pressure drag. As a rough approximation, assume that the nose and tail of the train are of the shape of the two halves of the prolate ellipsoid of Fig. 9.10, of 2.5 m diameter. Find the drag on the ellipsoid (pressure drag on the train), and compare this with the skin-friction drag on the train determined earlier.

9.25 A submerged 7.5-ft by 1.5-ft flat plate is dragged through 60°F water at 1.4 fps with the flat side normal to the direction of motion. (*a*) What is the approximate drag force? (*b*) How does this compare with the drag force of 0.1124 lb when pulled in the direction of the 7.5 ft length? [*Hint:* Assume that the drag coefficient for the plate of finite length is in the same ratio to the coefficient for the infinite plate as is the ratio of coefficients for the finite cylinder ($L/D = 5$) and the infinite cylinder of Fig. 9.13, for the same Reynolds number.]

9.26 A submerged 2.5-m by 0.5-m flat plate is dragged through 15°C water at 0.5 m/s with the flat side normal to the direction of motion. (*a*) What is the approximate drag force? (*b*) How does this compare with the drag force of 0.888 N when pulled in the direction of the 2.5 m length? [*Hint:* See Prob. 9.25.]

9.27 Compare the velocity of an 0.15-in-diameter drop of water falling through standard air at sea level, with that of a spherical bubble of air of the same size rising through water at the same temperature. Neglect the weight of the air.

9.28 Compare the velocity of a 3-mm-diameter drop of water falling through standard air at sea level, with that of a spherical bubble of air of the same size rising through water at the same temperature. Neglect the weight of the air.

9.29 Find the rate of fall of a spherical particle of sand ($s = 2.65$) in 15°C water if the diameter is (*a*) 0.1 mm; (*b*) 1.0 mm; (*c*) 10 mm. Express answers in mm/s.

9.30 A regulation football has a shape very similar to the prolate ellipsoid of Fig. 9.10, with a diameter of 6.78 in and a weight of 14.5 oz (Fig. P9.30). (*a*) Find the resistance when the ball is passed through still air (14.5 psia and 80°F) at a velocity of 45 fps. Neglect the effect of spin about the longitudinal axis. What is the deceleration at the beginning of the trajectory? (*b*) Find the percentage change in resistance if the air temperature is 30°F rather than 80°F, and the drag coefficient is unchanged.

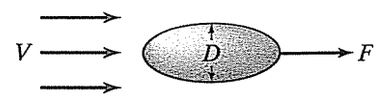


Figure P9.30

9.31 An eight-oar racing shell is traveling through 60°F water at a mean velocity of 12 mph. Each oar is 9 ft long, with a length of 6 ft from the oarlock to the center of the "spoon," which has a projected area of 120 in^2. Assume that all drag is due to the spoon, and that this drag is equal to that of a disk of equal area. (*a*) If the "stroke" is 32 per min, and if each oarsman sweeps a right-angle in one-fourth of his rowing cycle, what is the maximum thrust of the oars? It must be assumed that the shell moves at something less (say, 20%) than its mean velocity when the oars are in the water. (*b*) The maximum velocity occurs during the backstroke when the oarsmen shift their weight toward the stern. Why? (*c*) The oarsman on his backstroke moves at half the angular velocity of his forward stroke, while the shell moves at perhaps 10% above its mean velocity. Find the drag in 60°F air resulting from a "feathered" oar (turbulent boundary layer, Fig. 9.7) and an unfeathered one, in percentage of the forward thrust from part (*a*). What is the ratio of these two drag forces?

9.32 (*a*) Find the bending moment at the base of a 300-mm-diameter cylindrical light post 12 m high when it is subject to a uniform wind velocity of 30 m/s at standard sea level. Neglect end effects. (*b*) Discuss the consequences of considering the atmospheric boundary layer above the surface of the earth.

9.33 (*a*) Repeat Prob. 9.32(*a*) for the case where the pole has a uniform taper from 350 mm diameter at the base to 250 mm at its top. (*b*) Determine if the drag force would be larger or smaller if the pole had instead been tapered from 350 mm down to 150 mm. Why?

9.34 Beginning with the expression before Eq. (9.40), fill in the steps leading to Eq. (9.40). Take care to account for all changes in sign.

9.35 For the rotating cylinder in Sample Prob. 9.7, calculate the value of the lift coefficient. Assume the effective circulation is half the theoretical. Assuming the drag coefficient to be unchanged by the rotation of the cylinder, find the magnitude and direction of the total wind force on the rotor.

9.36 Assume the rotor in Sample Prob. 9.7 to be installed upright on a ship traveling due north at 20 knots (Fig. P9.36). The wind has an *absolute* velocity of 35 mph due east. If the drag coefficient of the cylinder is 1.0 and the "lines" of stagnation are separated by 120° on the rotor, find approximately the component of the total air force on the rotor in the direction of the ship's motion. Assume standard air.

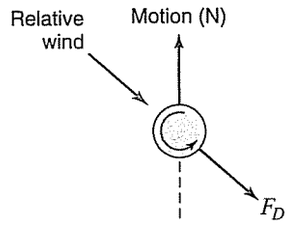


Figure P9.36

9.37 Consider a cylinder of radius a in a stream of ideal fluid in which the undisturbed velocity and pressure are U and p_u and the density is ρ. (*a*) Using Eq. (9.36) and the Bernoulli theorem, evaluate the dimensionless pressure coefficient $(p - p_u)/(\rho U^2/2)$ for every 10° over the surface of one quadrant of the cylinder, measuring the angle from the forward stagnation point. Plot the pressure coefficient to scale, plotting it radially from the cylinder surface. (*b*) What is the actual pressure in pounds per square inch on the surface of a 1-ft-diameter cylinder, 70° from the forward stagnation point, if the cylinder is 15 ft below the free surface of a stream of water at 60°F, flowing at 12 fps?

9.38 Consider a cylinder of radius a in a stream of ideal fluid in which the undisturbed velocity and pressure are U and p_u and the density is ρ. (*a*) Using Eq. (9.36) and the Bernoulli theorem, evaluate the dimensionless pressure coefficient $(p - p_u)/\rho U^2/2$ for every 10° over the surface of one quadrant of the cylinder, measuring the angle from the forward stagnation point. Plot the pressure coefficient to scale, plotting it radially from the cylinder surface. (*b*) What is the actual pressure in newtons per square meter on the surface of a 300-mm-diameter cylinder, 70° from the forward stagnation point, if the cylinder is 5 m below the free surface of a stream of water at 15°C, flowing at 4 m/s?

9.39 A double stagnation point is observed to occur on a 4-ft-diameter cylinder rotating in an air stream moving at 60 fps. Given standard atmospheric conditions at sea level, find (*a*) the lift force per foot length of the cylinder; (*b*) the lift coefficient.

9.40 A double stagnation point is observed to occur on a 1.2-m-diameter cylinder rotating in an air stream moving at 18 m/s. Given standard atmospheric conditions at sea level, find (*a*) the lift force per meter length of the cylinder; (*b*) the lift coefficient.

9.41 There have been many arguments over the validity and extent of the curve of a pitched baseball. According to tests (*Life,* July 27, 1953), a pitched baseball was found to rotate at 1400 rpm while traveling at 43 mph. The horizontal projection of the trajectory revealed a smooth curve of about 800 ft radius. If the ball had a circumference of 9 in and weighed 5 oz, find the transverse force required to produce the observed curvature. Assume the shape of the ball to be roughly that of a cylinder having a diameter equal to the ball's diameter and a length of two-thirds its diameter. Find the circulation that would be required to produce the transverse force. Compare this with that obtained by assuming no slip at the equator of the ball. Assume standard air at sea level.

9.42 Solve Prob. 9.41 for a speed of 69 km/h and a trajectory radius of 245 m, with a ball weighing 1.4 N and having a circumference of 230 mm.

9.43 A wing of 44-ft span and a 350-ft^2 "planform" area moves horizontally through standard atmosphere at 20,000 ft with a velocity of 250 mph. If the wing supports 9000 lb, find (*a*) the required value of the lift coefficient; (*b*) the downwash velocity, assuming an elliptical distribution of lift over the span; (*c*) the induced drag.

9.44 Determine the induced angle of attack and the induced drag if the plan form of the wing in Exer. 9.12.1 were rectangular.

9.45 Determine the induced angle of attack and the induced drag if the plan form of the wing in Prob. 9.43 were rectangular.

9.46 For the Clark Y airfoil of Fig. 9.24, evaluate the friction coefficient η of Eq. (9.48) for the values of C_L of 0.6, 1.0, and 1.4.

9.47 A kite has the shape of a rectangular airfoil with a chord length of 2.5 ft and a span of 4.5 ft. When rigged and oriented as shown in Fig. P9.47, the guideline exerts a tension T of 14 lb when the wind velocity V is 28 mph in standard air at 1500 ft altitude. Find C_L, C_{D0}, and the friction coefficient η.

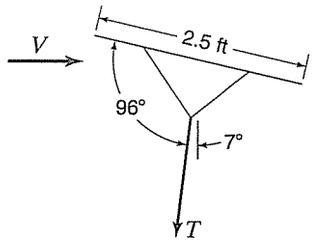


Figure P9.47

9.48 A rectangular airfoil with a 2.5-m chord and 15-m span has a drag coefficient of 0.062 and a lift coefficient of 0.95 at an angle of attack of 6.7°. What would be the corresponding (*a*) lift coefficient, (*b*) drag coefficient, and (*c*) angle of attack for a wing having the same profile but with an aspect ratio of 7.6?

9.49 Determine the angle of glide that will allow the sailplane in Exer. 9.13.1 to remain in the air for the longest time? (*Note:* A trial-and-error solution will be required here.)

9.50 A supersonic jet plane is flying horizontally at 1500 mph. How soon after it passes overhead at an elevation of 5000 ft will the shock wave be felt at sea level?

9.51 A supersonic jet plane is flying horizontally at 2800 km/h. How soon after it passes overhead at an elevation of 2 km will the shock wave be felt at sea level?

9.52 If the round-nosed and sharp-nosed projectiles of Fig. 9.27 each weigh 850 lb with a diameter of 18 in, and assuming they travel vertically downward nose first, find their terminal velocities in standard air at sea level.

9.53 If the round-nosed and sharp-nosed projectiles of Fig. 9.27 each weigh 3500 N with a diameter of 450 mm, and assuming they travel vertically downward nose first, find their terminal velocities in standard air at sea level.

CHAPTER 10
Steady Flow in Open Channels

10.1 OPEN CHANNELS

An open channel is one in which the stream is not completely enclosed by solid boundaries and therefore has a free surface subjected only to atmospheric pressure. The flow in such a channel is caused not by some external head, but rather only by the gravity component along the slope of the channel. Thus we often refer to open-channel flow as ***free-surface flow*** or ***gravity flow.*** This chapter will deal only with *steady* flow in open channels.

The principal types of open channel are natural streams and rivers; artificial canals; and sewers, tunnels, and pipelines flowing not completely full. Artificial canals may be built to convey water for purposes of water-power development, irrigation or city water supply, and drainage or flood control and for numerous other purposes. While there are examples of open channels carrying liquids other than water, few experimental data exist for such, and the numerical coefficients given here apply only to water at natural temperatures.

For convenience in dealing with large channel systems, we often divide them into reaches. A ***reach*** is a continuous stretch of a waterway, often chosen to have reasonably uniform properties like cross section, slope, and discharge.

The accurate solution of problems of flow in open channels is more challenging than in the case of pressure pipes. Not only are reliable experimental data more difficult to secure, but we meet a wider range of conditions than we do with pipes. Practically all pipes are round, but the cross sections of open channels can be of any shape, from circular to the irregular forms of natural streams. In pipes the degree of roughness ordinarily ranges from that of new smooth metal, on the one hand, to that of old corroded iron or steel pipes, on the other. But with open channels the surfaces vary from smooth timber or concrete to the rough or irregular beds of some rivers. Hence the friction coefficients to be used are more uncertain with open channels than with pipes.

We described uniform flow in Sec. 4.3 as it applies to hydraulic phenomena in general. In the case of open channels uniform flow means that the water cross section and depth remain constant over a certain reach of the channel as well as over time. This requires that the drop in potential energy due to the fall in elevation along the channel be exactly that consumed by the energy dissipation through boundary friction and turbulence.

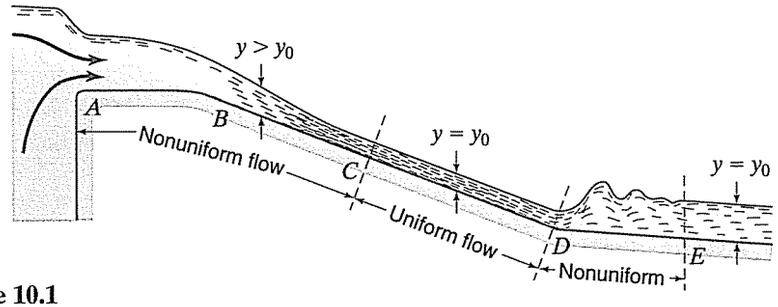


Figure 10.1
Steady flow down a chute or spillway.

Uniform flow is an equilibrium condition that flow tends to if the channel is sufficiently long with constant slope, cross section, and roughness. We can state this in another way: For any channel of given roughness, cross section, and slope, there exists for a given flow rate one and only one water depth, y_0, at which the flow will be uniform. Thus, in Fig. 10.1, the flow is accelerating in the reach from A to C, becomes established as uniform flow from C to D, suffers a violent deceleration due to the change of slope between D and E, and finally achieves a new depth of uniform flow somewhere beyond E. Acceleration occurs in the reach from B to C because the gravity component along the slope is greater than the boundary shear resistance. As the flow accelerates, the boundary shear increases because of the increase in velocity, until at C the boundary shear resistance becomes equal to the gravity component along the slope. Beyond C there is no acceleration, the velocity is constant, and the flow is uniform. We commonly refer to the depth in uniform flow as the ***normal depth,*** y_0.

Open-channel flow is usually fully rough; that is, it occurs at high Reynolds numbers. For open channels we define the Reynolds number by $\mathbf{R} = R_h V/\nu$, where R_h is the hydraulic radius (Sec. 8.3). Since $R_h = D/4$, the critical value of Reynolds number at which the changeover occurs from laminar flow to turbulent flow in open channels is 500, whereas in pressure conduits the critical value is 2000.

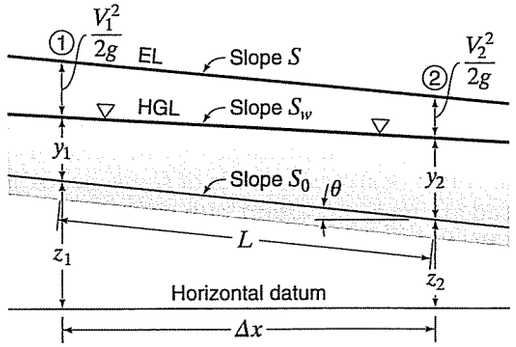


Figure 10.2
Open-channel flow—definition sketch.

In open-channel flow (Fig. 10.2) we refer to the channel ***bed slope*** S_0, the ***water surface slope*** S_w, and the slope of the energy line or the ***energy gradient*** S. It is quite evident that for open-channel flow the hydraulic grade line coincides with the water surface, provided there is no unusual curvature in the streamlines or stream tubes, for if we attach a piezometer tube to the side of the channel, the water will rise in it until its surface is level with the water surface in the channel at that point. We always measure water depth y vertically, and we usually define the distance between sections as the horizontal distance Δx between them. As a result, we define the three slopes just mentioned as

$$S_0 = \frac{z_1 - z_2}{\Delta x} = -\frac{\Delta z}{\Delta x} \tag{10.1}$$

$$S_w = \frac{(z_1 + y_1) - (z_2 + y_2)}{\Delta x} = -\frac{\Delta(z + y)}{\Delta x} \tag{10.2}$$

$$S = \frac{h_L}{L} = \frac{\left(z_1 + y_1 + V_1^2/2g\right) - \left(z_2 + y_2 + V_2^2/2g\right)}{L} \tag{10.3}$$

where x, y, Δx, and L are defined in Fig. 10.2. Note that the energy gradient S is defined as the head loss per length of flow path (not per horizontal distance). Also, we usually assume that $\alpha = 1.0$; this assumption is reasonable when the flow depth is less than the channel width. Furthermore, for most practical cases the angle θ between the channel bed and the horizontal is small; so L, the distance along the channel bed between the two sections, is usually almost equal to Δx, the horizontal distance between the two sections.

10.2 UNIFORM FLOW

In uniform flow (Fig. 10.3) the cross section through which flow occurs is constant along the channel, and so also is the velocity. Thus, $y_1 = y_2$ and $V_1 = V_2$ and the channel bed, water surface, and energy line are parallel to one another. Also,

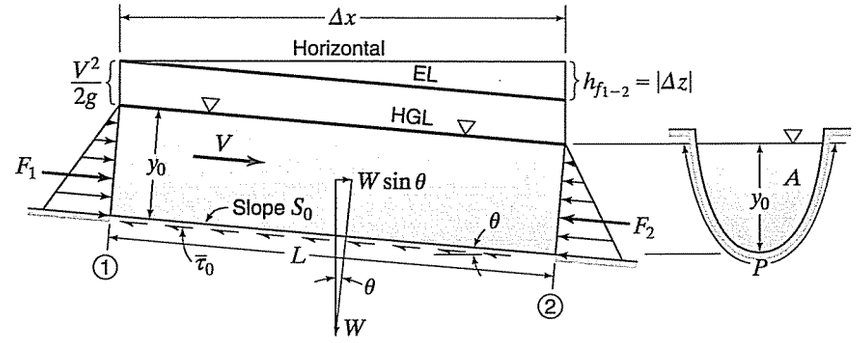


Figure 10.3
Uniform flow in open channel.

$S_w = S_0 = -\Delta z/\Delta x = \tan\theta$, while $S = h_f/L = \sin\theta$, where θ is the angle the channel bed makes with the horizontal.

In most open channels (rivers, canals, and ditches) the bed slope S_0 ($= \Delta z/\Delta x$) falls in the range of 0.01–0.0001. For some chutes, spillways, and sewers it may be larger, and for some large rivers it may be smaller. Because, therefore, usually $S_0 < 0.1$ so that $\theta < 5.7°$, it follows that in most cases $\sin\theta \approx \tan\theta$ (within 0.5%) and $S_0 = S_w \approx S$. In this chapter we will assume that $\theta < 5.7°$ and thus that $S \approx S_0$. In channels where $\theta > 5.7°$ we must make a distinction in the difference between S and S_0. Moreover, on such slopes air entrainment and pulsating flow (successions of traveling waves) are common.

In Sec. 8.4 we developed a general equation for frictional resistance in a pressure conduit. The same reasoning may now be applied to uniform flow with a free surface. Consider the short reach of length L along the channel between stations 1 and 2 in uniform flow with water cross section of area A (Fig. 10.3). As the flow is neither accelerating nor decelerating, we can consider the body of water contained in the reach in static equilibrium. Summing forces along the channel, the hydrostatic-pressure forces F_1 and F_2 balance each other, since there is no change in the depth y between the stations. The only force in the direction of motion is the gravity component, and this must be resisted by the average boundary shear stress $\bar{\tau}_0$, acting over the area PL, where P is the wetted perimeter of the section. Thus

$$\gamma AL\sin\theta = \bar{\tau}_0 PL$$

But $\sin\theta = h_f/L = S$. Solving for $\bar{\tau}_0$, we have

$$\bar{\tau}_0 = \gamma\frac{A}{P}S = \gamma R_h S \tag{10.4}$$

where R_h is the hydraulic radius,[1] discussed in Sec. 8.3, and for most slopes (with $\theta < 5.7°$) S_0 may be taken as equal to S. Substituting the value of $\bar{\tau}_0$ from Eq. (8.10) and replacing S with S_0,

$$\bar{\tau}_0 = C_f\rho\frac{V^2}{2} = \gamma R_h S_0$$

We can solve this for V in terms of either the friction coefficient C_f or the conventional friction factor f (Eq. 8.15) to give

$$V = \sqrt{\frac{2g}{C_f}R_h S_0} = \sqrt{\frac{8g}{f}R_h S_0} \tag{10.5}$$

[1] Strictly speaking, we should measure A and P in a plane at right angles to L. However, since depth is measured vertically, we calculate values of A and P as those in the vertical plane. The resulting values of R_h are identical regardless of which way we determine A and P.

The Chézy Formula

Antoine de Chézy (1718–1798), a French bridge engineer and hydraulics expert, proposed in 1775 that the velocity in an open channel varied as $\sqrt{R_h S_0}$. This led to the formula

$$V = C\sqrt{R_h S_0} \tag{10.6}$$

which is known by his name. It has been widely used both for open channels and for pipes under pressure. Comparing Eqs. (10.6) and (10.5), we see that $C = \sqrt{8g/f}$. Despite the simplicity of Eq. (10.6), it has the distinct drawback that C is not a pure number but has the dimensions $L^{1/2}T^{-1}$. This requires that we convert the values of C which are in SI units before we use them with BG units in the rest of the formula.

As C and f are related, the same considerations that are presented in Chap. 8 regarding the determination of a value for f apply also to C. For a small open channel with smooth sides, the problem of determining f or C is the same as that in the case of a pipe. But most channels are relatively large compared with pipes, thus giving Reynolds numbers that are higher than those commonly encountered in pipes. Also, open channels are frequently rougher than pipes, especially in the case of natural streams. A study of Fig. 8.11 reveals that, as the Reynolds number and the relative roughness both increase, the value of f becomes practically independent of **R** and depends only on the relative roughness.

The Manning Formula

One of the best as well as one of the most widely used formulas for uniform flow in open channels is that published by the Irish engineer Robert Manning[2] (1816–1897). Manning had found from many tests that the value of C in the Chézy formula varied approximately as $R_h^{1/6}$, and others observed that the proportionality factor was very close to the reciprocal of n, the coefficient of roughness in the previously used, but complicated and inaccurate, Kutter formula. This led to the formula that has since spread to all parts of the world. In SI units, the Manning formula is

In SI units:
$$V(m/s) = \frac{1}{n}R_h^{2/3}S_0^{1/2} \tag{10.7a}$$

The dimensions of the friction factor n are seen to be $TL^{-1/3}$. To avoid converting the numerical value of n for use with BG units, we change the formula itself so as to leave the value of n unaffected. Thus, in BG units, the Manning formula is

In BG units:
$$V(\text{fps}) = \frac{1.486}{n}R_h^{2/3}S_0^{1/2} \tag{10.7b}$$

[2] Robert Manning, Flow of Water in Open Channels and Pipes, *Trans. Inst. Civil Engrs. (Ireland)*, vol. 20, 1890.

where 1.486 is the cube root of 3.28, the number of feet in a meter. Despite the dimensional difficulties of the Manning formula, which have long plagued those attempting to put all fluid mechanics on a rational dimensionless basis, it continues to be popular because it is simple to use and reasonably accurate. Representative values of n for various surfaces are given in Table 10.1.

In terms of flow rate, Eqs. (10.7) may be expressed as

$$\text{In BG units:} \qquad Q(\text{cfs}) = \frac{1.486}{n} A R_h^{2/3} S_0^{1/2} \tag{10.8a}$$

$$\text{In SI units:} \qquad Q(\text{m}^3/\text{s}) = \frac{1}{n} A R_h^{2/3} S_0^{1/2} \tag{10.8b}$$

TABLE 10.1 **Values of n in Manning's formula**

	n	
Nature of surface	**Min**	**Max**
Lucite	0.008	0.010
Glass	0.009	0.013
Neat cement surface	0.010	0.013
Wood-stave pipe	0.010	0.013
Plank flumes, planed	0.010	0.014
Vitrified sewer pipe	0.010	0.017
Concrete, precast	0.011	0.013
Metal flumes, smooth	0.011	0.015
Cement mortar surfaces	0.011	0.015
Plank flumes, unplaned	0.011	0.015
Common-clay drainage tile	0.011	0.017
Concrete, monolithic	0.012	0.016
Brick with cement mortar	0.012	0.017
Cast iron, new	0.013	0.017
Riveted steel	0.017	0.020
Cement rubble surfaces	0.017	0.030
Canals and ditches, smooth earth	0.017	0.025
Corrugated metal pipe	0.021	0.030
Metal flumes, corrugated	0.022	0.030
Canals		
Dredged in earth, smooth	0.025	0.033
In rock cuts, smooth	0.025	0.035
Rough beds and weeds on sides	0.025	0.040
Rock cuts, jagged and irregular	0.035	0.045
Natural streams		
Smoothest	0.025	0.033
Roughest	0.045	0.060
Very weedy	0.075	0.150

Many workers define $AR_h^{2/3}$ as the ***conveyance*** K of the cross section. Then Eq. (10.8) indicates that the (uniform) flow rate depends on the surface roughness, the cross section's conveyance, and the bed slope.

Variation of n

In Sec. 8.12 we mentioned that e is a measure of the absolute roughness of the inside of a pipe. The question naturally arises whether e and n may be functionally related to each other.

To explore this, first note that we can combine Eq. (10.5) with Eq. (10.7), to yield

$$\text{In BG units:} \qquad n = 1.486 R_h^{1/6} \sqrt{\frac{f}{8g}} \tag{10.9a}$$

$$\text{In SI units:} \qquad n = R_h^{1/6} \sqrt{\frac{f}{8g}} \tag{10.9b}$$

Substituting numerical values for g gives

$$\text{In BG units:} \qquad n = 0.0926 f^{1/2} R_h^{1/6} \tag{10.10a}$$

$$\text{In SI units:} \qquad n = 0.1129 f^{1/2} R_h^{1/6} \tag{10.10b}$$

Thus we see that n is related to the friction factor, which depends on the relative roughness and Reynolds number and on the hydraulic radius, which is indicative of the size of the channel.

Second, since we know from Sec. 10.1 that open-channel flow is usually fully rough, we can take the fully-rough-pipe flow Eq. (8.50), substitute $4R_h$ for D per Sec. 10.1 and Eq. (8.21), and working in BG units for example, substitute for $\sqrt{f}$ from Eq. (10.9a). After simplifying and rearranging this results in

$$n = \frac{1.486 R_h^{1/6}}{4\sqrt{2g}\log\left(\frac{14.8R_h}{e}\right)} \quad \text{or} \quad e = 14.8 R_h \exp -\left(\frac{1.486(\ln 10) R_h^{1/6}}{4\sqrt{2g}\, n}\right)$$

where for SI units we simply replace 1.486 by 1.000. These provide a relationship between e and n, which is plotted as the solid lines in Fig. 10.4 for three representative values of the hydraulic radius. The dashed line is the plot of another correlation proposed by Powell[3] that gave a better fit to experimental data for rectangular channels and small values of hydraulic radius.

One notable feature of the curves in Fig. 10.4 is that a large relative error in e causes only a small error in n. Another important observation concerns the

[3] R. W. Powell, Resistance to Flow in Rough Channels, *Trans. Am. Geophys. Union,* vol. 31, no. 4, pp. 575–582, 1950.

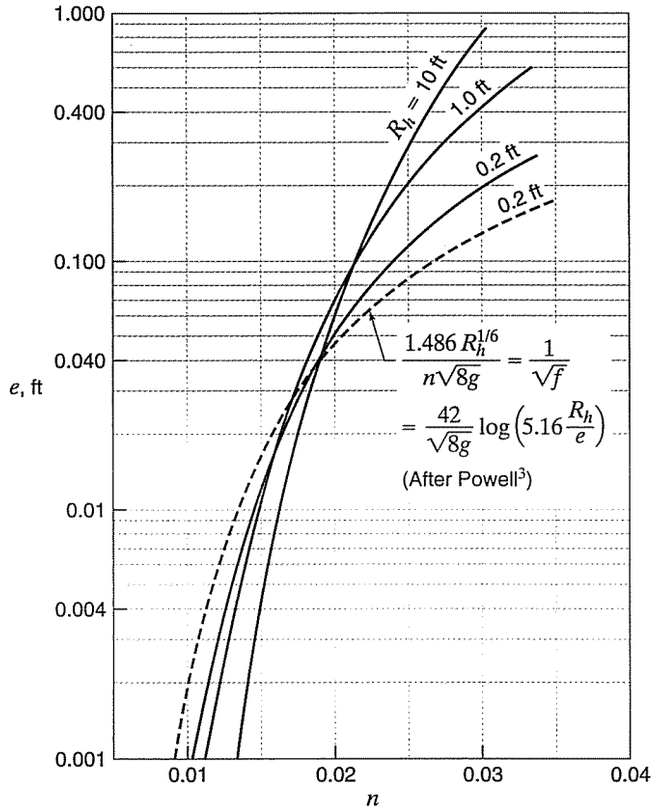


Figure 10.4
Correlation of n with absolute roughness e.

variation of n with channel size for the same absolute roughness. For channels that are quite rough, like natural channels with $e > 0.10$ ft, we see in Fig. 10.4 that n gets smaller with increasing hydraulic radius. This, being similar to the behavior of relative roughness e/D in pipes, has long been understood. However, at lower e values the curves cross, with the result that for smoother, artificial surfaces with $e < 0.02$ ft we find that n *increases* with increasing hydraulic radius. This contrary behavior was formerly not understood, with the result that some very large concrete-lined canal systems were designed using inappropriately low n values; when put into service, designers found that the higher true n values reduced the capacity of the canals by as much as 15%.

10.3 SOLUTION OF UNIFORM FLOW PROBLEMS

Uniform flow problems usually involve the application of Manning's equation (Eqs. 10.8). The selection of an appropriate value for the Manning roughness factor n is critical to the accuracy of the results of a problem. When the channel surface is concrete or some other structural material, it is possible to select a

reasonably accurate value for n, but for the case of a natural channel one must rely on judgment and experience, and in many instances the selected value may be quite inaccurate.

We encounter a number of different types of problems when using Manning's equation. For example, to find the normal depth of flow for a particular flow rate in a given channel, we require a trial-and-error solution, because y_0 is involved in A and R_h in complicated ways (see, for example, Sample Prob. 10.1). On the other hand, we can solve for the expected flow in a particular channel under given conditions directly. Various types of sliderules, nomographs, tables,[4] and computer programs,[5] are available to serve as an aid to the solution of open-channel problems. Some of these provide helpful visual representations of the interdependence of the various factors, but their applicability is usually limited.

SAMPLE PROBLEM 10.1 The channel in Fig. S10.1 has a bed slope of 0.0006 and a Manning's n of 0.016. Using only a basic scientific calculator, (*a*) find the uniform flow depth when the flow rate is 225 cfs and (*b*) compute the corresponding value of the absolute roughness e. Also, find the uniform flow depth using (*c*) equation solving software, and (*d*) an equation solver on a programmable scientific calculator.

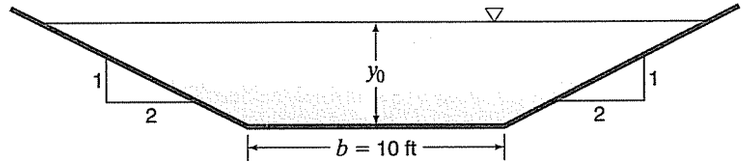


Figure S10.1

Solution

(*a*)
$$A = (10 + 2y_0)y_0$$

and
$$R_h = \frac{A}{P} = \frac{(10 + 2y_0)y_0}{10 + 2\sqrt{5}y_0}$$

Eq. (10.8*a*): $$Q = 225 = \frac{1.486}{0.016}(10 + 2y_0)y_0\left[\frac{(10 + 2y_0)y_0}{10 + 2\sqrt{5}y_0}\right]^{2/3}(0.0006)^{1/2} \quad (1)$$

[4] E. F. Brater, H. W. King, J. E. Lindell, and C. Y. Wei, *Handbook of Applied Hydraulics,* 7th ed., McGraw-Hill Book Company, New York, 1996.
[5] See citations in Appendixes C–E.
▦ : Programmed computing aids (Appendix C) could help solve problems marked with this icon.

By trial (see Sample Probs. 3.5 and 5.8), $y_0 = 3.41$ ft, uniform flow depth ***ANS***

Note: The accuracy with which flow depth can be measured can differ considerably, depending on many factors. In a laboratory, we can sometimes measure depths to 0.001 ft (0.3 mm), whereas in the field, with turbulent flow and wind, surface fluctuations (which we can average) may make it difficult to achieve an accuracy of 0.1 ft (30 mm).

(*b*) $$A = [10 + 2(3.41)]3.41 = 57.3 \text{ ft}^2$$
$$P = 10 + 2\sqrt{5}(3.41) = 25.2 \text{ ft}$$
$$R_h = \frac{A}{P} = \frac{57.3}{25.2} = 2.27 \text{ ft}$$

Eq. (10.10*a*): $$0.016 = 0.0928 f^{1/2}(2.27)^{1/6}$$

from which $$f^{1/2} = 0.1503 \quad (f = 0.0226)$$

Eq. (10.9): $$\frac{1}{0.1503} = 2 \log\left(\frac{14.8 \times 2.27}{e}\right)$$

from which $$e = 0.01585 \text{ ft} \qquad \textbf{\textit{ANS}}$$

(*c*) Using equation solving software such as Excel or Mathcad (Appendixes C.2–C.3 and D.2–D.3), open a file (worksheet) and enter the four governing equations,

Eq. (10.8): $$Q = \frac{c}{n} A R_h^{2/3} S_0^{1/2}$$

and $$A = (b + m y_0) y_0, \quad P = b + 2y_0\sqrt{1 + m^2}, \quad R_h = \frac{A}{P}$$

These four equations involve 10 variables, four of which are those to be found. The quantity c must be 1.486 for problems in BG units and 1.0 for those in SI units. We define the side slope m as the horizontal run over the vertical rise; for rectangular channels $m = 0$, in Fig. S10.1 $m = 2$.

Assign numerical values to the six known variables (Q, c, b, m, n, S_0) as earlier. Then assign estimated values to the four unknown variables, for example $y_0 = 1.0$, $A = 10$, $P = 10$, $R_h = 1.0$.

The results are

$$y_0 = 3.406 \text{ ft}, \quad A = 57.27 \text{ ft}^2, \quad P = 25.23 \text{ ft}, \quad R_h = 2.270 \text{ ft} \qquad \textbf{\textit{ANS}}$$

Note: Some software packages include polynomial solvers (Appendixes B, C.3, and D.3). If we cube and rearrange Eq. (1), it becomes a polynomial of degree 10 (Appendix B). A polynomial solver will rapidly find all 10 roots, nine of which are either negative or imaginary.

(*d*) To use an equation solver on a programmable scientific calculator (Appendixes C.1, D.1, Sec. 8.17), we recall that such solvers can solve only one nonlinear equation (which can be implicit, see Appendix B) for one unknown.

We can create such a single equation by substituting the last three equations in part (*c*) into Eq. (10.8) to eliminate A, P, and R_h, which yields

$$Q = \frac{c}{n} \frac{(b y_0 + m y_0^2)^{5/3}}{(b + 2y_0\sqrt{1 + m^2})^{2/3}} S_0^{1/2}$$

Using an "equation writer," key in this equation, and store it permanently, assigning it a name. Then activate the equation solving feature, select the desired equation from the catalog of equations, and make it the "current" equation.

Display the menu of variables in the equation, and enter numeric values for them all (Q, c, b, m, n, S_0) as before plus an estimate of y_0 (1 ft, say). Instruct the solver to solve for y_0. The result is

$$y_0 = 3.406 \text{ ft} \qquad \textbf{\textit{ANS}}$$

Note: Some programmable scientific calculators include polynomial solvers (Appendix C.1), which we can use in the same way as those in software, see the note for part (*c*).

In applying Manning's equation to channel shapes such as that in Fig. 10.5, which simulates a river with overbank flow conditions, the usual procedure is to break the section into several parts, as indicated in the figure. We assume that there is no resistance along the dashed vertical lines. Actually the flow in area A_2 tends to speed up the flow in area A_1, while the flow in A_1 tends to slow down the flow in area A_2. These two effects come very close to balancing out one another. If A/P for the total cross section had been computed by the usual method, that is, $R_h = (A_1 + A_2 + A_3)/(P_1 + P_2 + P_3)$, it would imply that the effect of boundary resistance is uniformly distributed over the flow cross section, which, of course, is not the case.

Another advantage of breaking the total section into parts is that we can take into consideration possible variations in Manning's n. Thus, for the channel shown in Fig. 10.5, in BG units

$$Q = \frac{1.486}{n_1} A_1 R_{h_1}^{2/3} S_0^{1/2} + \frac{1.486}{n_2} A_2 R_{h_2}^{2/3} S_0^{1/2} + \cdots \tag{10.11}$$

where $R_{h_1} = A_1/P_1$, $R_{h_2} = A_2/P_2$, etc. The A and P are defined in Fig. 10.5. We can write an equation of similar form for SI units in which case the 1.486 is replaced by unity.

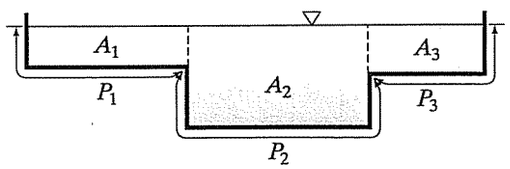


Figure 10.5

EXERCISES

10.3.1 For the channel of Sample Prob. 10.1, compute the flow rate for depths of 2, 4, 6, and 8 ft. Plot a curve of Q versus y.

10.3.2 Water flows uniformly in a trapezoidal channel with a 16-ft-wide bed and 1.9:1 (H:V) side slopes. If the bed slope is 1.2 ft/mile and $n = 0.015$, find the flow rate when the depth is 9 ft.

10.3.3 Water flows uniformly in a 2.5-m-wide rectangular channel at a depth of 300 mm. The channel slope is 0.0028 and $n = 0.014$. Find the flow rate in m^3/s.

10.3.4 At what depth will water flow in a 4-m-wide rectangular channel if $n = 0.018$, $S_0 = 0.0009$, and $Q = 7\ m^3/s$?

10.3.5 Figure X.10.3.5 shows a tunnel section on the Colorado River Aqueduct. The area of the water cross section is 191 ft^2, and the wetted perimeter is 39.1 ft. The flow is 1600 cfs. If $n = 0.013$ for its concrete lining, find the slope.

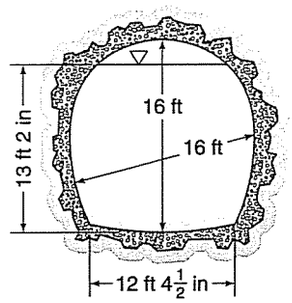


Figure X10.3.5

10.3.6 For the channel cross section of Fig. X10.3.6, $a = 1$ m, $b = 3$ m, $d = 2$ m, $w = 8$ m, and $n = 0.015$. What bed slope is required so that the flow will be 16 m^3/s when the depth of flow is 1.50 m?

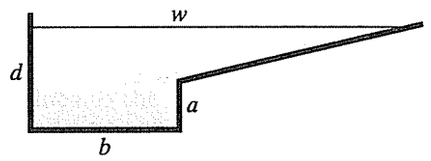


Figure X10.3.6

10.3.7 Water flows in a rectangular flume 6 ft wide made of unplaned timber ($n = 0.013$). Find the necessary channel slope if the water flows uniformly at a depth of 3 ft and at 20 fps.

10.3.8 Water flows in a rectangular flume 1.5 m wide made of unplaned timber ($n = 0.013$). Find the necessary channel slope if the water flows uniformly at a depth of 0.6 m and at 4.5 m/s.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.3.9 Refer to Fig. 10.5. Suppose the widths of A_1, A_2, and A_3 are 120, 50, and 240 ft and the total depths are 3, 12, and 4 ft. Compute the flow rate if $S_0 = 0.0018$, $n_1 = n_3 = 0.06$, and $n_2 = 0.030$.

10.3.10 Refer to Fig. 10.5. Suppose the widths of A_1, A_2, and A_3 are 25, 8, and 45 m and the total depths are 0.4, 2.8, and 1.1 m. Compute the flow rate if $S_0 = 0.0018$, $n_1 = n_3 = 0.035$, and $n_2 = 0.022$.

10.4 VELOCITY DISTRIBUTION IN OPEN CHANNELS

Due to friction around the wetted perimeter, we expect strong velocity variations across flow cross sections in open channels. A small frictional effect must also occur at the water surface, due to air drag.

Vanoni[6] has demonstrated that the universal logarithmic velocity-distribution law for pipes (Eq. 8.40) also applies to a two-dimensional open channel, i.e., one of uniform depth that is infinitely wide (see Sec. 10.5). We can write this equation as

$$\frac{u - u_{max}}{\sqrt{gy_0S}} = \frac{2.3}{K}\log\frac{y}{y_0}$$

where y_0 = depth of water in channel
u = velocity at a distance y from channel bed
K = von Kármán constant, having a value of about 0.40
S = channel bed slope

Integrating this expression over the depth yields the more useful relation

$$u = V + \frac{1}{K}\sqrt{gy_0S}\left(1 + 2.3\log\frac{y}{y_0}\right) \tag{10.12}$$

which expresses the distribution law in terms of the mean velocity V. This equation is plotted in Fig. 10.6, together with velocity measurements that were made on the centerline of a rectangular flume 2.77 ft (0.844 m) wide with a water depth of 0.59 ft (0.180 m). We see that the filament whose velocity u is equal to V lies at a distance of $0.632y_0$ beneath the surface.

Velocity measurements made in a trapezoidal channel result in the isovels (contours of equal velocity) shown in Fig. 10.7. The point of maximum velocity is seen to lie beneath the surface, the result of air drag. From this two-dimensional velocity distribution the values of the correction factors for kinetic energy and momentum were computed, and are given with the figure. Although they are greater than the corresponding values for pipe flow (Eqs. 8.45), the treatment in this chapter follows the earlier procedure of assuming the values of α and β to be unity, unless stated otherwise. Any thorough analysis would of course have to take account of their true values.

[6] V. A. Vanoni, Velocity Distribution in Open Channels, *Civil Eng.*, vol. 11, pp. 356–357, 1941.

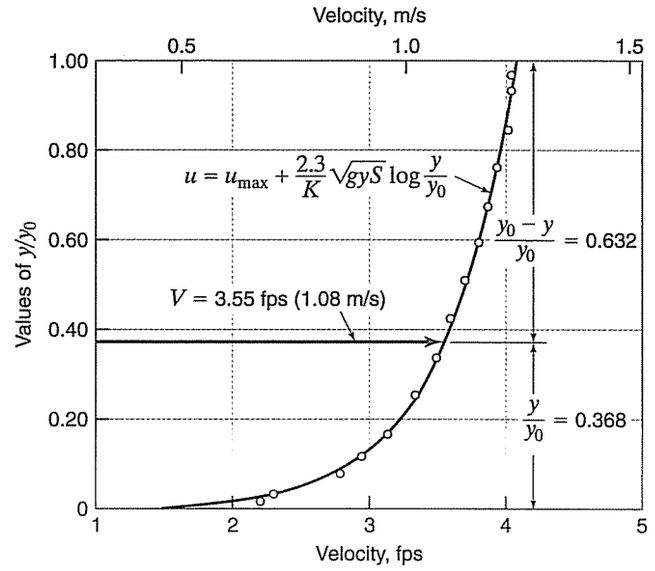


Figure 10.6
Velocity profile at the center of a flume 2.77 ft (0.844 m) wide for a flow 0.59 ft (0.180 m) deep (after Vanoni). The circles represent measurements.

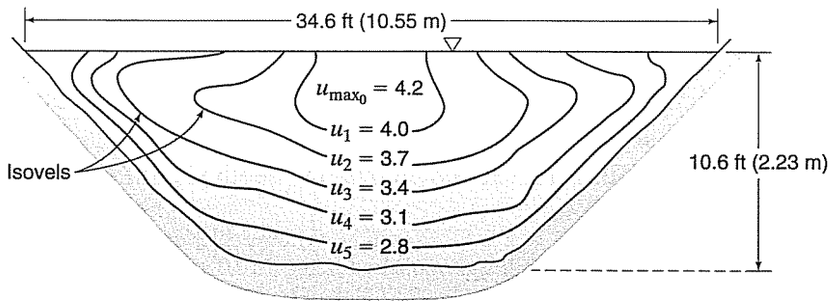


Figure 10.7
Velocity distribution in a trapezoidal canal. $V = 3.32$ fps (1.012 m/s), $A = 230.5$ ft^2 (21.4 m^2), $S_0 = 0.000057$, $\alpha = 1.105$, $\beta = 1.048$.

EXERCISES

10.4.1 In Fig. X10.4.1 with uniform flow in a very wide open channel ($R_h = y$), $a = 3.20$ ft. Find V and b if $n = 0.015$.

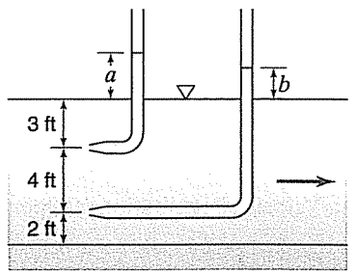


Figure X10.4.1

10.4.2 Water flows uniformly in a very wide rectangular channel ($R_h = y$) at a depth y_0 of 2.0 m. $S_0 = 0.005$ and $n = 0.018$. Calculate the velocities at y-values of 0.2, 0.4, 0.8, 1.2, and 2.0 m above the bed and plot the velocity profile. Note the value of the maximum velocity at the water surface.

10.5 "WIDE AND SHALLOW" FLOW

If the flow in a channel is sufficiently wide compared with its depth, it is reasonable to expect that the channel sides will have negligible effect on the flow in the central portion of the channel. Experiments have confirmed this to be the case provided the width-to-depth ratio $b/y > 10$. Then the central flow may be considered to be the same as in a channel of infinite width—in other words, to be two-dimensional flow.

For a small portion of such a two-dimensional flow with width Δb, the flow cross-sectional area is $y\,\Delta b$ and the wetted perimeter is Δb, so that the hydraulic radius is

Wide and shallow:
$$R_h = \frac{A}{P} = \frac{y\Delta b}{\Delta b} = y \tag{10.13}$$

This convenient result is commonly assumed for ***wide and shallow open channels.*** The same result is obtained by considering the hydraulic radius of a rectangular channel,

$$R_h = \frac{A}{P} = \frac{by}{b + 2y} = \frac{y}{1 + 2y/b}$$

As the ratio $b/y \to \infty$, we see that $R_h \to y$.

Analytically, the percentage error in R_h when using y as the hydraulic radius for a rectangular channel is

$$\frac{100(y - R_h)}{R_h} = 100\left(y - \frac{by}{b + 2y}\right)\frac{b + 2y}{by}$$

i.e.,
$$\text{Error \% in } R_h = 200\frac{y}{b}$$

This results in a 20% error for $b/y = 10$, a 5% error for $b/y = 40$, and a 2% error for $b/y = 100$. If we make the assumption that a channel is "wide and shallow" and replace R_h by y in our analysis, it is important to check that the b/y ratio is sufficiently large to justify this.

Solution for the uniform flow depth is greatly simplified when a channel is wide and shallow, because trial and error is no longer necessary. If we choose BG units, substituting for A and R_h into Manning's Eq. (10.8b) yields

$$Q = \frac{1.486}{n}(by_0)y_0^{2/3}S_0^{1/2}$$

from which

Wide and shallow (BG units):
$$y_0 = \left(\frac{nQ}{1.486bS_0^{1/2}}\right)^{0.6}$$

SAMPLE PROBLEM 10.2 A 12-m-wide stream has a reasonably rectangular cross section, and the average depth is computed to be 112 mm. What would be the percentage error in the hydraulic radius if this stream were assumed to be "wide and shallow"? Would it be reasonable to make this assumption?

Solution

$$\frac{b}{y} = \frac{12}{0.112} = 107.1$$

$$\text{Error percent} = \frac{200}{107.1} = 1.867\% \qquad \textit{ANS}$$

This error is small enough that the assumption is reasonable. ***ANS***

EXERCISES

10.5.1 Water flows 0.15 in deep in a 9-in-wide rectangular flume. What would be the percentage error in the hydraulic radius if this were assumed to be wide and shallow open-channel flow?

10.5.2 Water flows 7 mm deep in a 0.5-m-wide rectangular flume. What would be the percentage error in the hydraulic radius if this were assumed to be wide and shallow open-channel flow?

10.5.3 At a location where the cross section of a stream is reasonably rectangular, the width is 15 ft and the average depth is computed to be 20.6 in. What would be the percentage error in the hydraulic radius if this stream were assumed to be "wide and shallow"? Would it be reasonable to make this assumption?

10.6 MOST EFFICIENT CROSS SECTION

Any of the open-channel formulas already given show that, for a given slope and roughness, the velocity increases with the hydraulic radius. Therefore, for a given area of water cross section, the rate of discharge will be a maximum when R_h is a maximum, i.e., when the wetted perimeter P and so the frictional resistance is a minimum. We call such a section the ***most efficient cross section*** for the given area, and we emphasize that here we are addressing only the *hydraulic efficiency*. Or for a given rate of discharge, the cross-sectional area will be a minimum when the design is such as to make R_h a maximum (and thus P a minimum). This section would be the most efficient cross section for the given rate of discharge. These issues are important, because reducing P tends to reduce channel construction costs. However, other factors influence the overall cost of an open channel. For example, the cost of deep excavation is higher than shallow excavation. Consequently most channels are designed to flow with depths less than the hydraulic optimum, resulting in R_h values a few percent smaller than

$R_{h_{max}}$. In situations where right-of-way costs are very high, or where right-of-way width is limited, open channels are sometimes deeper than the hydraulic optimum.

Of all geometric figures, the circle has the least perimeter for a given area. The hydraulic radius of a semicircle is the same as that of a circle. Hence a semicircular open channel will discharge more water than one of any other shape, assuming that the area, slope, and surface roughness are the same. Semicircular open channels are often built of pressed steel and other forms of metal, but for other types of construction such a shape is impractical. For wooden flumes designers usually select a rectangular shape. Canals excavated in earth usually have a trapezoidal cross section, with side slopes less than the angle of repose of the saturated bank material. Besides hydraulic efficiency, such factors as those mentioned help determine the best cross section.

We can determine the shape of the most efficient trapezoidal cross section by using its geometric properties and a little calculus. Consider the trapezoid of Fig. 10.8, with bed width b and side slope m:1. It follows that $A = (b + my)y$ and $P = b + 2y\sqrt{1 + m^2}$. Eliminating b, we obtain

$$P = \frac{A}{y} + y(2\sqrt{1 + m^2} - m)$$

and

$$R_h = \frac{A}{P} = \frac{A}{\frac{A}{y} + y(2\sqrt{1 + m^2} - m)}$$

Case 1: For a given A and m, we find $R_{h_{max}}$ by differentiating R_h with respect to y while holding A and m constant, and setting the result equal to zero. From this we obtain

$$y_{opt} = \sqrt{\frac{A}{2\sqrt{1 + m^2} - m}}$$

$$R_{h_{max}} = \frac{y_{opt}}{2}$$

and $$P_{opt} = 2y_{opt}(2\sqrt{1 + m^2} - m), \qquad b_{opt} = 2y_{opt}(\sqrt{1 + m^2} - m)$$

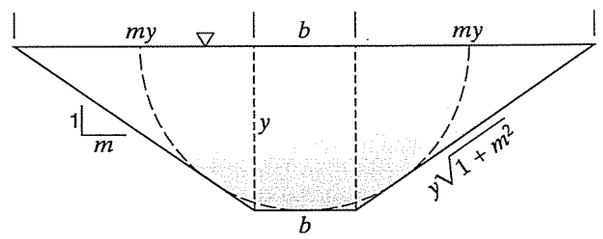


Figure 10.8

We may find Q_{opt} by substituting $R_{h_{max}}$ into Eq. (10.8).

Case 2: For a given Q, m, n, and S_0, we rearrange Eq. (10.8*a*) for BG units into

$$A = \frac{Qn}{1.486R_h^{2/3}S_0^{1/2}}$$

and use this to eliminate the variable A from the geometric relations. Then we find $R_{h_{max}}$ by differentiating R_h with respect to y while holding Q, m, n, and S_0 constant, and setting dR_h/dy to zero. From this we obtain

$$y_{opt} = 2^{1/4}\left[\frac{Qn}{1.486(2\sqrt{1+m^2}-m)S_0^{1/2}}\right]^{3/8}$$

$$A_{opt} = y_{opt}^2(2\sqrt{1+m^2}-m)$$

and that $R_{h_{max}}$, P_{opt}, and b_{opt} are given by the same equations as in Case 1. For SI units the equations are the same except we must replace 1.486 by 1.000.

Of these results for any trapezoid, the equation for $R_{h_{max}}$ is easy to remember. It is also interesting (see Prob. 10.12) that the optimum trapezoid always envelopes a semicircle whose center is in the water surface (shown dashed in Fig. 10.8).

For the common rectangular cross section, this is a trapezoid with $m = 0$, so these results then reduce to $A = 2y^2$, $P = 4y$, $b = 2y$, and $R_h = y/2$. Thus the most efficient rectangular section has its depth equal to half its width, and this of course still envelopes a semicircle.

These results for trapezoidal cross sections are for given values of side slope m. We can find the hydraulically best shape of trapezoid (value of m) by expressing $R_{h_{max}}$ as a function of only known quantities and m, e.g., as a function of only A and m when A is given, in which case

$$R_{h_{max}} = \frac{y_{opt}}{2} = \frac{1}{2}\sqrt{\frac{A}{2\sqrt{1+m^2}-m}}$$

and then differentiating $R_{h_{max}}$ with respect to m while holding A constant. The result is that $dR_{h_{max}}/dm$ is zero when $m = \infty$ or

$$\left(\frac{2m}{\sqrt{1+m^2}} - 1\right) = 0$$

We obtain the same result if instead Q, n, S_0, and m are given. Thus $R_{h_{max}}$ is maximum when $m = 1/\sqrt{3}$, i.e., the side slopes are at 60° to the horizontal and the trapezoid is a half-hexagon. We note that this is the trapezoid that most closely envelopes the semicircle.

In a similar manner we can show that the most efficient triangular cross section is the one that has a total vertex angle of 90°.

SAMPLE PROBLEM 10.3 The first 95 miles of the Delta-Mendota Canal in California, designed to carry 4600 cfs, has the following dimensions (referring to Fig. 10.8): $b = 48$ ft, $m = 1.5$, $y = 16.7$ ft. Using the same cross-sectional area and side slopes, how much would the most efficient cross section increase the present hydraulic radius and flow capacity? What would be the corresponding depth and bed width?

Solution

Existing: $$A = (48 + 1.5\times 16.7)16.7 = 1220 \text{ ft}^2$$

$$P = 48 + 2\left(\frac{\sqrt{13}}{2}\right)16.7 = 108.2 \text{ ft}$$

so $$R_h = \frac{A}{P} = \frac{1220}{108.2} = 11.27 \text{ ft}$$

which we note is considerably different from $y/2 = 8.35$ ft.

For an optimum cross section we note this is Case 1 (given A), for which

$$y_{opt} = \sqrt{\frac{1220}{2\sqrt{3.25}-1.5}} = 24.07 \text{ ft} \qquad \textit{ANS}$$

Note that this is 44% greater than the existing depth.

Also $$(R_h)_{max} = \frac{y_{opt}}{2} = 12.04 \text{ ft}$$

This is only 6.8% greater than the existing hydraulic radius. ***ANS***

From Manning's Eq. (10.8) for the same cross-sectional area:

$$\frac{Q_{opt}}{Q} = \left[\frac{(R_h)_{max}}{R_h}\right]^{2/3} = \left(\frac{12.04}{11.27}\right)^{2/3} = 1.0450$$

Therefore Q_{opt} is 4.50% greater than the existing capacity. ***ANS***

$$b_{opt} = 2(24.07)(\sqrt{3.25} - 1.5) = 14.58 \text{ ft} \qquad \textit{ANS}$$

EXERCISES

10.6.1 What diameter of semicircular channel will have the same capacity as a 3-ft-deep rectangular channel 8 ft wide. Assume S_0 and n are the same for both channels. Compare the lengths of the wetted perimeters.

10.6.2 A rectangular flume of planed timber ($n = 0.012$) slopes 0.5 ft per 1000 ft. (*a*) Compute the rate of discharge if the width is 7 ft and the depth of water is 3.5 ft (Fig. X10.6.2). (*b*) What would be the rate of discharge if the width were 3.5 ft and the depth of water 7 ft? (*c*) Which of the two forms would have the greater capacity and which would require less lumber?

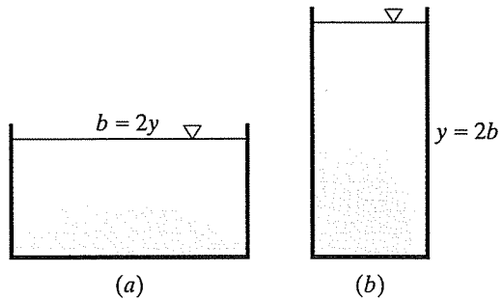


Figure X10.6.2

10.6.3 A rectangular flume of planed timber ($n = 0.012$) slopes 1 m per km. (*a*) Compute the rate of discharge if the width is 2 m and the depth of water is 1 m (Fig. X10.6.2). (*b*) What would be the rate of discharge if the width were 1 m and the depth of water 2 m? (*c*) Which of the two forms would have the greater capacity and which would require less lumber?

10.7 CIRCULAR SECTIONS NOT FLOWING FULL

In circular pipes flow frequently occurs at partial depth, in which case they are behaving as open channels. As we can visualize from Fig. 10.9, at depths just slightly less than full depth the wetted perimeter (frictional resistance) is reduced in greater proportion than the flow area, so the hydraulic radius A/P is greater than when full. In accordance with the Manning formula, therefore, the maximum rate of discharge in such a section occurs at slightly less than full depth.

We can explore such maximum condition analytically. Referring to Fig. 10.9,

$$y = 0.5D(1 - \cos\theta) = D\sin^2(\theta/2)$$

$$A = \frac{D^2}{4}(\theta - \sin\theta\cos\theta) = \frac{D^2}{4}\left(\theta - \tfrac{1}{2}\sin 2\theta\right)$$

$$P = D\theta$$

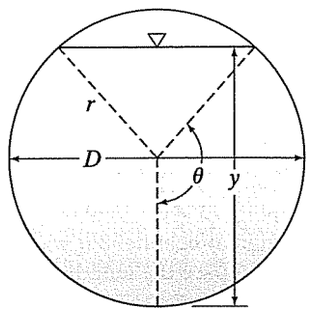


Figure 10.9
Circular section not full.

where θ is expressed in radians. This gives

$$R_h = \frac{A}{P} = \frac{D}{4}\left(1 - \frac{\sin\theta\cos\theta}{\theta}\right) = \frac{D}{4}\left(1 - \frac{\sin 2\theta}{2\theta}\right)$$

For the maximum rate of *discharge,* the Manning formula indicates that $AR_h^{2/3}$ must be a maximum. Substituting the preceding expressions for A and R_h into $AR_h^{2/3}$ and differentiating with respect to θ, setting equal to zero and solving for θ gives $\theta = 151.2°$, which corresponds to $y = 0.938D$ for the condition of maximum discharge. By differentiating $R_h^{2/3}$ the maximum *velocity* is found to occur at $0.813D$.

These analyses of optimum conditions *are theoretical only,* and not recommended for general use, because they assume a constant roughness coefficient in Manning's equation. In Sec. 10.2 we saw how, for constant e, the friction factor n varied with R_h. For circular pipes investigators have found that the value of n increases by as much as 28% from the full depth to about one-quarter full depth, where it appears to be a maximum (Fig. 10.10). This effect causes the actual maximum discharge and velocity to occur at water depths of about 0.97 and 0.94 full depth, respectively, as indicated in Fig. 10.10.[7] The interesting result is that in the small topmost depth range where the discharge is equal to or greater than Q_{full}, for a given discharge two different depths may occur.

The simplest way to handle the problem of a partially full circular section is to compute the velocity or flow rate for the pipe-full condition and then adjust to

[7] See Design and Construction of Urban Stormwater Management Systems, *ASCE Manuals and Reports on Engineering Practice No.* 77, p. 146, 1992.

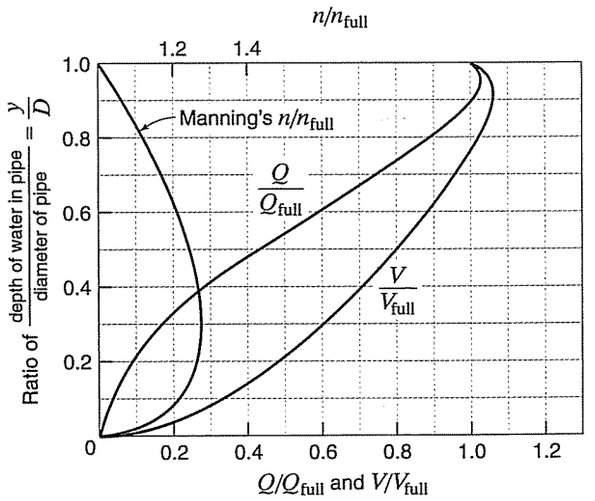


Figure 10.10
Hydraulic characteristics of circular pipe flowing partly full (including for the effects of n varying with depth).

partly full conditions by using a chart such as Fig. 10.10. On this chart the effect of the variation of Manning's n with depth is already taken into consideration.

Despite the analysis just made of optimum conditions, circular sections are often designed to carry the design capacity when flowing full, since the conditions producing maximum flow frequently include sufficient backwater to place the conduit under slight pressure.

The rectangle, trapezoid, and circle are the simplest geometric shapes from the standpoint of hydraulics, but other forms of cross section are often used, either because they have certain advantages in construction or because they are desirable from other standpoints. Thus oval- or egg-shaped sections are common for sewers and similar channels where there may be large fluctuations in the rate of discharge. They are used because it is desirable that the velocity, when a small quantity is flowing, is kept high enough to prevent the deposit of sediment, and when the conduit is full, the velocity should not be so high as to cause excessive wear of the pipe wall. For example, in sanitary sewers we usually maintain a velocity of at least 2 fps to prevent deposition of sediment, while in concrete channels conveying storm water, velocities in excess of 10 or 12 fps may result in excessive abrasion of the channel sides and bottom.

SAMPLE PROBLEM 10.4 A 36-in-diameter pipe flows just full when it carries 30 cfs. What will be the discharge and depth when the velocity is 2 fps?

Solution

$$V_{\text{full}} = \frac{Q_{\text{full}}}{A} = \frac{30}{\pi 3^2/4} = 4.24 \text{ fps}$$

At 2 fps: $V/V_{\text{full}} = 2/4.24 = 0.471$

Fig. 10.10 for $V/V_{\text{full}} = 0.471$: $y/D \approx 0.20$

So $y \approx 0.20(36) = 7.2 \text{ in}$ ***ANS***

Fig. 10.10 for $y/D = 0.20$: $Q/Q_{\text{full}} \approx 0.077$

So $Q \approx 0.077(30) = 2.31 \text{ cfs}$ ***ANS***

EXERCISES

10.7.1 Water flows uniformly in a 14-ft-diameter concrete pipe ($n = 0.014$, $S_0 = 0.0004$) at a depth of 6 ft. Using Fig. 10.10, determine the flow rate and the average flow velocity.

10.7.2 At what depth will water flow at 0.30 m³/s in a 1.20-m-diameter concrete pipe on a slope of 0.003? (*a*) Assume $n = 0.013$. (*b*) Repeat with $n = 0.015$ and compare results.

10.8 LAMINAR FLOW IN OPEN CHANNELS

Laminar flow in open channels is sometimes encountered in industrial processes where a very viscous liquid is flowing in a trough or similar conveyance structure. More commonly, however, laminar flow occurs as ***sheet flow,*** a thin sheet of flowing liquid, such as that in drainage from sidewalks, streets, and airport runways.

Sheet flow is essentially two-dimensional, and can be analyzed in that fashion. Consider two-dimensional uniform laminar flow at depth y_0 as shown in Fig. 10.11. At a distance y above the channel bed the shear stress in the flow is τ and the velocity is v. The forces acting on a unit width of the liquid represented by the shaded area include the hydrostatic forces which cancel out, the gravity force[8] component parallel to the slope $\gamma(y_0 - y)LS$ and the shear force τL exerted along the lower boundary of the shaded liquid by the liquid below it. Since the flow is uniform, there is no acceleration and thus these two forces must balance one another. Hence

$$\gamma(y_0 - y)LS = \tau L$$

From this expression, it can be seen that the shear stress must vary linearly from zero at the liquid surface to a maximum value at the channel bed.

Since the flow is laminar, we can use Eq. (2.9) to replace τ with $\mu\, dv/dy$. Making this substitution, we get

$$\gamma(y_0 - y)S = \mu\frac{dv}{dy}$$

Separating variables and integrating, noting that $v = 0$ when $y = 0$, gives

$$dv = \frac{\gamma}{\mu}(y_0 - y)S\,dy$$

and

$$v = \frac{\gamma}{\mu}\left(y_0 y - \frac{y^2}{2}\right)S = \frac{g}{\nu}\left(y_0 y - \frac{y^2}{2}\right)S \qquad (10.14)$$

[8] We always measure depth y vertically. Thus $\gamma(y_0 - y)L$ is not precisely the weight of the shaded volume of liquid. However, for the small slopes usually encountered it is an excellent approximation.

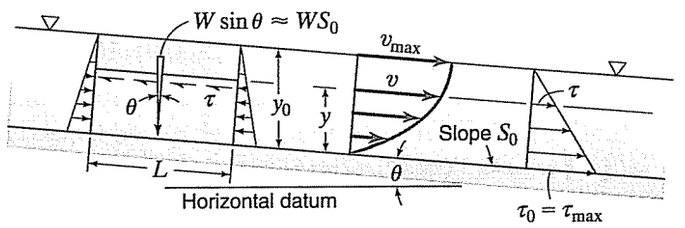


Figure 10.11
Laminar flow in open channel of infinite width and uniform depth, showing the velocity profile and the shear-stress distribution.

This is the equation of a parabola; thus the velocity profile is parabolic, as it was for laminar flow in a pipe (Sec. 8.7).

We can now integrate Eq. (10.14) over the depth from $y = 0$ to $y = y_0$ to obtain an expression for q, the flow rate per unit width:

$$dq = \int_0^{y_0} v\,dy$$

$$q = \frac{g}{\nu}\frac{y_0^3}{3}S \tag{10.15}$$

From Eqs. (10.14) and (10.15), it can be shown that the average velocity V for this case is equal to $\frac{2}{3}v_{max}$. In contrast, for laminar flow in a pipe flowing full it was shown (Sec. 8.6) that $V = \frac{1}{2}v_{max}$. The total flow Q through any width B will be qB.

Equation (10.15) shows that if the flow is laminar, the flow rate is independent of the roughness. However, in situations where there are significant irregularities in the surface over which the liquid flows, the flow may be disturbed such that it is not everywhere laminar. Consequently, we must apply Eqs. (10.14) and (10.15) with caution.

SAMPLE PROBLEM 10.5 Water is observed to flow uniformly across a glass roof at a depth of 1 mm. (*a*) If the temperature is 15°C and the slope of the roof is 0.08, what is the discharge rate from the edge of the roof per meter length? (*b*) How fast should we expect an oil mark on the water to move?

Solution

Table A.1 for 15°C: $\nu = 1.139\times10^{-6}$ m²/s

(*a*) Assuming laminar flow:

Eq. (10.15): $$q = \frac{9.81}{1.139\times10^{-6}}\frac{(0.001)^3}{3}0.08 = 0.000230 \text{ m}^3/\text{s per meter}$$

Since $b/y > 1/0.001 = 1000$, per Eq. (10.13): $R_h = y_0 = 0.001$ m

$$V = \frac{Q}{A} = \frac{q}{y_0} = \frac{0.000230}{0.001} = 0.230 \text{ m/s}$$

Sec. 10.1: $$\mathbf{R} = \frac{R_h V}{\nu} = \frac{(0.001)0.230}{1.139\times10^{-6}} = 201$$

As $\mathbf{R} < \mathbf{R}_c = 500$, flow is laminar and Eqs. (10.14) and (10.15) apply. Therefore $q = 0.000230$ m³/s per meter ***ANS***

(*b*) Laminar flow Eq. (10.14) at the surface, where $y = y_0$:

$$v = \frac{g}{\nu}\frac{y_0^2}{2}S = \frac{9.81}{1.139\times10^{-6}}\frac{(0.001)^2}{2}0.08 = 0.345 \text{ m/s} \quad \textbf{ANS}$$

EXERCISES

10.8.1 Eastern lubricating oil (SAE 30) at 100°F flows down a flat plate 12 ft wide. (*a*) What is the maximum rate of discharge at which laminar flow may be ensured, assuming that the critical Reynolds number is 500? (*b*) What should be the slope of the plate to secure a depth of 8 in at this flow rate?

10.8.2 Eastern lubricating oil (SAE 30) at 30°C flows down a flat plate 3 m wide. (*a*) What is the maximum rate of discharge at which laminar flow may be ensured, assuming that the critical Reynolds number is 500? (*b*) What should be the slope of the plate to secure a depth of 150 mm at this flow rate?

10.8.3 At what rate (cfs/ft of width) will 70°F water flow in a wide and shallow, smooth channel on a slope of 0.00010 if the depth is 0.02 ft? Assume laminar flow and justify this assumption by computing the Reynolds number.

10.8.4 At what rate (L/s per meter of width) will water at 20°C flow in a wide and shallow, smooth channel on a slope of 0.0004, if the depth is 6.0 mm? Assume laminar flow and justify the assumption by computing the Reynolds number.

10.9 SPECIFIC ENERGY AND ALTERNATE DEPTHS OF FLOW IN RECTANGULAR CHANNELS

For *any* cross-section shape, the ***specific energy*** E at a particular section is defined as the energy head referred to the channel bed as datum. Thus

$$E = y + \alpha\frac{V^2}{2g} \tag{10.16}$$

where α is the kinetic energy correction factor (Sec. 5.1), which accounts for velocity variations across the section. Friction at the channel walls reduces velocities near the wetted perimeter, as indicated in Fig. 10.7 for which $\alpha = 1.105$. As noted in Sec. 10.4, the value of α is usually assumed to be 1.0; for typical velocities this results in only a small error in E.

For *rectangular* channels that we discuss in this section, provided they are not unusually narrow so that α is large (Sec. 5.1), a representative average value of the flow q per unit width can be expressed as $q = Q/b$. The average velocity $V = Q/A = qb/by = q/y$ and so Eq. (10.16) with $\alpha = 1$ can be expressed as

$$E = y + \frac{1}{2g}\frac{q^2}{y^2} \tag{10.17}$$

Let us consider how E will vary with y if q remains constant. Physically, this situation would occur if the slope of a rectangular channel could be changed with the flow rate remaining constant. Manning's equation (Eq. 10.8) indicates that on a steep slope, with a given flow rate, the normal depth of flow y_0 will be relatively small in contrast to a larger depth on a flatter slope. That this is so can be seen more easily for a wide and shallow channel by writing the Manning

equation in terms of the flow q per unit width, noting that $A = by$ and $R_h \approx y$ for $b \gg y$. Thus, for uniform flow ($y = y_0$), in BG units

Wide and shallow, uniform flow:

$$q = \frac{Q}{b} = \frac{1.486}{nb} A R_h^{2/3} S_0^{1/2} = \frac{1.486}{n} \frac{by_0}{b} y_0^{2/3} S_0^{1/2} = \frac{1.486}{n} y_0^{5/3} S_0^{1/2} \qquad (10.18)$$

For the case of constant q, we can restate Eq. (10.17) as

$$(E - y)y^2 = \frac{q^2}{2g} = \text{constant} \qquad (10.19)$$

A plot of E vs y is hyperbola-like with asymptotes $(E - y) = 0$ (that is, $E = y$) and $y = 0$. Such a curve, shown in Fig. 10.12, is known as the ***specific-energy diagram.*** Actually each different value of q will give a different curve, as shown in Fig. 10.12. For a particular q, we see there are two possible values of y for a given value of E. These are known as ***alternate depths.*** Equation (10.19) is a cubic equation with three roots, the third root being negative has no physical meaning. The two alternate depths represent two totally different flow regimes—slow and deep on the upper limb of the curve and fast and shallow on the lower limb of the curve. Point C represents the dividing point between the two regimes of flow. At C, for a given q, the value of E *is a minimum* and we refer to the flow at this point as ***critical flow.*** The depth of flow at that point is the ***critical depth*** y_c and the velocity is the ***critical velocity*** V_c. A relation for critical depth in a wide rectangular channel can be found by differentiating E of Eq. (10.17) with respect to y to find the value of y for which E is a minimum. Thus

$$\frac{dE}{dy} = 1 - \frac{q^2}{gy^3} \qquad (10.20)$$

and when E is a minimum, $y = y_c$ and $dE/dy = 0$, so that

$$0 = 1 - \frac{q^2}{gy_c^3}, \quad \text{or} \quad q^2 = gy_c^3 \qquad (10.21)$$

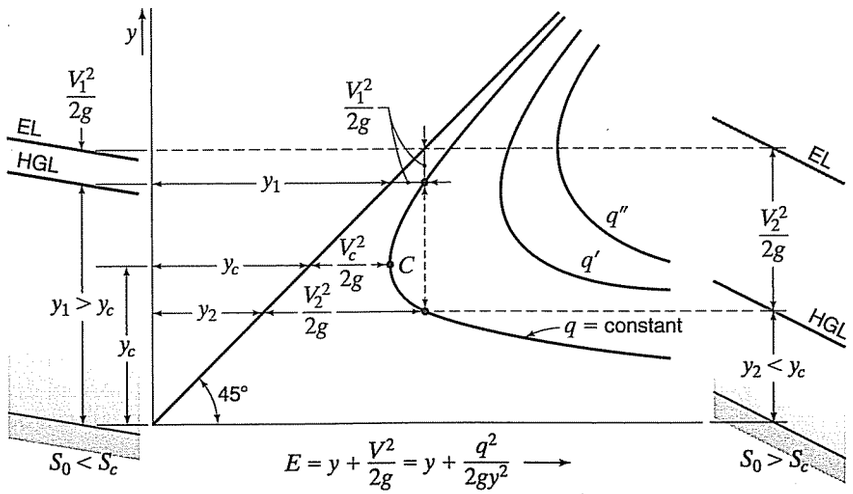


Figure 10.12
Specific-energy diagram for three constant rates of discharge in a rectangular channel. (Bed slopes are greatly exaggerated.)

Substituting $q = Vy = V_c y_c$ gives

$$V_c^2 = gy_c \quad \text{and} \qquad V_c = \sqrt{gy_c} = \frac{q}{y_c} \qquad (10.22)$$

where the subscript c indicates critical flow conditions (minimum specific energy for a given q).

We can also express Eq. (10.22) as

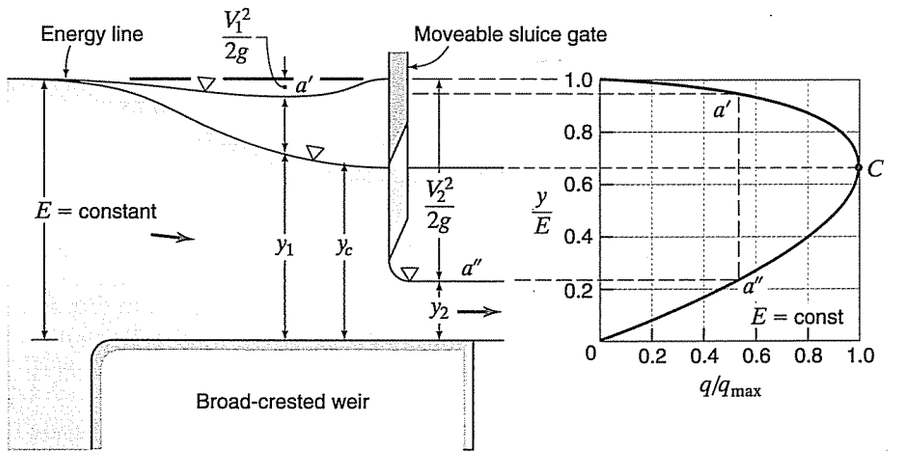
$$y_c = \frac{V_c^2}{g} = \left(\frac{q^2}{g}\right)^{1/3} \qquad (10.23)$$

From Eq. (10.22),
$$\frac{V_c^2}{2g} = \tfrac{1}{2} y_c \qquad (10.24)$$

So
$$E_c = E_{\min} = y_c + \frac{V_c^2}{2g} = \tfrac{3}{2} y_c \qquad (10.25)$$

and
$$y_c = \tfrac{2}{3} E_c = \tfrac{2}{3} E_{\min} \qquad (10.26)$$

A different approach to alternate depths is to solve Eq. (10.19) for q and note the variation in q for changing values of y for a constant value of E. Physically this situation is encountered when water flows from a larger reservoir of constant surface elevation over a high, frictionless, broad-crested weir provided with a moveable sluice gate near its downstream end (Fig. 10.13*a*). As the gate is opened the flow rate increases until the opening becomes just large enough for



(*a*) Side view of flow over a broad-crested weir (*b*) The dimensionless discharge curve

Figure 10.13
Variation of depth y and discharge per unit width q for constant specific energy E.

critical depth to occur. With further opening of the gate, there is no increase in flow rate. As long as the water impinges on the gate (Fig. 10.13*a*), the flow is subcritical (a') upstream of the gate and supercritical (a'') downstream of the gate.

Rewriting Eq. (10.19) gives

$$q = y\sqrt{2g(E - y)} \tag{10.27}$$

This is the equation of the curve that is shown in dimensionless form in Fig. 10.13*b*. It is seen that maximum discharge for a given specific energy occurs when the depth is between $0.6E$ and $0.7E$. This may be established more exactly by differentiating Eq. (10.27) with respect to y and equating to zero. Thus

$$\frac{dq}{dy} = \sqrt{2g}\left(\sqrt{E - y} - \frac{1}{2}\frac{y}{\sqrt{E - y}}\right) = 0$$

from which

$$y_c = \tfrac{2}{3}E \tag{10.28}$$

where y_c is called the ***critical depth*** *for the given specific energy*. This equation is identical to Eq. (10.26). Thus there is a maximum value of q for a given E as indicated by point C of Fig. 10.13*b*. We often refer to this curve as the ***discharge curve.*** The flow depicted by the upper limb of the curve has characteristics similar to those of the upper limb of that in Fig. 10.12. Similarly, the flow depicted by the lower limb of the curve has characteristics similar to the lower limb of that of Fig. 10.12. The significance of these two regimes of flow is discussed in Sec. 10.10. Point C of Fig. 10.13*b* represents critical flow conditions.

The discharge is a maximum when $y = y_c$, as indicated on the discharge curve of Fig. 10.13*b*. We can obtain an expression for q_{max} by substituting $E = 1.5y_c$ from Eq. (10.28) into Eq. (10.27), to yield

$$q_{max} = \sqrt{gy_c^3} \tag{10.29}$$

In Fig. (10.12) we can see that when the flow is near critical, a small change in specific energy results in a large change in depth. Hence flow at or near critical depth is unstable and there will be an undulating water surface. Because of this it is undesirable to design channels to flow near critical depth.

We summarize the foregoing discussion as axioms of open-channel flow related to conditions at a given section in a *rectangular channel:*

1. A flow condition, i.e., a certain rate of discharge flowing at a certain depth, is completely specified by any two of the variables y, q, V, and E, except the combination q and E, which yields in general two alternate stages of flow.
2. For any value of E a critical depth exists, given by Eq. (10.26), for which the flow is a maximum.
3. For any value of q a critical depth exists, given by Eq. (10.23), for which the specific energy is a minimum.
4. When flow occurs at critical depth, both Eq. (10.22) and (10.26) are satisfied and the velocity head is one-half the depth.
5. For any flow condition other than critical, an alternate stage exists at which the same rate of discharge is carried by the same specific energy. We can find the alternate depth from either the specific-energy diagram

(Fig. 10.12) or the discharge curve (Fig. 10.13*b*), by extending a vertical line to the alternate limb of the curve. Analytically, the alternate depth is found by solving Eq. (10.17).



SAMPLE PROBLEM 10.6 Water in a rectangular channel flows 0.5 m deep at a velocity of 6 m/s. Find (*a*) the alternate depth, (*b*) the critical depth for this discharge, and (*c*) the percentage by which the specific energy exceeds the minimum.

Solution

(*a*) Eq. (10.16): $$E = 0.5 + \frac{6^2}{2(9.81)} = 2.33 \text{ m}$$

$$q = Vy = 6(0.5) = 3 \text{ m}^2\text{/s per m}$$

Eq. (10.17): $$2.33 = y + \frac{3^2}{2(9.81)y^2} \tag{1}$$

i.e., $$y^3 - 2.33y^2 + 0.459 = 0$$

We can solve this cubic equation with a polynomial solver (Appendixes B and C.1). Or, knowing that 0.5 m is one root of this equation, we can in effect divide through by $(y - 0.5)$ and solve the resulting quadratic by obtaining the other two roots from Eq. (B.9):

$$y = -0.409 \text{ m (meaningless)} \quad \text{or} \quad y = 2.24 \text{ m} \qquad \textbf{\textit{ANS}}$$

(*b*) Eq. (10.23): $$y_c = \left(\frac{3^2}{9.81}\right)^{1/3} = 0.972 \text{ m} \qquad \textbf{\textit{ANS}}$$

(*c*) Eq. (10.25): $$E_c = E_{min} = 1.5y_c = 1.5(0.972) = 1.458 \text{ m}$$

$$\frac{E}{E_{min}} = \frac{2.33}{1.458} = 1.602, \quad \text{so exceedance} = 60.2\% \qquad \textbf{\textit{ANS}}$$

Note: A popular alternative manual approach is to solve (1) by trial and error (see Sample Probs. 3.5 and 5.8); values of y are tried until the right side equals the left. As there are two meaningful roots, helpful guidance will be provided by first finding y_c and E_{min} and making a sketch of the specific energy diagram. Some programmable scientific calculators have equation solvers, which in effect automate the trial and error (Appendixes C.1 and D.1). Equation solving software packages can also conveniently solve such equations (Appendixes C.2–C.3 and D.2–D.3).

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

EXERCISES

10.9.1 A long straight rectangular channel 12 ft wide is observed to have a wavy water surface at a depth of about 5 ft (Fig. X10.9.1). Estimate the rate of discharge.

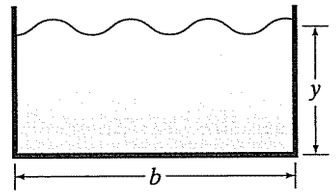


Figure X10.9.1

10.9.2 A long straight rectangular channel 4 m wide is observed to have a wavy water surface at a depth of about 3 m (Fig. X10.9.1). Estimate the rate of discharge.

10.9.3 Water flows down a wide and shallow rectangular channel of concrete ($n = 0.015$) laid on a slope of 0.004. Find the depth and rate of flow in SI units for critical conditions in this channel.

10.10 SUBCRITICAL AND SUPERCRITICAL FLOW

In Sec. 10.9 we referred to the upper and lower portions of the specific-energy diagram (Fig. 10.12) and the discharge curve (Fig. 10.13). The upper limb of those curves, where velocities are less than critical, represent ***subcritical*** (also known as tranquil or upper-stage) flow, while the lower limb of the curves where velocities are greater than critical represent ***supercritical*** (also known as rapid or lower-stage) flow. We discussed how to identify the critical point separating these two portions in Sec. 10.9.

The bed slope required to give uniform flow at critical depth ($y_0 = y_c$) for a given discharge is known as the ***critical slope*** S_c. Note that S_c varies with discharge. We obtain an expression for the critical slope of a *wide and shallow* rectangular channel ($R_h = y$) when we combine Eq. (10.23) for critical flow with Eq. (10.18) for uniform flow, eliminating q as

Wide and shallow, BG units:
$$S_c = \left(\frac{n}{1.486}\right)^2 \frac{g}{y_c^{1/3}} \tag{10.30}$$

Substituting for y_c from Eq. (10.23), we can also write this as

Wide and shallow, BG units:
$$S_c = \left(\frac{n}{1.486}\right)^2 \frac{g^{10/9}}{q^{2/9}} \tag{10.31}$$

When using SI units, we simply replace the 1.486 by 1 in these equations.

For channels with cross sections other than wide and shallow, an expression for the critical slope is more complex than the above. However, we seldom need to calculate S_c because, as we shall see, we can more easily determine how S_0 compares with S_c by comparing y_0 with y_c.

If the bed slope $S_0 > S_c$, the slope is known as a ***steep slope*** for the given discharge. Normal depth for uniform flow on such a slope will be less than

critical depth and hence normal flow will be supercritical. In contrast, if $S_0 < S_c$, the normal depth will be greater than critical and normal flow is subcritical. Such a slope is referred to as a ***mild slope.*** By referring to Eq. (10.30), we see that the hydraulic steepness of a channel slope is determined by more than its elevation gradient. A steep slope for a channel with a smooth lining could be a mild slope for the same flow with a rough lining. Even for a given channel with a given boundary roughness, the slope may be mild for a low rate of discharge and steep for a higher one.

From Sec. 7.4 we recall that the Froude number is defined as $V/\sqrt{gL}$. If for rectangular channels the depth of flow is used to represent the significant length parameter in the Froude number (that is, $\mathbf{F} = V/\sqrt{gy}$), we find by comparing this with Eq. (10.22) that the flow is critical if $\mathbf{F} = 1.0$, the flow is subcritical if $\mathbf{F} < 1.0$, and the flow is supercritical if $\mathbf{F} > 1.0$.

Another convenient way to determine the type of flow in rectangular channels is to compare the velocity head with half the depth. Figure 10.12 clearly indicates that if $V^2/2g > y/2$ the flow is supercritical, and if $V^2/2g < y/2$ the flow is subcritical. Important characteristics of subcritical, critical, and supercritical flow in rectangular channels are summarized in Table 10.2.

TABLE 10.2 **Characteristics of subcritical, critical, and supercritical flow in rectangular channels**

Characteristic	Subcritical	Critical	Supercritical
Depth of flow, y	$y > y_c$	$y = y_c = \left(\frac{q^2}{g}\right)^{1/3}$	$y < y_c$
Velocity of flow, V	$V < V_c$	$V = V_c = \sqrt{gy}$	$V > V_c$
Slope for uniform flow, S_0	Mild slope $S_0 < S_c$	Critical slope $S_0 = S_c$ [Eq. (10.30) if wide and shallow]	Steep slope $S_0 > S_c$
Froude number, $\mathbf{F} = \frac{V}{\sqrt{gy}} = \frac{q}{\sqrt{gy^3}}$	$\mathbf{F} < 1.0$	$\mathbf{F} = 1.0$	$\mathbf{F} > 1.0$
Disturbance waves (Sec. 10.20)	Will propagate in all directions	Will hold fast, not propagate upstream	Will form standing wave with $\sin\beta = c/V$ downstream only
Velocity head compared with half-depth	$\frac{V^2}{2g} < \frac{y}{2}$	$\frac{V^2}{2g} = \frac{y}{2}$	$\frac{V^2}{2g} > \frac{y}{2}$
Can be followed by a hydraulic jump? (Sec. 10.16)	No	No	Yes

EXERCISES

10.10.1 A rectangular channel 50 ft wide ($n = 0.015$) carries a flow of 375 cfs. For this flow, find the critical depth, the critical velocity, and the critical bed slope.

10.10.2 A 10-ft-wide rectangular channel ($n = 0.020$) carries 400 cfs. Find the critical bed slope from Eq. (10.30) and by using the Manning formula. Which of these answers is correct, and why? What is the percentage error in the incorrect answer?

10.11 CRITICAL DEPTH IN NONRECTANGULAR CHANNELS

For nonrectangular channels, the definition of the Froude number is

$$\mathbf{F} = \frac{V}{\sqrt{gy_h}}$$

where we define $y_h = A/B$ as the ***hydraulic depth*** (or ***hydraulic mean depth***), where A is the cross-sectional flow area and B is the width of the flow area at the water surface. The expression just given for Froude number is the same as $\mathbf{F} = Q\sqrt{B/(gA^3)}$.

Let us now consider an irregularly shaped flow section (Fig. 10.14) of area A carrying a flow Q. Thus Eq. (10.17) becomes

$$E = y + \frac{Q^2}{2gA^2} \tag{10.32}$$

Differentiating with respect to y and setting to zero to find y_c, that is, the value of y for which E is a minimum,

$$\frac{dE}{dy} = 1 - \frac{Q^2}{2g}\left(\frac{2}{A^3}\frac{dA}{dy}\right) = 0$$

As A may or may not be a reasonable function of y, it is helpful to observe that $dA = B\,dy$, and thus $dA/dy = B$, ***the width of the water surface.*** Substituting this

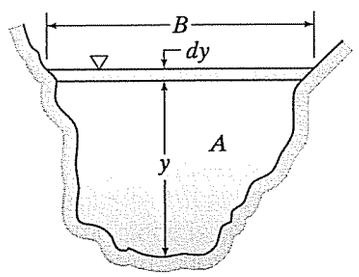


Figure 10.14

in the preceding expression results in

$$\frac{Q^2}{g} = \left(\frac{A^3}{B}\right)_{y=y_c} \tag{10.33}$$

as the equation that must be satisfied for critical flow. For a given cross section, the right-hand side is a function of y only. We generally need a trial-and-error solution to find y_c, which is the value of y that causes A and B to satisfy Eq. (10.33). Once we find this, we call A and B A_c and B_c. It is easy to confirm that Eq. (10.33) causes $\mathbf{F} = 1$.

We can next solve for V_c, the critical velocity in the irregular channel, by observing that $Q = A_cV_c$. Substituting this in Eq. (10.33) yields

$$\frac{V_c^2}{g} = \frac{A_c}{B_c}, \quad \text{or} \quad V_c = \sqrt{\frac{gA_c}{B_c}} = \sqrt{g(y_h)_c} \tag{10.34}$$

If the channel is rectangular, $A_c = B_cy_c$, and Eq. (10.34) reduces to Eq. (10.22); also, then $Q = B_cq$, and Eq. (10.33) reduces to Eq. (10.23).

The cross sections most commonly encountered in open-channel hydraulics are not rectangular but are ***trapezoidal*** or part-full circular sections. We discussed the latter in Sec. 10.17. For the very common trapezoidal cross section, practicing hydraulic engineers in the past made use of numerous tables, curves, and computer programs to avoid tedious repeated trial-and-error solutions of Eq. (10.33) for y_c. Now it is more convenient to replace A and B by the appropriate functions of y_c, b, and m (defined as in Fig. 10.8), and to solve the resulting equation

$$\frac{Q^2}{g} = \frac{(by_c + my_c^2)^3}{b + 2my_c}$$

using an equation solver in a programmable scientific calculator or in a mathematics software package.

SAMPLE PROBLEM 10.7 In Fig. S10.7 water flows uniformly at a steady rate of 14.0 cfs in a very long triangular flume that has side slopes 1:1. The flume is on a slope of 0.006, and $n = 0.012$. (*a*) Is the flow subcritical or supercritical? (*b*) Find the relation between $V_c^2/2g$ and y_c for this channel.

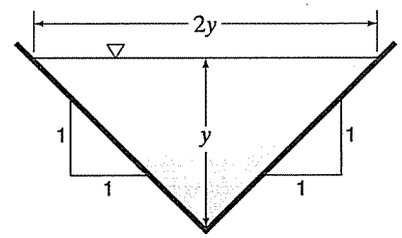


Figure S10.7

Solution

(*a*)
$$A = \tfrac{1}{2}(2y)y = y^2$$
$$P = 2\sqrt{2}y = 2.83y$$
$$R_h = \frac{A}{P} = 0.354y$$

Eq. (10.8*a*) for uniform flow: $$14 = \frac{1.486}{0.012} y_0^2 (0.354y_0)^{2/3}(0.006)^{1/2}$$

from which $y_0 = 1.494$ ft. When $y = y_c$, from Eq. (10.33):

$$\frac{(14)^2}{32.2} = \frac{(y_c^2)^3}{2y_c}$$

and
$$y_c = 1.648 \text{ ft}$$

Since $1.648 \text{ ft} = y_c > y_0 = 1.494$ ft, the flow is supercritical. ***ANS***

Note: If the data in this problem had been given in SI units rather than in BG units, the procedure for solution would have been the same except that Eq. (10.8*b*) would have been used instead of Eq. (10.8*a*).

(*b*) Eq. (10.34):
$$\frac{V_c^2}{g} = \frac{A_c}{B_c} = \frac{y^2}{2y} = \frac{y}{2}$$

so
$$\frac{V_c^2}{2g} = \frac{y_c}{4} \quad \textit{ANS}$$

Consequently we see that the relation between $V^2/2g$ and y for critical flow conditions depends on the geometry of the flow section. If the vertex angle of the triangle had been different from 90°, the relation would have been different.

EXERCISES

10.11.1 A flow of 12 cfs of water is carried in a 90° triangular flume built of planed timber ($n = 0.011$). Find the critical depth and the critical slope.

10.11.2 A flow of 0.28 m^3/s of water is carried in a 90° triangular flume built of planed timber ($n = 0.011$). Find the critical depth and the critical slope.

10.11.3 A circular conduit of well-laid (smooth) brickwork when flowing half full is to carry 340 cfs at a velocity of 8.2 fps. (*a*) What will be the necessary fall per mile? (*b*) Will the flow be subcritical or supercritical?

10.11.4 A trapezoidal channel has a 4-m-wide bed and $m = 2$ (i.e., $2H{:}1V$). What is the critical flow depth when it is carrying 85 m^3/s?

10.12 OCCURRENCE OF CRITICAL DEPTH

When flow changes from subcritical to supercritical or vice versa, the depth must pass through critical depth. In the former, the phenomenon gives rise to what is known as a ***control section.*** In the latter a hydraulic jump (Sec. 10.18) usually occurs. In Fig. 10.15 is depicted a situation where the flow changes from subcritical to supercritical. Upstream of the break in slope there is a ***mild*** slope, the flow is subcritical, and $y_{0_1} > y_c$. Downstream of the break there is a ***steep*** slope, the flow is supercritical, and $y_{0_2} < y_c$. At the break in slope the depth passes through critical depth. This point in the stream is referred to as a control section since the depth at the break controls the depth upstream. A similar situation occurs when water from a reservoir enters a canal in which the uniform depth is less than critical. In such an instance (Fig. 10.16), the depth passes through critical depth in the vicinity of the entrance to the canal. Therefore, this section is also known as a control section. By measuring the depth at a control section, we can compute a reasonably accurate value of Q by applying Eq. (10.23) for rectangular channels or Eq. (10.33) for nonrectangular channels.

At a control section there is a specific relationship between the stage and the discharge, regardless of the channel roughness and slope. This indicates that such locations are desirable sites for gaging stations. It also means that when the upstream or downstream flow conditions change, the transmission of such changes through a control section are restricted. Note that not only critical depth creates a control section; control structures, such as sluice gates and orifices, may also be used to create control sections.

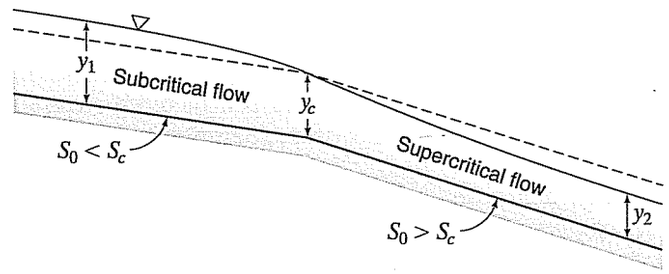


Figure 10.15
Change in flow from subcritical to supercritical at a break in slope.

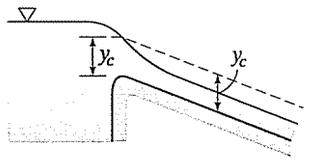


Figure 10.16
Hydraulic drop entering a steep slope.

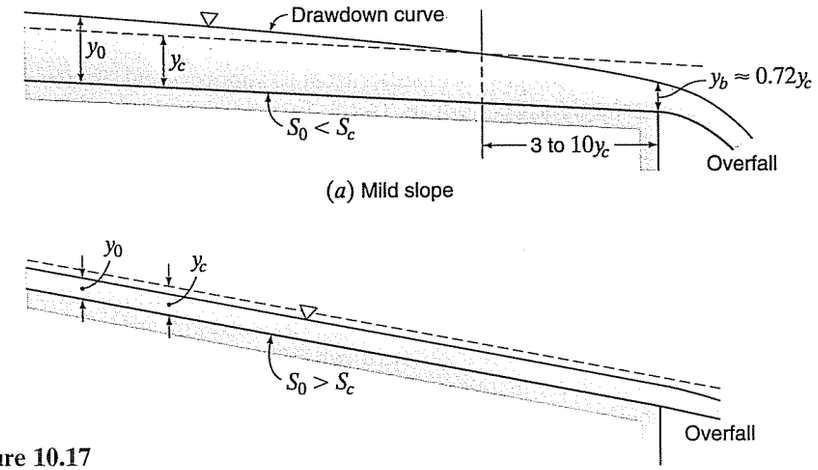


Figure 10.17
Free overfall.

Another instance where critical depth occurs is that of a ***free overfall*** (Fig. 10.17*a*) with subcritical flow in the channel prior to the overfall. Since friction produces a constant diminution in energy in the direction of flow, it is obvious that at the point of overfall the total energy must be less than at any point upstream. As critical depth is the value for which the specific energy is a minimum, one would expect critical depth to occur at the overfall. However, the value for the critical depth is derived on the assumption that the water is flowing in straight lines. In the free overfall, gravity creates a curvature of the streamlines, with the result that the depth at the brink is less than critical depth. Experimenters have found that the depth at the brink $y_b \approx 0.72y_c$. Also, critical depth generally occurs upstream of the brink a distance of somewhere between $3y_c$ and $10y_c$. If the flow is supercritical, there is no drawdown curve (Fig. 10.17*b*).

Critical depth *may* occur in a channel if the bottom is humped or if the sidewalls are moved in to form a contracted section. However, in such cases critical depth will not always occur (Sample Prob. 10.8).

10.13 HUMPS AND CONTRACTIONS

Let us examine the case of a hump or weir in a rectangular channel (Fig. 10.18). We shall assume the hump is streamlined and neglect head loss. In Fig. 10.18*a* we see a small hump. If the flow upstream of the hump is *subcritical,* there will

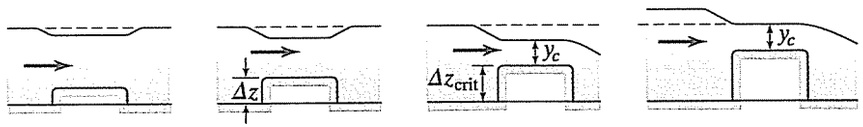


Figure 10.18
Subcritical flow over a hump in a rectangular channel. Flow rate is the same in all four cases. Δz = hump height; Δz_{crit} = critical hump height.

be a slight depression in the water surface over the hump. Inspection of the subcritical (upper) portion of the specific-energy diagram (Fig. 10.12) confirms this statement. The hump (locally raised bed) causes a drop in specific energy E, since the energy line is unchanged. A decrease in E with unchanged q is accompanied by a decrease in y that exceeds the decrease in E, so there must be a depression in the water surface over the hump. If the height of the hump Δz is increased with no change in q, there will be a further drop in the water surface over the hump (Fig. 10.18*b*) for the same reason. Further increases in hump height create further depression of the water surface over the hump until finally the depth on the hump becomes critical (Fig. 10.18*c*). We shall call the minimum height of the hump that causes critical depth the ***critical hump height,*** Δz_{crit}. If the hump is made still higher (Fig. 10.18*d*), critical depth remains on the hump and the depth upstream of the hump increases (tapering off with distance as in Fig. 10.21) until it gains sufficient energy to be able to flow over at the same rate. The more the hump is raised, the greater will the upstream depths become, and the further upstream will these effects extend. We refer to this phenomenon as ***damming action.*** With damming action, immediately downstream of y_c the flow must be supercritical (to comply with Fig. 10.20). In such a situation, the hump, if it has a flat top, becomes a ***broad-crested weir*** (Sec. 11.12).

Let us next consider contractions in rectangular channels. We shall again assume them to be streamlined so that we can neglect any head loss they may cause. Contractions change q, the flow per unit width, and we note that curves for increasing q move to the right on Fig. 10.12. So it follows from Fig. 10.12 that a contraction, which causes q to increase, has a similar effect on y as does a hump, which causes E to decrease. If the approaching flow is subcritical, a small contraction will cause a slight depression in the water surface through the contraction. Further contraction creates further depression in the water surface until critical depth occurs in the contraction. This happens when point C on Fig. 10.12 has energy E. Upon further contraction, the depth remains critical in the contracted section and upstream depths increase. However, unlike for humps, the depth in the contracted section continues to increase. This is because the critical depth there depends upon q via Eq. (10.23), and q increases with contraction. Figure 10.12 indicates how contraction beyond the onset of critical depth requires the flow to have extra energy, and this again comes from the damming action, which causes the upstream water surface to rise for a considerable distance. We shall here call the maximum width (i.e., minimum contraction) at which critical depth occurs in the contracted section the ***critical contracted width.***

With *supercritical* flow, humps and contractions behave differently than they do with subcritical flow according to the trends indicated by the supercritical (lower) limbs of Figs. 10.12 and 10.13. Thus, when the flow is supercritical, the water depth at the hump or contraction must be slightly *greater* than the depth immediately upstream. And as the hump height or amount of contraction is made larger, the water depth at the hump or contraction *increases,* until it reaches critical depth. Beyond that, further enlargement of the hump height or amount of contraction causes damming action. However, as we shall see in Secs. 10.16–10.19, the damming action effects on the upstream water surface will be much less extended and very different from those for subcritical flow.

A hump may be combined with a contraction to give a section not unlike a venturi meter (Sec. 11.7). We may apply the same principles discussed above for the hump and the contraction to such a situation. It is important to note that the hump changes E and the contraction changes q in the *single* equation (10.17) to be solved; using a separate equation for each effect would be incorrect. Accompanying such calculations by a labeled sketch of the specific-energy diagram is extremely helpful.

SAMPLE PROBLEM 10.8 In Fig. S10.8 uniform flow of water occurs at 135 cfs in a 20-ft-wide rectangular channel at a depth of 2.00 ft. (*a*) Is the flow subcritical or supercritical? (*b*) If a hump of height $\Delta z = 0.30$ ft is placed across the bottom of the channel, calculate the water depth on the hump, and the change in the water surface level at the hump. (*c*) If the hump height is raised to $\Delta z = 0.60$ ft, what then are the water depths immediately upstream and downstream of the hump? (*d*) If the 0.30-ft hump is accompanied by a local contraction to 18 ft, find the water depth on the hump. In all cases neglect head losses over the hump and through the contraction.

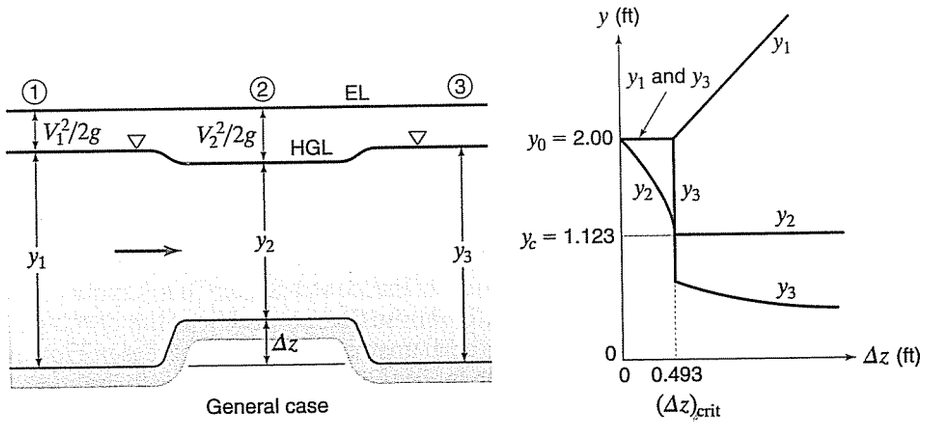


Figure S10.8

Solution

(*a*) $$q = 135/20 = 6.75 \text{ cfs/ft}$$

Eq. (10.23): $$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(6.75)^2}{32.2}\right]^{1/3} = 1.123 \text{ ft}$$

Since the normal depth (2.00 ft) is greater than the critical depth, the flow is subcritical and the channel slope is mild. ***ANS***

▦: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

(*b*) First find the critical hump height. Write the energy equation between sections 1 and 2, assuming critical flow on the hump.

Eq. (10.17): $$E_1 = 2.00 + \frac{1}{2(32.2)}\left(\frac{6.75}{2.00}\right)^2 = 2.18 \text{ ft}$$

Energy: $$E_1 = \Delta z_{crit} + (E_{min})_2$$

Using Eq. (10.25): $$2.18 = \Delta z_{crit} + 1.5(1.123)$$

$$\Delta z_{crit} = 0.493 \text{ ft}$$

Thus the minimum-height hump that will produce critical depth on the hump is 0.493 ft.

Since the actual hump height, $\Delta z = 0.30$ ft, is less than the critical hump height, 0.493 ft, critical flow does not occur on the hump and there is no damming action.

Let us now find the depth y_2 on the hump.

Energy in ft of head from section 1 to section 2:

$$2.18 = 0.30 + y_2 + \frac{1}{2(32.2)}\left(\frac{6.75}{y_2}\right)^2$$

We can solve this with an equation solver in a programmable scientific calculator (Appendixes C.1 and D.1) or in computer software (Appendixes C.2–C.3 and D.2–D.3). Alternatively, we can rearrange it into a cubic equation as follows,

$$y^3 - 1.877y^2 + 0.707 = 0$$

We can solve this cubic equation by trial and error (see Sample Probs. 3.5 and 5.8) followed by Eq. (B.9), or with a polynomial solver (Appendixes B and C.1). We obtain three roots, 1.601 ft, 0.817 ft, and a negative answer (−0.541 ft) that has no physical meaning. Since $\Delta z < \Delta z_{crit}$, the flow on the hump must be subcritical; that is, $y_2 > y_c$. Hence $y_2 = 1.601$ ft. ***ANS***

The change in the water surface over the hump = 2.00 − (0.30 + 1.60) = 0.10 ft drop. ***ANS***

(*c*) In this case the hump height $\Delta z = 0.60$ ft, which is greater than the critical hump height. Hence there will be damming action and critical depth ($y_c = 1.123$ ft) will occur on the hump. Writing the energy equation for this case, from section 1 to section 2:

$$y_1 + \frac{1}{2(32.2)}\left(\frac{6.75}{y_1}\right)^2 = 0.60 + 1.5(1.123) = 2.28 \text{ ft}$$

Solving the resulting cubic for y_1 gives three roots, 2.13 ft, 0.660 ft, and a negative answer (−0.504 ft) that has no physical meaning. In this case damming action occurs and the depth y, upstream of the hump is 2.13 ft. On the hump the depth passes through critical depth of 1.123 ft and just downstream of the hump the depth will be 0.660 ft. ***ANS***

The depth will then increase in the downstream direction following an M_3 water surface profile (Fig. 10.20) until a hydraulic jump (Sec. 10.18) occurs to

return the depth to the normal uniform flow depth of 2.00 ft. The depth just downstream of the hump cannot be 2.13 ft, because then there would be no way for the flow to dissipate its extra energy and return to normal depth; this is explained in Sec. 10.16 and Fig. 10.20.

This higher type of hump is commonly built of concrete and is used for flow measurement purposes. We discuss such structures under broad-crested weirs, in Sec. 11.12. The right-hand sketch of Fig. S10.8 shows the relation between the hump height Δz, and the water depths y_1, y_2, and y_3 for the condition where $Q = 135$ cfs.

(*d*) At section 2: $q = 135/18 = 7.50$ cfs/ft

$$y_{c2} = \left(\frac{7.50^2}{32.2}\right)^{1/3} = 1.204 \text{ ft}$$

$$(E_{min})_2 = 1.5(1.204) = 1.807 \text{ ft}$$

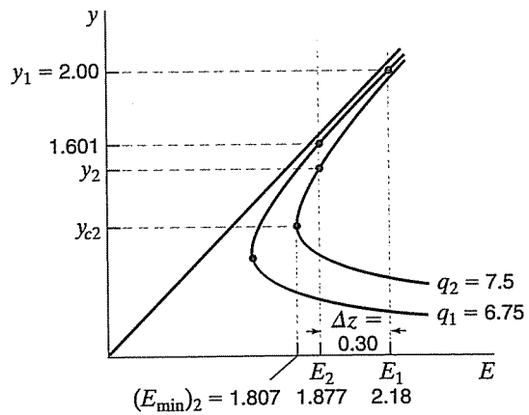
$$\Delta z_{crit} = E_1 - (E_{min})_2 = 2.18 - 1.807 = 0.370 \text{ ft}$$

So Δz is less than Δz_{crit}; therefore the combination of hump and contraction will not cause critical depth or damming action.

Energy from section 1 to section 2:

$$E_2 = E_1 - \Delta z = 2.18 - 0.300 = 1.877 \text{ ft}$$

Eq. (10.17): $$1.877 = y_2 + \frac{1}{2(32.2)}\left(\frac{7.50}{y_2}\right)^2$$



From the sketch of the specific-energy diagram, we expect $1.204 < y_2 < 1.601$ ft. The sketch helps direct manual trials.

By trials, by polynomial solver, or by equation solver, $y_2 = 1.476$ ft. ***ANS***

TABLE 10.3 **Occurrence of damming action**

Cause	Damming action occurs when:
Hump only (constant q)	$\Delta z > \Delta z_{crit} = E_u - E_{min} = E_u - \frac{3}{2}y_c$
Contraction only (constant E)	$b_2 < b_{crit}$, so $E_u < (E_{min})_2 = \frac{3}{2}y_{c2}$
Hump with contraction	Δz and b_2 are such that $E_u - \Delta z < (E_{min})_2 = \frac{3}{2}y_{c2}$

Subscript u represents conditions with no constrictions.
Subscript 2 represents a station *at* the constriction.

An unnecessary difficulty sometimes baffles students trying to solve problems like part (*c*) of Sample Prob. 10.8. It happens if they forget to first calculate Δz_{crit}, and forget that damming action may have occurred. When they try to solve for the water depth on the hump, they cannot get a solution, or get only negative and imaginary roots. This is because without damming action E on the hump is less than E_{min} (see Fig. 10.12), and they are trying to solve an impossible situation.

Similar logic can be applied to solve problems involving depressions in the channel bed (negative humps) and channel expansions (negative contractions). Of course opposite effects are produced to those that occur with positive humps and contractions.

Because identifying damming action is an important key to solving flows past constrictions, and the relevant relationships can be confusing, we summarize situations when damming action will occur in Table 10.3; with the contrary inequalities it will not occur. Students are strongly urged to sketch appropriate specific-energy diagrams, like the one at the end of Sample Prob. 10.8, to help clarify situations.

EXERCISES

10.13.1 A rectangular channel 3 m wide carries 4 m³/s of water in subcritical uniform flow at a depth of 1.2 m. A frictionless hump is to be installed across the bed. Find the critical hump height (i.e., the minimum hump which causes y_c on it).

10.13.2 Suppose that the depth of uniform flow in a 4-ft-wide rectangular channel is 1.10 ft. Find the change in water-surface elevation caused by a 1-ft-wide bridge pier placed in the middle of the channel. The flow rate is 50 cfs.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.13.3 Suppose that the depth of uniform flow in a 1.2-m-wide rectangular channel is 270 mm. Find the change in water-surface elevation caused by a 300-mm-wide bridge pier placed in the middle of the channel. The flow rate is 1.1 m^3/s.

10.13.4 A rectangular channel 12 ft wide carries 24 cfs in uniform flow at a depth of 0.80 ft. Find the local change in water-surface elevation caused by a frictionless hump 0.12 ft high across the floor of the channel.

10.14 NONUNIFORM, OR VARIED, FLOW

As a rule, uniform flow is found only in artificial channels of constant shape and slope, although even under these conditions the flow for some distance may be nonuniform, as shown in Fig. 10.1. But with a natural stream, the slope of the bed and the shape and size of the cross section usually vary to such an extent that true uniform flow is rare. So we can expect the application of the equations given in Sec. 10.2 to natural streams to yield results that are only approximations to the truth. In order to apply these equations at all, we must divide the stream into lengths (reaches) within which the conditions are approximately the same.

In the case of artificial channels that are free from the irregularities found in natural streams, it is possible to apply analytical methods to the various problems of nonuniform flow. In many instances, however, particularly when dealing with natural channels, we must resort to trial solutions and graphical methods.[9]

In the case of pressure conduits, we have dealt with uniform and nonuniform flow without drawing much distinction between them. We can do this because in a closed pipe the area of the water section, and hence the mean velocity, is fixed at every point. But in an open channel these conditions are not fixed, and the stream adjusts itself to the size of cross section that the energy gradient (i.e., slope of the energy line) requires.

In an open stream on a falling grade, without friction, the effect of gravity is to tend to produce a flow with a continually increasing velocity along the path, as in the case of a freely falling body. However, the gravity force is opposed by frictional resistance. The frictional force increases with velocity, while gravity is constant; so eventually the two will be in balance, and uniform flow will occur. When the two forces are not in balance, the flow is nonuniform.

There are two types of nonuniform flow. In one the changing conditions extend over a long distance, and we call this ***gradually varied flow.*** In the other the change may take place very abruptly, and the transition is thus confined to a short distance. We designate this as a ***local nonuniform phenomenon*** or ***rapidly varied flow.*** Gradually varied flow can occur with either subcritical or supercritical flow, but the transition from one condition to the other is ordinarily abrupt, as between D and E in Fig. 10.1. Other cases of local nonuniform flow occur at the entrance and exit of a channel, at changes in cross section, at bends, and at obstructions such as dams, weirs, or bridge piers.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

[9] For details on various methods of solving gradually varied open-channel flow, see Richard H. French, *Open-Channel Hydraulics,* McGraw-Hill Book Company, New York, pp. 201–247, 1985.

10.15 ENERGY EQUATION FOR GRADUALLY VARIED FLOW

The principal forces involved in flow in an open channel are inertia, gravity, net hydrostatic force due to change in depth, and friction. The first three represent the useful kinetic and potential energies of the liquid, while the fourth dissipates useful energy into the useless kinetic energy of turbulence and eventually into heat because of the action of viscosity. Referring to Fig. 10.19, the total energy of the elementary volume of liquid shown is proportional to

$$H = z + y + \alpha\frac{V^2}{2g} \tag{10.35}$$

where $z + y$ is the potential energy head above the arbitrary datum, and $\alpha V^2/2g$ is the kinetic energy head, V being the mean velocity in the section. Each term of the equation represents energy in foot-pounds per pound of fluid (or newton-meters per newton of fluid in SI units). Once again, as in Secs. 10.1 and 10.2, we define L as the distance along the channel bed between any two sections and Δx as the horizontal distance between the sections. For all practical purposes, these two distances can be considered as being equal to one another.

We will generally find the value of α to be higher in open channels than in pipes, as was explained in Sec. 10.4. It may range from 1.05 to 1.40, and in the case of a channel with an obstruction the value of α just upstream may be as high as 2.00 or even more. As the value of α is not known unless the velocity distribution is determined, it is often omitted from the equations, but an effort should

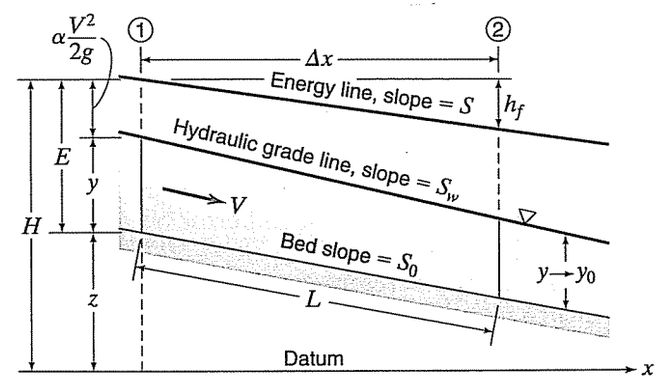


Figure 10.19
Energy relations for nonuniform flow.

be made to employ it if a high degree of accuracy is necessary. In the numerical problems in this chapter, α is assumed to be unity.

The energy equation for steady flow between two sections (1) and (2) of Fig. 10.19 a distance Δx apart is

$$z_1 + y_1 + \alpha_1 \frac{V_1^2}{2g} = z_2 + y_2 + \alpha_2 \frac{V_2^2}{2g} + h_f \tag{10.36}$$

As $z_1 - z_2 = S_0 \Delta x$ and $h_f \approx S \Delta x$, the energy equation may also be written (with $\alpha_1 = \alpha_2 = 1$) in the form

$$y_1 + \frac{V_1^2}{2g} = y_2 + \frac{V_2^2}{2g} + (S - S_0)\Delta x \tag{10.37}$$

We can make an approximate analysis of gradually varied, nonuniform flow by considering a length of stream to consist of a number of successive reaches, in each of which uniform flow occurs. Greater accuracy results from smaller depth variations in each reach. The Manning formula (Eq. 10.7) is rearranged and applied to average conditions in each reach to provide an estimate of the value of the energy gradient S for that reach as follows:

In BG units:
$$S = \left(\frac{n\overline{V}}{1.486 \overline{R}_h^{2/3}} \right)^2 \tag{10.38a}$$

In SI units:
$$S = \left(\frac{n\overline{V}}{\overline{R}_h^{2/3}} \right)^2 \tag{10.38b}$$

where $\overline{V}$ and $\overline{R}_h$ are the means of the respective values at the two ends of the reach. With this value of S, with S_0 and n known, and with the depth and velocity at both ends of the reach known, we can compute the length Δx from Eq. (10.37), rearranged as follows:

$$\Delta x = \frac{(y_1 + V_1^2/2g) - (y_2 + V_2^2/2g)}{S - S_0} = \frac{E_1 - E_2}{S - S_0} \tag{10.39}$$

Only the depth y must be prescribed at each end; then the values of V and R_h follow from the discharge and cross-section shape. As the calculation for Δx is direct, this approach is much easier than prescribing Δx and trying to solve for y_2, which would require trial and error because V_2, R_{h2}, and therefore S are all functions of y_2.

In practice, therefore, we divide the depth range of interest into small increments, usually equal, which define reaches whose lengths can be found by using Eq. (10.39). We must begin the discretization process from a section where y is known, i.e., from calculations for a control section (Sec. 10.12) or from a field measurement at any section. If the resulting value of Δx is negative, from Eq. (10.39) and Fig. 10.19 we can deduce this means that section (2) is *upstream* of section (1).

SAMPLE PROBLEM 10.9 At a certain section in a very smooth rectangular channel 6 ft wide the depth is 3.00 ft when the flow rate is 160 cfs. Compute the distance to the section where the depth is 3.20 ft if $S_0 = 0.0020$ and $n = 0.012$.

Solution

The calculations are shown in the following table, where $\overline{V} = (V_1 + V_2)/2$ and $\overline{R}_h = (R_{h1} + R_{h2})/2$. The total distance is calculated to be 71.3 ft.

y, ft	$A = 6y$, ft^2	$P = 6 + 2y$, ft	$R_h = A/P$, ft	$V = Q/A$ fps	$V^2/2g$, ft	$E = y + \frac{V^2}{2g}$, ft	Numerator $E_1 - E_2$, ft	$\overline{V}$, fps	$\overline{R}_h$, ft	S Eq. (10.38)	Denominator $S - S_0$	$\Delta x = \frac{E_1 - E_2}{S - S_0}$, ft
3.00	18.00	12.00	1.500	8.889	1.2269	4.2269						
							−0.0221	8.746	1.512	0.002 873	0.000 873	−25.3
3.10	18.60	12.20	1.525	8.602	1.1490	4.2490						
							−0.0293	8.468	1.536	0.002 637	0.000 637	−46.0
3.20	19.20	12.40	1.548	8.333	1.0783	4.2783						

ANS: $\Sigma(\Delta x) = -71.3$

Notes:

1. The accuracy of the calculated total distance (71.3 ft) could be improved by taking more steps, i.e., smaller Δy.
2. As the resulting Δx are negative, the sections progress *upstream,* and the water depth decreases downstream.
3. The *magnitude* (absolute value) of Δx increases with depth (sketch the profile by reaches to see this), so the surface curvature is concave down.
4. $y_c = 2.81$ ft (from Eq. 10.23) and $y_0 = 3.50$ ft (from Eq. 10.8*a*), so $y_c < y < y_0$. In Sec. 10.16 we shall see that this tells us the water surface must have the shape of an M_2 profile, with depth decreasing downstream and curved concave down.

An important issue in calculations of this type is working with sufficient precision, i.e., carrying enough significant figures. In two instances we must deal with the small difference between two numbers whose values are close to one another. In the example just presented, we see that we lost two significant figures in calculating $E_1 - E_2$ and we lost one in calculating $S - S_0$. Therefore the calculations for these quantities, and for V and R_h on which they are based, need to be carried to more significant figures so that the final results are always reliable to three significant figures.

If the data for the preceding example had been given in SI rather than BG units, the procedure for solution would have been the same except that

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Eqs. (10.38*b*) and (10.7*a*) would have been used rather than Eqs. (10.38*a*) and (10.7*b*). Note that $\overline{V}$ is *not* calculated from $Q/\overline{A}$, nor from $\overline{y} = (y_1 + y_2)/2$; these would give different results. Because of the extensive calculations required for many reaches, we commonly solve this type of problem by computer, either by using applications software (Appendix C.5), or writing a program (Appendix C.6), or using a spreadsheet (Appendix C.2).

Section 10.16 will explain that depth ranges for this procedure must not cross the normal or critical depths. Therefore, to check this, first compute y_0 from Eq. (10.8) and y_c from Eq. (10.23) for rectangular channels or from Eq. (10.33) for other channels.

EXERCISES

10.15.1 A rectangular flume of planed timber ($n = 0.012$) is 5 ft wide and carries 60 cfs of water. The bed slope is 0.0006, and at a certain section the depth is 3 ft. Find the distance (in one reach) to the section where the depth is 2.5 ft. Is the distance upstream or downstream?

10.15.2 A rectangular flume of timber ($n = 0.013$) is 1.6 m wide and carries 1.7 m³/s of water. The bed slope is 0.0005, and at a certain section the depth is 1.0 m. Find the distance (in one reach) to the section where the depth is 0.85 m. Is the distance upstream or downstream?

10.15.3 A test on a rectangular glass flume 9 in wide yielded the following data on a reach of 40-ft length: with still water, $z_1 - z_2 = 0.009$ ft; with a measured flow of 0.1446 cfs, $y_1 = 0.381$ ft, $y_2 = 0.386$ ft. Find the value of Manning's roughness coefficient n using only one reach.

10.15.4 A test on a rectangular glass flume 250 mm wide yielded the following data on a reach of 9-m length: with still water, $z_1 - z_2 = 2.7$ mm; with a measured flow of 4.3 L/s, $y_1 = 110.2$ mm, $y_2 = 111.7$ mm. Find the value of Manning's roughness coefficient n using only one reach.

10.16 WATER-SURFACE PROFILES IN GRADUALLY VARIED FLOW (RECTANGULAR CHANNELS)

As there are some 12 different circumstances giving rise to as many different fundamental types of gradually varied flow, having a logical scheme of type classification is helpful. In general, any problem of varied flow, no matter how complex it may appear, with the stream passing over dams, under sluice gates, down steep chutes, on the level, or even on an upgrade, can be broken down into reaches such that the flow within any reach is either uniform or falls within one of the given nonuniform classifications. We then analyze the stream one reach at a time, proceeding from one reach to the next until the desired result is obtained.

The following treatment is based, for simplicity, on *channels of rectangular section.* The section will be considered sufficiently wide and shallow so that we may confine our attention to a section 1 ft (or 1 m) wide through which the velocity

is essentially uniform. It is important to bear in mind that this development is based on a constant value of the discharge per unit width q, which can only be used with rectangular channels, and on one value of n, the roughness coefficient.

Differentiating Eq. (10.35) with respect to x, the horizontal distance along the channel, the rate of energy dissipation is found (with $\alpha = 1$) to be

$$\frac{dH}{dx} = \frac{dz}{dx} + \frac{dy}{dx} + \frac{1}{2g}\frac{d(V^2)}{dx} \tag{10.40}$$

The energy gradient $S = -dH/dL \approx -dH/dx$, while the slope of the channel bed is $S_0 = -dz/dx$, and the slope of the hydraulic grade line or water surface is given by $S_w = -dz/dx - dy/dx$.

Commencing with the last term of Eq. (10.40), we may observe that since $V = q/y$,

$$\frac{1}{2g}\frac{d(V^2)}{dx} = \frac{1}{2g}\frac{d}{dx}\left(\frac{q^2}{y^2}\right) = -\frac{q^2}{g}\frac{1}{y^3}\frac{dy}{dx}$$

Substituting this, plus the S and S_0 terms, into Eq. (10.40), yields

$$-S = -S_0 + \frac{dy}{dx}\left(1 - \frac{q^2}{gy^3}\right)$$

or

$$\frac{dy}{dx} = \frac{S_0 - S}{1 - q^2/gy^3} = \frac{S_0 - S}{1 - V^2/gy} = \frac{S_0 - S}{1 - \mathbb{F}^2} \tag{10.41}$$

Evidently, if the value of dy/dx as determined by Eq. (10.41) is positive, the water depth will be increasing along the channel; if negative, it will be decreasing. Let us examine the numerator and denominator of Eq. (10.41).

Looking first at the numerator, we can consider S as the energy gradient [such as would be obtained from Eq. (10.38)] that would carry the given discharge at depth y with uniform flow. For a wide and shallow rectangular channel, substituting $V = q/y$ and $R_h = y$ into Eq. (10.38*a*), we obtain

Wide and shallow: $$S = \left(\frac{nq}{1.486y^{5/3}}\right)^2$$

Similarly substituting these into Eq. (10.7*b*), we obtain

Wide and shallow: $$S_0 = \left(\frac{nq}{1.486y_0^{5/3}}\right)^2$$

Comparing these two expressions, we see that $S/S_0 = (y_0/y)^{10/3}$. Consequently, for constant q and n, when $y > y_0$, $S < S_0$, and the numerator $(S_0 - S)$ is positive. Conversely, when $y < y_0$, $S > S_0$, and the numerator is negative.[10]

[10] Combining these deductions with the fact that E must decrease downstream if $S > S_0$, and vice versa, yields the result that E must decrease downstream when $y < y_0$ and vice versa.

To investigate the denominator of Eq. (10.41), we observe that if $\mathbf{F} = 1$ (critical flow, Sec. 10.10, $y = y_c$), the denominator is zero and $dy/dx = \infty$; if $\mathbf{F} > 1$ (supercritical flow, $y < y_c$), the denominator is negative; and if $\mathbf{F} < 1$ (subcritical flow, $y > y_c$), the denominator is positive.

In this manner the signs of the numerator and the denominator of Eq. (10.41) can be found for any depth by comparing it with y_0 and y_c. These two signs together give the sign of dy/dx, which in turn defines the slope of the water surface.

The foregoing analyses have been combined graphically into a series of water-surface profiles in Fig. 10.20. The surface profiles are classified according to slope and depth as follows. If S_0 is positive, the bed slope is termed ***mild*** (M) when $y_0 > y_c$, ***critical*** (C) when $y_0 = y_c$, and ***steep*** (S) when $y_0 < y_c$; if $S_0 = 0$, the channel is ***horizontal*** (H); and if S_0 is negative, the bed slope is called ***adverse*** (A). If the stream surface lies above both the normal (uniform flow) and critical depth lines, it is of type 1; if between these lines, it is of type 2; and if below both lines, it is of type 3. The 12 forms of surface curvature are labeled accordingly in Fig. 10.20. Each is discussed in more detail in the next section.

Note that the scale of the drawings in Fig. 10.20 is greatly compressed in the horizontal direction. The problems at the end of this chapter demonstrate that gradually varied flow generally extends over many hundreds of feet, and if plotted to an undistorted scale, the rate of change in depth would be scarcely discernible. Note also that, since even a hydraulically steep slope varies only a few degrees from the horizontal, it makes little difference whether the depth y is measured vertically (as shown) or perpendicular to the bed.

We see that some of the curves of Fig. 10.20 are concave upward while others are concave downward. Although the mathematical proof for this is not given, the physical explanation is not hard to find. In the case of the type 1 curves, the surface must approach a horizontal asymptote as the velocity is progressively slowed down because of the increasing depth. Likewise all curves that approach the normal or uniform depth line must approach it asymptotically, because uniform flow will only prevail at sections remote from disturbances, as pointed out in Sec. 10.1. Theoretically the curves that cross the critical-depth line must do so vertically, as the denominator of Eq. (10.41) becomes zero in this case. So these end conditions govern the surface concavity. The critical-slope curves, for which $y_0 = y_c$, constitute exceptions, since it is not possible for a curve to be both tangent and perpendicular to the same critical-uniform depth line.

Profiles that get deeper downstream we call ***backwater curves,*** and those that decrease in depth downstream we call ***drawdown curves.***

To the right of each water-surface profile is shown a representative example of how this particular curve can occur. Many of the examples show a rapid change from a depth below the critical to a depth above the critical. This is a local phenomenon, known as the hydraulic jump, which is discussed in detail in Sec. 10.18.

The qualitative analysis of water-surface profiles has been restricted to rectangular sections. The curve forms of Fig. 10.20 are, however, applicable to any

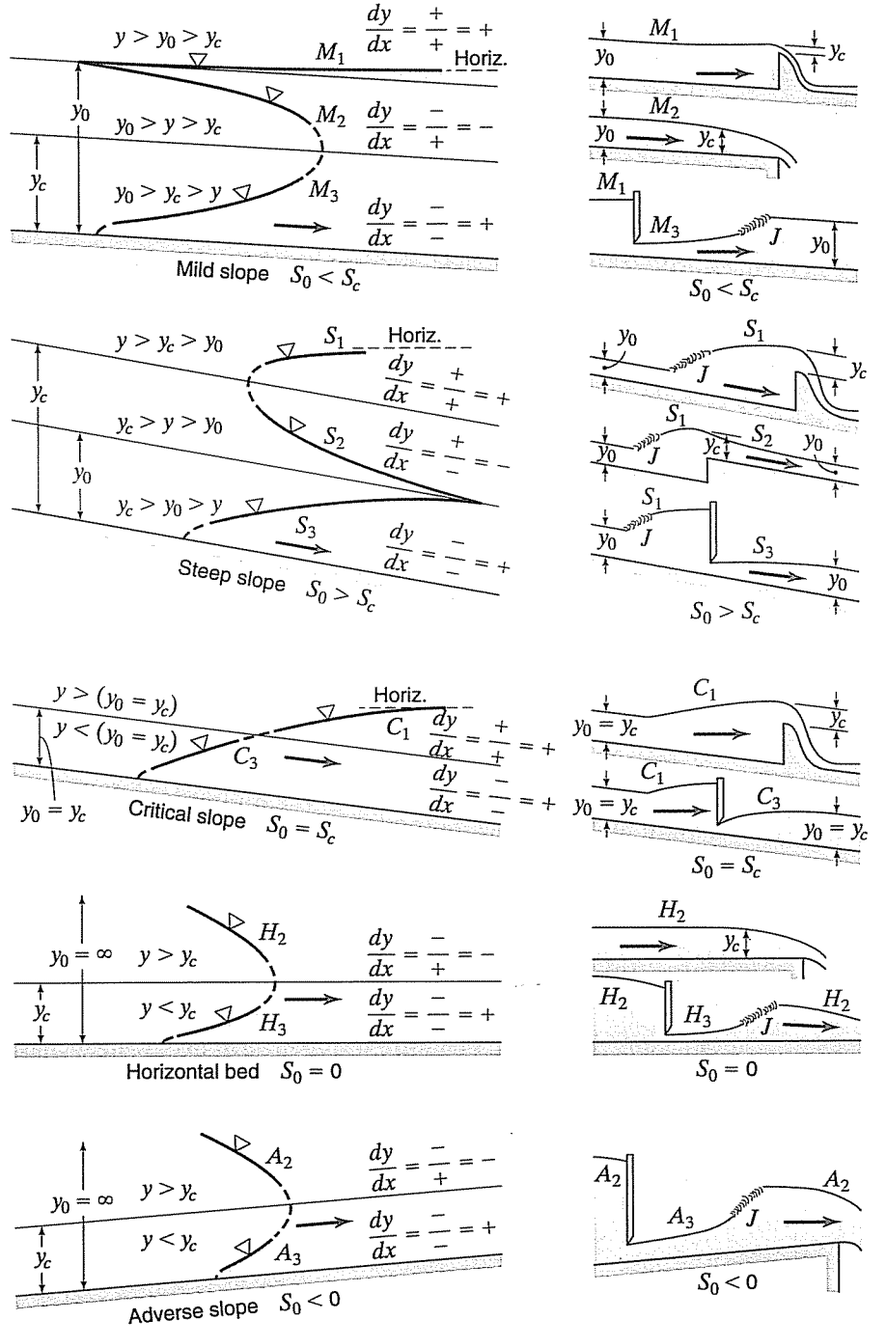


Figure 10.20
Various types of nonuniform flow, with flow from left to right.

channel of uniform cross section if y_0 is the depth for uniform flow and y_c is the depth that satisfies Eq. (10.33). The surface profiles can even be used qualitatively in the analysis of natural stream surfaces as well, provided that local variations in slope, shape, and roughness of cross section, etc., are taken into account. The step-by-step method presented in Sec. 10.15 for the solution of steady nonuniform-flow problems is not restricted to uniform channels, and is therefore suited to water-surface-profile computations for any stream whatever.

A very important precaution, when using the stepwise procedure to solve for water-surface profiles, is to *always first determine the critical and normal depths,* y_c and y_0. Then we can easily confirm that the profile sought does not cross either of these depths. For if the ends of a reach are on different sides of either of these depths, the results are always invalid, even though sometimes they may look numerically reasonable. Along similar lines, if part of a reach is at uniform flow, this can also lead to impossible results when solving for depth. We can avoid such a situation by first calculating the gradually varied profile distance to y_0.

10.17 EXAMPLES OF WATER-SURFACE PROFILES

The M_1 Curve

The most common case of gradually varied flow is where the depth is already above the critical and is increased still further by a dam, as indicated in Fig. 10.21. Referring to the specific-energy diagram of Fig. 10.12, this case is found on the upper limb of the diagram, for here also, as the depth increases, the velocity diminishes without any abrupt transitions, so that a smooth surface curve is obtained. Note that it is possible for the water surface *elevation* to rise downstream, in addition to the depth. In the case of flow in an artificial channel with a constant bed slope, the water-surface curve would be asymptotic at infinity to the surface for uniform flow, as we noted before. But the problems that are usually of more important interest are those concerned with the effect of a dam on a natural stream and the extent to which it raises the water surface at various points upstream. The resulting water-surface profile in such a case is commonly known as a ***backwater curve.***

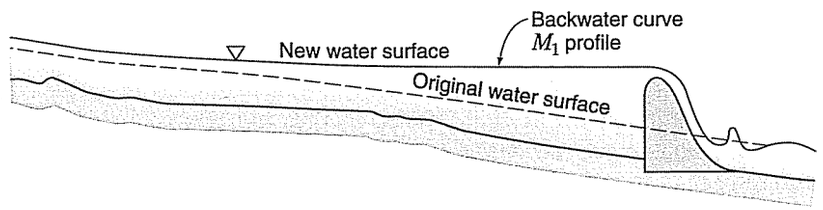


Figure 10.21
Backwater curve in a natural stream.

For an artificial channel where the conditions are uniform, save for the variation in water depth, we can solve the problem using Eqs. (10.38) and (10.39). Usually, the solution commences at the dam, where conditions are assumed to be known, and the lengths of successive reaches upstream, corresponding to assumed increments of depth, are computed. A tabular type of solution (Sample Prob. 10.9) is the most helpful, with column headings corresponding to the various elements of Eqs. (10.38) and (10.39), the last column being $\Sigma \Delta x$, which sums up the length from the dam to the point in question. It is important, if accuracy is desired, to keep the depth increment small within any reach; a depth change of 10% or less is fairly satisfactory. The smaller the depth increment used in this step-by-step procedure, the greater the accuracy of the final result. We can conveniently solve this type of problem that requires successive calculations with a computer.

For a natural stream, such as that shown in Fig. 11.11, the solution is not so direct, because the form and dimensions of a cross section cannot be assumed and then the distance to its location computed. As there are various slopes and cross sections at different distances upstream, we must assume the value of Δx in Eq. (10.37), and then we can compute the depth of stream at this section by trials, as in Prob. 10.45. The solution is then pursued in similar fashion on a reach-by-reach basis. The accuracy of the results depends on the selection of a proper value for Manning's n, which is difficult when dealing with natural streams. For this reason and because of irregularities in the flow cross sections, the refinements of Eq. (10.37) are not always justified, and it is often satisfactory to assume uniform flow by applying Eq. (10.8) to each successive reach.

The M_2 Curve

This curve, representing accelerated subcritical flow on a slope that is flatter than critical, exists, like the M_1 curve, because of a control condition *downstream.* In this case, however, the control is not an obstruction but the removal of the hydrostatic resistance of the water downstream, as in the case of the free overfall shown in Fig. 10.17*a*. As in the M_1 curve, the surface will approach the depth for uniform flow at an infinite distance upstream. Practically, because of slight wave action and other irregularities, the distinction between the M_2, or ***drawdown curve,*** and the curve for uniform flow disappears within a finite distance.

The M_3 Curve

This occurs because of an upstream control, like the sluice gate shown in Fig. 10.20. The bed slope is not sufficient to sustain lower-stage flow, and at a certain point determined by energy and momentum relations, the surface will pass through a hydraulic jump unless this is made unnecessary by the existence of a free overfall before the M_3 curve reaches critical depth.

The *S* Curves

We can analyze these in much the same fashion as the *M* curves, having due regard for downstream control in the case of subcritical flow and for upstream control for supercritical flow. Thus a dam or an obstruction on a steep slope produces an S_1 curve upstream (Fig. 10.20), which approaches the horizontal asymptotically but cannot so approach the uniform depth line, which lies below the critical depth. Therefore the S_1 curve must be preceded by a hydraulic jump. The S_2 curve shows accelerated lower-stage flow, smoothly approaching uniform depth. Such a curve will occur whenever a steep channel receives flow at critical depth, as from an obstruction (as shown) or reservoir. The sluice gate on a steep channel will produce the S_3 curve, which also approaches smoothly the uniform depth line.

The *C* Curves

These curves, with the anomalous condition ($dy/dx = \infty$) at $y_0 = y_c$, do not occur frequently. As noted in Sec. 10.9, we should avoid creating conditions that produce them because of the inherent instability of such flows.

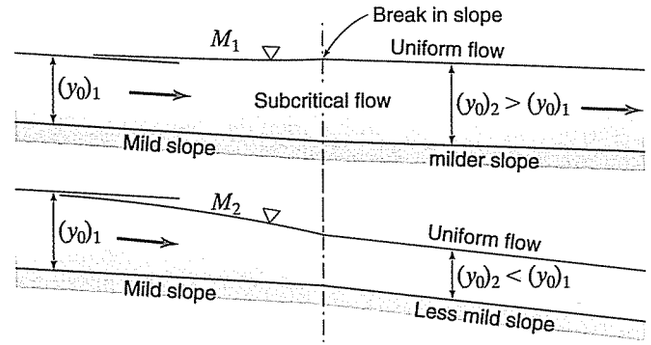
The *H* and the *A* Curves

These curves have in common the fact that there is no condition of uniform flow possible. The H_2 and A_2 drawdown curves are similar to the M_2 curve, but even more noticeable. The value of $y_b = 0.72y_c$ given in Sec. 10.12 applies strictly only to the H_2 curve, but it is approximately true for the M_2 curve also. The sluice gate on the horizontal and adverse slopes produces H_3 and A_3 curves that are like the M_3 curve, but they do not exist for as great a distance as the M_3 curve before a hydraulic jump occurs. Of course, it is not possible to have a channel of any appreciable length carry water on a horizontal grade, and even less so on an adverse grade.

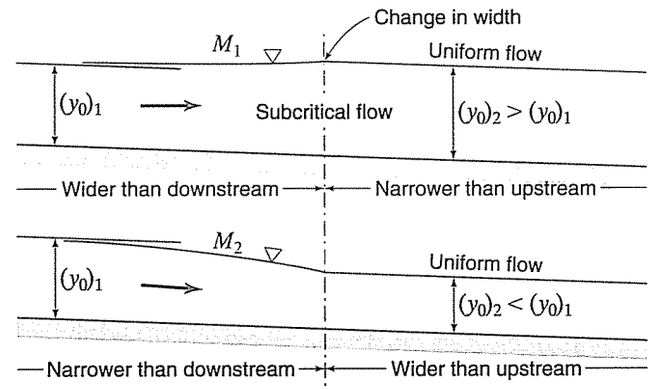
Other Examples

Some other interesting water-surface profiles occur when the slope of a channel of uniform section changes abruptly from a mild to a milder slope or to a less mild slope. In this case the flow is everywhere subcritical. Similar water-surface profiles occur when a channel on a constant slope that is mild throughout its entire length has an abrupt change in width to an either narrower or wider channel. These possibilities are depicted in Fig. 10.22.

Other water-surface profiles include those that occur when the slope of a channel changes abruptly from steep to either steeper or less steep. In this case the flow is supercritical. Similar profiles occur when a channel on a constant slope that is steep throughout its entire length has an abrupt change in width to an either wider or narrower channel. As an exercise it is suggested that the reader sketch profiles similar to Fig. 10.22 for the steep-slope situations. In these cases it will be found that, with steep slopes (supercritical flow), uniform flow



(*a*) Constant section with change in slope



(*b*) Constant mild slope with change in width

Figure 10.22
Subcritical flow water-surface profiles.

occurs upstream of the change in either slope or width, while with mild slopes (subcritical flow), uniform flow prevails downstream of the change.

EXERCISES

10.17.1 Under what different conditions can supercritical flow leaving a hump ($\Delta z > \Delta z_{crit}$) return to normal depth without passing through a hydraulic jump? Explain your answers.

10.17.2 Classify the water-surface profile of Exer. 10.15.1 as one of the forms shown in Fig. 10.20. Show all necessary calculations.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.17.3 Classify the water-surface profile of Exer. 10.15.2 as one of the forms shown in Fig. 10.20. Show all necessary calculations.

10.17.4 Repeat Exer. 10.17.2 when the channel slope is −0.00040.

10.17.5 Repeat Exer. 10.17.3 when the channel slope is −0.00040.

10.17.6 The flow in a 15-ft-wide rectangular channel that has a constant bottom slope is 1400 cfs. A computation using Manning's equation indicates that the normal depth is 6.0 ft. At a certain section the depth of flow in the channel is 2.8 ft. Does the depth increase, decrease, or remain the same as one proceeds downstream from this section? Sketch a physical situation where this type of flow will occur.

10.17.7 A laboratory flume ($n = 0.012$) is 10 in wide and set on a slope of 0.0003. With a measured flow of 0.1516 cfs, the depth is observed to vary between 0.361 and 0.366 ft. Classify the water-surface profile as one of the forms of Fig. 10.20. Show all necessary calculations, and sketch the profile.

10.17.8 A laboratory flume ($n = 0.012$) is 250 mm wide and set on a slope of 0.0003. With a measured flow of 4.3 L/s, the depth is observed to vary between 110.2 and 111.7 mm. Classify the water-surface profile as one of the forms of Fig. 10.20. Show all necessary calculations, and sketch the profile.

10.17.9 In an 8-ft-wide rectangular channel ($S_0 = 0.003, n = 0.015$) water flows at 300 cfs. A low dam (broad-crested weir) placed across the channel raises the water to a depth of 8.0 ft. Analyze the water-surface profile upstream from the dam.

10.18 THE HYDRAULIC JUMP

By far the most important of the local nonuniform-flow phenomena is that which occurs when supercritical flow has its velocity reduced to subcritical. We have seen in the surface profiles of Fig. 10.20 that there is no ordinary means of changing from supercritical to subcritical flow with a smooth transition, because theory calls for a vertical slope of the water surface. The result, then, is a marked discontinuity in the surface, characterized by a steep upward slope of the profile, broken throughout with violent turbulence, and known universally as the ***hydraulic jump.***

The specific reason for the occurrence of the hydraulic jump can perhaps best be explained by referring to the M_3 curve of Fig. 10.20. Downstream of the sluice gate the flow decelerates because the slope is not great enough to maintain supercritical flow. The specific energy decreases as the depth increases (proceeding to the left along the lower limb of the specific-energy diagram, Fig. 10.12). Were this condition to progress until the flow reached critical depth, the specific energy would need to increase as the depth increased from the critical to the uniform flow depth downstream. But this is a physical impossibility. Therefore the jump forms before the necessary energy is lost.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

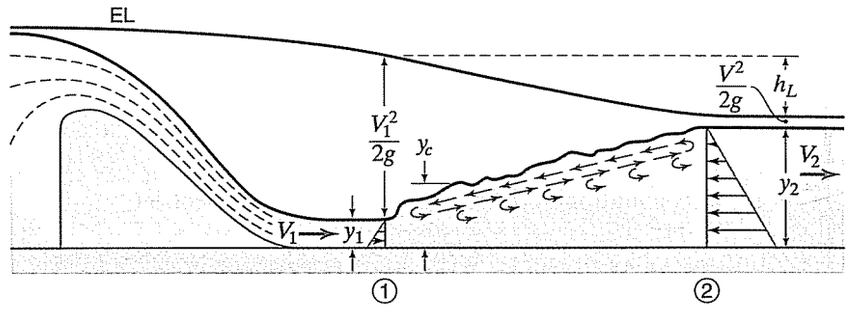


Figure 10.23
Hydraulic jump on horizontal bed following a spillway; horizontal scale foreshortened between sections 1 and 2 approximately $2\frac{1}{2}$:1.

The hydraulic jump can also occur from an upstream condition of uniform supercritical flow to a nonuniform S_1 curve downstream when there is an obstruction on a steep slope, as illustrated in Fig. 10.20, or again from a nonuniform upstream condition to a nonuniform downstream condition, as illustrated by the H_3–H_2 or the A_3–A_2 combinations. An M_3–M_2 combination is also possible. In addition to the foregoing cases, where the channel bed continues at a uniform slope, a jump will form when the slope changes from steep to mild, as on the apron at the base of the spillway, illustrated in Fig. 10.23. This is an excellent example of the jump serving a useful purpose, for it dissipates much of the destructive energy of the high-velocity water, thereby reducing downstream erosion. The turbulence within hydraulic jumps has also been found to be very useful and effective for mixing fluids, and jumps have been used for this purpose in water treatment plants and sewage treatment plants.

Depth Relations—General

We will derive the equation relating the depths before and after a hydraulic jump for the case of a horizontal channel bottom (the H_3–H_2 combination of Fig. 10.20). For channels on a gradual slope (i.e., less than about 3° or $S_0 = 0.05$) the gravity component of the weight is relatively small and can be neglected without introducing significant error. The friction forces acting are negligible because of the short length of channel involved and therefore the only significant forces are hydrostatic forces. Applying momentum principle Eq. (6.7*a*) to the volume of fluid between sections 1 and 2 of Fig. 10.23, and using Eq. (3.16) for the end forces, we obtain for any shape of channel cross section

$$\sum F_x = \gamma h_{c_1} A_1 - \gamma h_{c_2} A_2 = \frac{\gamma}{g} Q(V_2 - V_1)$$

where h_c is the depth to the centroid of the end area. We can rearrange this equation to give

$$\frac{\gamma}{g} Q V_1 + \gamma h_{c_1} A_1 = \frac{\gamma}{g} Q V_2 + \gamma h_{c_2} A_2 \qquad (10.42)$$

This states that the momentum plus the pressure force on the cross-sectional area is constant, or dividing by γ and observing that $V = Q/A$,

$$F_m = \frac{Q^2}{Ag} + Ah_c = \text{constant} \tag{10.43}$$

This equation applies to any shape of cross section.

Depth Relations—Rectangular Channel

In the case of a rectangular channel, we have $h_c = y/2$, $A = by$, and we can again use the flow q per unit width, so that $Q = bq$. Substituting for these into Eq. (10.43), it reduces for a unit width to

$$f_m = \frac{q^2}{gy_1} + \frac{y_1^2}{2} = \frac{q^2}{gy_2} + \frac{y_2^2}{2} = \text{constant} \tag{10.44}$$

A curve of values of f_m for different values of y is plotted to the right of the specific-energy diagram shown in Fig. 10.24. Both curves are plotted for the condition of 2 cfs/ft of width. As the loss of energy in the jump does not affect the "force" quantity f_m, the latter is the same after the jump as before, and therefore any vertical line intersecting the f_m curve serves to locate two ***conjugate depths,*** y_1 and y_2. These depths represent possible combinations of depth that could occur before and after the jump.

Thus, in Fig. 10.24, the line for the initial water level y_1 intersects the f_m curve at a as shown, giving the value of f_m, which must be the same after the jump. The vertical line ab then fixes the value of y_2. This depth is then transposed to the specific-energy diagram to determine the value cd of $V_2^2/2g$. The value of $V_1^2/2g$ is the vertical distance ef.

We can show that for a given flow rate in a rectangular channel the minimum value of f_m (Eq. 10.44) occurs at the same depth as the minimum value of E.

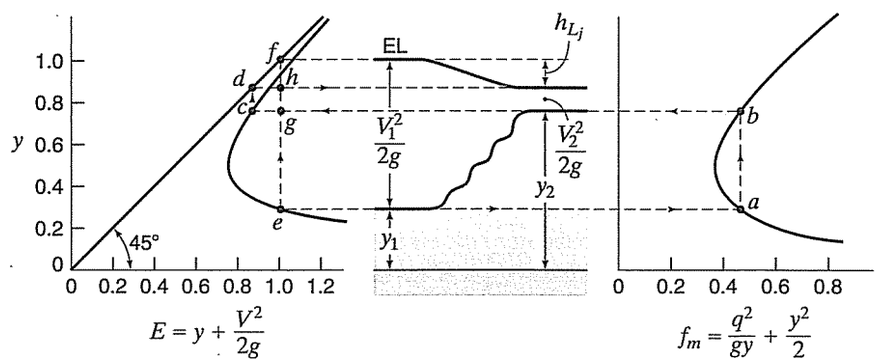


Figure 10.24
Energy and momentum relations in a hydraulic jump.

Differentiating f_m with respect to y and equating to zero gives

$$f_m = \frac{q^2}{gy} + \frac{y^2}{2}$$

$$\frac{df_m}{dy} = -\frac{q^2}{gy^2} + \frac{2y}{2} = 0$$

and

$$y = \left(\frac{q^2}{g}\right)^{1/3}$$

This expression is identical to Eq. (10.23). Thus we have shown that for a given q the minimum value of f_m occurs at the same depth as does the minimum value of E. We can see this in Fig. 10.24.

When the rate of flow and the depth before or after the jump are given, we see that Eq. (10.44) becomes a cubic equation when solving for the other depth. We can easily reduce this to a quadratic, however, by observing that $y_2^2 - y_1^2 = (y_2 + y_1)(y_2 - y_1)$, so that

$$\frac{q^2}{g} = y_1 y_2 \frac{y_1 + y_2}{2} \tag{10.45}$$

From Eq. (10.23) we also know that $q^2/g = y_c^3$, so if $y_1 < y_c$ then we must have $y_2 > y_c$. This proves that the hydraulic jump must always cross the critical depth.

Solving Eq. (10.45) by the quadratic formula gives

$$y_2 = \frac{y_1}{2}\left(-1 + \sqrt{1 + \frac{8q^2}{gy_1^3}}\right) \tag{10.46a}$$

or

$$y_1 = \frac{y_2}{2}\left(-1 + \sqrt{1 + \frac{8q^2}{gy_2^3}}\right) \tag{10.46b}$$

These equations relate the depths before and after a hydraulic jump (i.e., the conjugate depths) in a rectangular channel. They give good results if the channel slope is less than about 0.05 or 3°. For steeper channel slopes the effect of the gravity component of the weight of liquid between sections 1 and 2 of Fig. 10.23 must be considered.

Energy Loss

The head loss h_{L_j} caused by the jump is the drop in energy from 1 to 2. Or

Any channel:
$$h_{L_j} = E_1 - E_2 = \left(y_1 + \frac{V_1^2}{2g}\right) - \left(y_2 + \frac{V_2^2}{2g}\right) \tag{10.47}$$

On Fig. 10.24 points e and c on the specific-energy diagram represent the conjugate depths, and the horizontal distance between them ($= cg = fh$) is the head

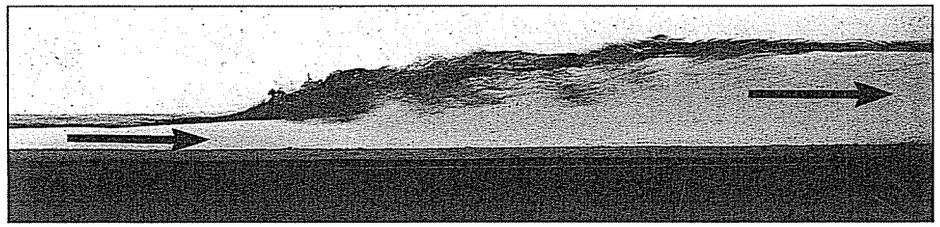


Figure 10.25
Hydraulic jump. (Courtesy of the Archives, California Institute of Technology)

loss. Replacing V by q/y and using Eq. (10.45), it can be shown that also

Rectangular channel:
$$h_{L_j} = \frac{(y_2 - y_1)^3}{4y_1y_2} \tag{10.48}$$

Examples of depth and energy loss calculations are included in Sample Prob. 10.10.

Jump Length

Although the length of a jump is difficult to predict, a good approximation for jump length is about $5y_2$. We can see this relation is approximately true by examining Fig. 10.25, a photograph of a hydraulic jump in a horizontal channel, caused by a sluice gate upstream. In most cases $4 < L_j/y_2 < 6$.

Types of Jump

Since the flow must be supercritical in order for a jump to occur, $\mathbf{F}_1$, the Froude number of the flow just upstream of the jump, must be greater than 1.0. As $\mathbf{F}_1$ increases, the jump becomes more turbulent and more energy is dissipated. Ranges of different jump behaviors have been identified, some more desirable than others. These are summarized in Table 10.4. In particular, the oscillating jump is to be avoided; the main stream of water flowing through the jump oscillates between the bed and the surface, generating waves that can be damaging for great distances downstream.

TABLE 10.4 **Types of hydraulic jump**[a]

Name	$\mathbf{F}_1$	Energy dissipation	Characteristics
Undular jump	1.0–1.7	<5%	Standing waves
Weak jump	1.7–2.5	5–15%	Smooth rise
Oscillating jump	2.5–4.5	15–45%	Unstable; avoid
Steady jump	4.5–9.0	45–70%	Best design range
Strong jump	>9.0	70–85%	Choppy, intermittent

[a] U.S. Bureau of Reclamation, Research Studies on Stilling Basins, Energy Dissipators, and Associated Appurtenances, *Hydraul. Lab. Rept.* Hyd-399, 1955.

In some areas of the world with high tides, when they enter estuaries and the like, a small hydraulic jump forms, which travels upstream and can cause minor damage. This phenomenon is known as a ***surge*** or a ***tidal bore.*** It may be analyzed by similar methods to those used for gravity waves (Sec. 10.20).

Stilling Basins

When water from elevated water bodies like reservoirs flows over spillways, through outlet works, or down chutes, it can have tremendous energy. This energy can cause great damage to the structure itself, to the ground supporting the structure (and therefore ultimately to the structure), and to the downstream channels. The hydraulic jump is one of the most effective methods of dissipating such flow energy, as we noted earlier. But hydraulic jumps themselves can also be very destructive. The objective of a ***stilling basin*** is to initiate a jump for energy dissipation and to contain it within a structure that will minimize damage.

The basin floor may be recessed or have an adverse slope to provide sufficient tailwater depth; a ***sill*** across the downstream end is also commonly used for this purpose. To increase the turbulence and to encourage the jump to form immediately as the water enters the basin, rows of ***chute blocks*** and/or staggered ***baffle blocks*** are commonly arranged across the basin. Sometimes those blocks with an end sill can reduce the length of the jump, and so the basin.

Because high entry velocities can cause cavitation damage (Sec. 5.10) and erosion by sediment, the basin must be finished with quality concrete brought to a smooth surface.

EXERCISES

10.18.1 A hydraulic jump occurs in a 20-ft-wide rectangular channel carrying 250 cfs on a slope of 0.005. The depth after the jump is 4.0 ft. (*a*) What must be the depth before the jump? (*b*) What are the losses of energy and power in the jump?

10.18.2 A hydraulic jump occurs in a 5-m-wide rectangular channel carrying 8 m^3/s on a slope of 0.006. The depth after the jump is 1.75 m. (*a*) What must be the depth before the jump? (*b*) What are the losses of energy and power in the jump?

10.18.3 The hydraulic jump may be used as a crude flowmeter. Suppose that in a horizontal rectangular channel 6 ft wide the observed depths before and after a hydraulic jump are 0.80 and 3.60 ft, respectively. Find the rate of flow and the head loss.

10.19 LOCATION OF HYDRAULIC JUMP

The problem of determining the location of a hydraulic jump involves a combined application of the principles discussed in Secs. 10.17 and 10.18. Examples of the location of a hydraulic jump are shown in Figs. 10.20 and 10.26. In all these cases shown on mild and steep slopes one of the conjugate depths is a normal depth. In case (1) of Fig. 10.26 the jump occurs downstream of the break. The reasons for this are illustrated by the following example. For convenience here we shall designate y' to be the conjugate of y.

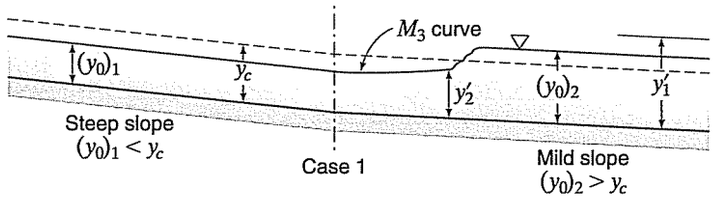


Figure 10.26
Examples of the location of a hydraulic jump where one conjugate depth is normal. [*Note*: y_1' is the conjugate depth of $(y_0)_1$ and y_2' is the conjugate depth of $(y_0)_2$.]

SAMPLE PROBLEM **10.10** Analyze the water-surface profile in a long rectangular channel lined with concrete ($n = 0.013$). The channel is 10 ft wide, the flow rate is 400 cfs, and the channel slope changes abruptly from 0.0150 to 0.0016. Find also the horsepower loss in the resulting jump.

Solution

Eq. (10.8*a*): $$400 = \frac{1.486}{0.013}(10y_{0_1})\left(\frac{10y_{0_1}}{10 + 2y_{0_1}}\right)^{2/3}(0.015)^{1/2}$$

We can solve this equation by any of the methods described in Sample Prob. 10.1. We obtain

$$y_{0_1} = 2.17 \text{ ft} \qquad \text{(normal depth on the upper slope)}$$

Using a similar procedure, we find the normal depth y_{0_2} on the lower slope to be 4.81 ft.

Eq. (10.23): $$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{\left(\frac{400}{10}\right)^2}{32.2}\right]^{1/3} = 3.68 \text{ ft}$$

Thus flow is supercritical ($y_{0_1} < y_c$) before the break in slope and subcritical ($y_{0_2} > y_c$) after the break, so a hydraulic jump must occur. One of the two profiles (case 1 or case 2) in Fig. 10.26 must occur.

Let us first explore case 2. Applying Eq. (10.46*a*) to determine the depth conjugate to the 2.17-ft normal depth on the upper slope, we get

$$y_1' = \frac{2.17}{2}\left\{-1 + \left[1 + \frac{8(40)^2}{32.2(2.17)^3}\right]^{1/2}\right\} = 5.77 \text{ ft}$$

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Therefore a jump on the upper slope must rise to 5.77 ft, after which the surface must rise still more along an S_1 curve (Fig. 10.20). When the flow enters the lower slope, the depth would therefore be greater than $y_{0_2} = 4.81$ ft and so in zone 1. Because an M_1 curve (Fig. 10.20) cannot bring the water surface down from above 5.77 ft to 4.81 ft (normal depth), such a profile and jump cannot occur.

Let us now explore case 1. Applying Eq. (10.46*b*) to determine the depth conjugate to the 4.81-ft normal depth on the lower slope, we get

$$y_2' = \frac{4.81}{2}\left\{-1 + \left[1 + \frac{8(40)^2}{32.2(4.81)^3}\right]^{1/2}\right\} = 2.74 \text{ ft}$$

This lower conjugate depth of 2.74 ft will occur downstream of the break in slope. The water surface on the lower slope *can* rise from 2.16 ft to $y_2' = 2.74$ ft, via an M_3 curve (Fig. 10.20). So this case will occur.

We can find the location of the jump (i.e., its distance below the break in slope) by applying Eq. (10.39) to the M_3 curve between depths 2.16 ft at the break and 2.74 ft at the start of the jump:

$$\Delta x = \frac{E_1 - E_2}{S - S_0}$$

$$E_1 = 2.17 + \frac{(40/2.17)^2}{2(32.2)} = 7.45 \text{ ft}$$

$$E_2 = 2.74 + \frac{(40/2.74)^2}{2(32.2)} = 6.05 \text{ ft}$$

$$\overline{V} = \frac{1}{2}\left(\frac{40}{2.17} + \frac{40}{2.74}\right) = 16.53 \text{ fps}$$

$$\overline{R}_h = \frac{1}{2}\left(\frac{21.7}{14.34} + \frac{27.4}{15.47}\right) = 1.641 \text{ ft}$$

From Eq. (10.38*a*): $$S = \left[\frac{(0.013)(16.53)}{1.486(1.641)^{2/3}}\right]^2 = 0.01081$$

Eq. (10.39): $$\Delta x = \frac{7.452 - 6.054}{0.01081 - 0.00160} = 151.8 \text{ ft}$$

Note that we could compute this distance more accurately by dividing it into more reaches, i.e., by using more, smaller depth increments.

Thus the depth on the upper slope is 2.17 ft; downstream of the break the depth increases gradually (M_3 curve) to 2.74 ft over a distance of approximately 152 ft; then a hydraulic jump occurs from a depth of 2.74 ft to 4.81 ft; downstream of the jump the depth remains constant (i.e., normal) at 4.81 ft. ***ANS***

Eq. (10.48): $$h_{L_j} = \frac{(4.81 - 2.74)^3}{4(4.81)2.74} = 0.1695 \text{ ft}$$

So $$P \text{ loss} = \frac{\gamma Q h_{L_j}}{550} = \frac{62.4(400)0.1695}{550} = 7.69 \text{ hp} \qquad \textbf{\textit{ANS}}$$

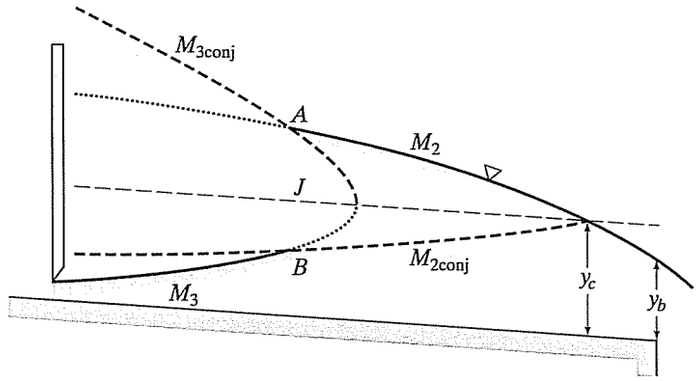
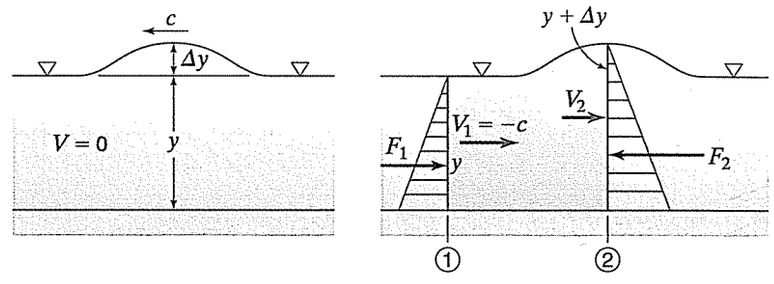


Figure 10.27
Example of the location of a hydraulic jump J when neither conjugate depth is normal.

In the two examples shown in Fig. 10.26, and in some other examples in Fig. 10.20, the flow is at normal depth either before or after the hydraulic jump. These situations are straightforward, and are easy to handle as shown in the preceding illustrative example. There are instances, however, where the flow is not normal either before or after the jump. Such a situation will occur, for example, when water flows out from under a sluice gate and there is a sufficiently high weir downstream of the gate. If the channel is on a steep slope, a hydraulic jump will occur between an S_3 and an S_1 water-surface profile, while if the channel is on a mild slope, the jump will occur between an M_3 and an M_1 profile (Fig. 10.20). Another instance would be between M_3 and M_2 profiles, as shown in Fig. 10.27. This combination is similar to the H_3–H_2 and A_3–A_2 combinations shown on the right-hand side of Fig. 10.20. To find the location of such a jump, varied flow calculations must be made in the downstream direction from the sluice gate and in the upstream direction from the free overfall (or from the weir in the other cases), at the same time computing the conjugate depths for one of these curves using Eq. (10.46). Where the conjugate of one profile crosses the other profile is the only place where the energies are correct for a hydraulic jump to occur. This is at point A or B (the same location) in Fig. 10.27. Graphically plotting these various curves greatly aids the solution process. Also, it will be helpful to reduce the size of the varied flow increments in the neighborhood of the intersection point, which will probably have to be found by interpolation.

10.20 VELOCITY OF GRAVITY WAVES

Consider a small wave of height Δy traveling at a velocity (or ***celerity***) c to the left across the surface of stationary water whose depth is y (Fig. 10.28*a*). Let us now replace this with an equivalent steady flow to the right having a velocity $V_1 = -c$ (Fig. 10.28*b*). Because the wave *velocity or celerity is relative to the water,* the wave is now standing still with respect to the observer. Applying the



(*a*) As seen by a stationary observer

(*b*) As seen by an observer moving with the wave

Figure 10.28
Gravity wave of small amplitude.

principle of continuity to Fig. 10.28*b* we have $V_1 y = V_2(y + \Delta y)$, where V_2 is the average velocity of flow past section 2 in Fig. 10.28*b*.

Applying the momentum principle, $F = \rho Q \Delta V$, to the control volume between sections 1 and 2 for a unit width of channel gives

$$\gamma \frac{y^2}{2} - \gamma \frac{(y + \Delta y)^2}{2} = \frac{\gamma}{g} y V_1 (V_2 - V_1)$$

Substituting the expression for V_2 from continuity and letting $c = -V_1$ in the above gives

$$c = \sqrt{g(y + \Delta y)\left(\frac{y + \frac{1}{2}\Delta y}{y}\right)} \approx \sqrt{g(y + \Delta y)} \approx \sqrt{gy} \qquad (10.49)$$

The latter expressions apply if surface disturbance is relatively small, that is, $\Delta y \ll y$. We see that the velocity (celerity) of a wave will increase as the depth of the water increases. But this is the velocity relative to the water. If the water is flowing, the absolute speed of travel of the wave will be the resultant of the two velocities. Thus

$$c_{abs} = c \pm V \qquad (10.50)$$

Let us now consider only "small" waves, for which $c = \sqrt{gy}$. We are all familiar with the circular waves that spread on a still water surface when disturbed, as when we throw a stone into a pond. When the water is also moving, these spreading patterns are carried with the water, in accordance with Eq. (10.50). In Fig. 10.29 we see how the wave pattern produced by a repeated, regular disturbance varies with water velocity. The familiar pattern of Fig. 10.29*a* is produced when the water is static. In Fig. 10.29*b* the water moving with a low, subcritical velocity carries the circular waves to the right. For the somewhat higher velocity of $V = \sqrt{gy}$, conditions are critical (Sec. 10.10), and $V = c$, so that waves trying to move upstream (to the left in Fig. 10.29*c*) are brought to a standstill as shown; they "hold fast." Thus we see that small disturbance waves

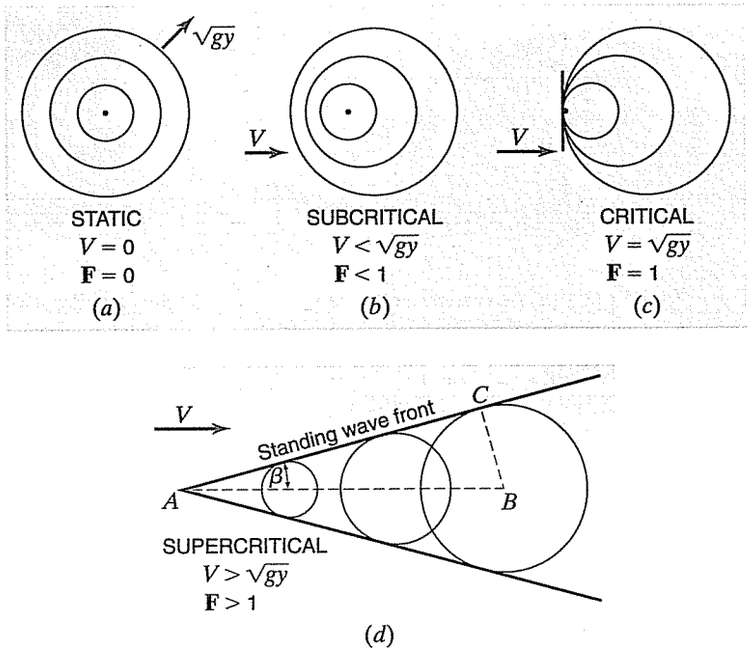


Figure 10.29
Variation of gravity wave patterns with flow conditions.

can only travel upstream if $V < c$, i.e., if the flow is subcritical. For still higher, supercritical velocities, the circles are carried downstream as shown in Fig. 10.29*d*; they *cannot travel upstream*. As the center of the disturbance is carried in time t from A to B for example, the radius grows from B to C. So we see that

Small waves:
$$\sin\beta = \frac{BC}{AB} = \frac{ct}{Vt} = \frac{\sqrt{gy}}{V} = \frac{1}{\mathbf{F}} \tag{10.51}$$

If the disturbance is not intermittent but is continuous, as is usual, we shall not see the individual circles, but we shall see where they reinforce one another. This occurs perpendicular to the flow at the disturbance site for critical flow, see Fig. 10.29*c*, and along line AC in Fig. 10.29*d* for supercritical flow. The perpendicular wave front with critical flow is a special case of Fig. 10.29*d*, because when $V = c$ we have $\mathbf{F} = 1$ and $\beta = 90°$. AC is known as a ***standing wave front*** or a ***disturbance line,*** and is comparable to the oblique shock wave discussed in Sec. 13.11. The angle β is called the ***wave angle,*** and this obviously decreases as V increases. When the channel shape must change in any way, such as at bridge piers, bends, and transitions, these standing waves form with supercritical flow, and because they reflect off side walls, complex patterns of ***cross waves*** can result. Every small irregularity will similarly cause oblique standing waves in supercritical flow. There are many irregularities in natural streams of course, and in order to run rapids canoeists learn to "read" the water surface to interpret subsurface conditions.

If the waves are larger, in other words not small compared to the water depth, we see from Eq. (10.49) that the wave velocity (celerity) is greater than $\sqrt{gy}$, and from Eq. (10.51) that β must also be larger.

EXERCISE

10.20.1 A thin rod is placed vertically in a stream that is 4 ft deep, and the resulting small disturbance wave makes an angle of about 60° with the axis of the stream. Find the approximate velocity of the stream.

10.21 FLOW AROUND CHANNEL BENDS

When a body moves along a curved path of radius r at constant speed, it has a normal acceleration $a_n = V^2/r$ toward the center of the curve, and hence the body must be acted on by a force in that direction. In Fig. 10.30 we can see that this force comes from the unbalanced pressure forces due to the difference in liquid levels between the outer and inner banks of the channel. Assuming that the velocity V across the rectangular section is uniform and that $r \gg B$, we can write, for a unit length of channel, $\Sigma F_n = ma_n$,

$$\frac{\gamma y_2^2}{2} - \frac{\gamma y_1^2}{2} = \frac{\gamma}{g} B \frac{y_1 + y_2}{2} \frac{V^2}{r}$$

which by algebraic transformation becomes

$$\Delta y = y_2 - y_1 = \frac{V^2 B}{gr} \tag{10.52}$$

where B is the top width of the water surface as shown in Fig. 10.30. We can show that Eq. (10.52) applies to any shape of cross section. If the effects of velocity distribution and variations in curvature across the stream are considered,

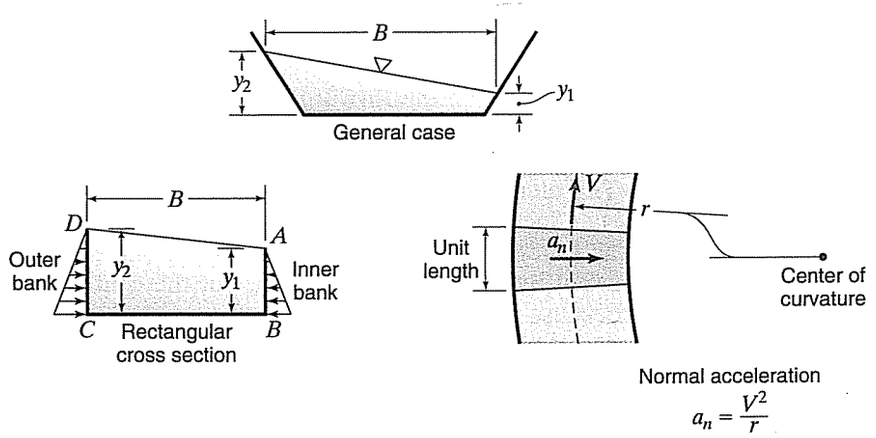


Figure 10.30
Flow in an open-channel bend.

the difference in water depths between the outer and inner banks may be as much as 20% more than that given by Eq. (10.52). If the actual velocity distribution across the stream is known, we can divide the width into sections and compute the difference in elevation for each section. The total difference in surface elevation across the stream is the sum of the differences for the individual sections.

With supercritical flow the complicating factor is the effect of disturbance waves, generated by the very start of the curve. These waves, one from the outside wall and one from the inside, traverse the channel, making an angle β with the original direction of flow, as discussed in Sec. 10.20. The result is a criss-cross wave pattern. The water surface along the outside wall will rise from the beginning of the curve, reaching a maximum at the point where the wave from the inside wall reaches the outside wall. The wave is then reflected back to the inside wall and so on around the bend. Hence in supercritical flow around a bend the increase in water depth on the outer wall is equal to that created by centrifugal effects plus or minus the height of the waves. Field observations indicate that these waves change the subcritical depths by up to $\pm \Delta y/2$, where Δy is the value given by Eq. (10.52). Thus the water-depth increase on the outer wall varies between zero and Δy, i.e., double the subcritical rise, and the depth decrease on the inner wall varies similarly.

Several schemes to lessen the surface rise from wave effects have been investigated. The bed of the channel can be banked so that all elements of the flow are acted upon simultaneously, which is not possible when the turning force comes from the wall only. As in a banked-railway curve, this requires a transition section with a gradually increasing superelevation preceding the main curve. Another method is to introduce a counterdisturbance to offset the disturbance wave caused by the curve. Such a counterdisturbance can be provided by a section of curved channel of twice the radius of the main section, by a spiral transition curve, or by diagonal sills on the channel bed, all preceding the main curve.

Flow around a bend in an open channel is complicated by the development of a secondary flow as the liquid travels around the curve. We discussed the development of secondary flow in the bends of pressure conduits in Sec. 8.26.

The flow around a channel bend is called ***spiral flow*** because superposition of the secondary flow on the forward motion of the liquid causes the liquid to follow a path like a corkscrew (or spiral). We can explain the occurrence of spiral flow by referring to Fig. 10.31. The water surface is superelevated at the outside wall. The element EF is subjected to a centrifugal force mV^2/r, which is balanced by an increased hydrostatic force on the left side, due to the superelevation of the water surface at C above that at D. The element GH is subjected to the same net hydrostatic force inward, but the centrifugal force outward is much less because the velocity is decreased by friction near the bottom. This results in a cross flow inward along the bottom of the channel, which is balanced by an outward flow near the water surface; hence the spiral. This spiral flow is largely responsible for the commonly observed erosion of the outside bank of a river bend, with consequent deposition and building up of a sandbar near the inside bank.

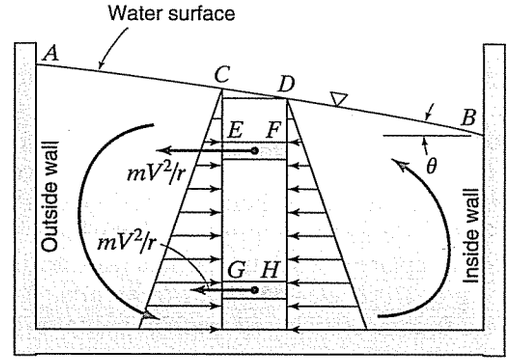


Figure 10.31
Schematic sketch of flow around a bend in a rectangular channel curving to the right, looking downstream, with spiral flow counterclockwise.

SAMPLE PROBLEM 10.11 A rectangular channel 2 m wide carries 4.2 m^3/s in uniform flow at a depth of 0.5 m. What will be the maximum and minimum water depths at the inside and outside walls of a bend in the channel of radius 10 m to the centerline?

Solution

$$V = Q/A = 4.2/(2 \times 0.5) = 4.2 \text{ m/s}$$

Eq. (10.51):
$$\mathbf{F} = \frac{V}{\sqrt{gy}} = \frac{4.2}{\sqrt{9.81 \times 0.5}} = 1.896$$

$\mathbf{F} > 1$, so the flow is supercritical.

Eq. (10.52):
$$\Delta y = \frac{(4.2)^2 2}{9.81(10)} = 0.360 \text{ m}$$

Because of wave action with supercritical flow, Δy varies from zero to double the value computed. Thus the extreme water depths are as follows:

	Inside wall	Outside wall
y_{max}	0.500 m	0.860 m
y_{min}	0.140 m	0.500 m

ANS

EXERCISE

10.21.1 A rectangular channel 5 m wide carries 8 m^3/s in uniform flow at a depth of 2 m. What will be the maximum difference in water-surface elevations between the inside and outside of a circular bend in this channel if the radius of the bend at the centerline is 20 m?

10.22 TRANSITIONS

Special transition sections are often used to join channels of different size and shape in order to avoid undesirable flow conditions and to minimize head loss. If the flow is subcritical, a straightline transition (Fig. 10.32) with an angle θ of about 12.5° is fairly satisfactory. We also call this a ***wedge transition.*** Without the transition, i.e., with an abrupt change in section with square corners, the corresponding head losses are about two to four times larger. Intermediate between these two types is to replace the abrupt corners with vertical quarter-cylinder segments; this is known as a ***cylinder-quadrant transition.*** At Froude numbers between about 0.5 and 1.0, complex ***warped transitions*** that avoid any sharp angles are advisable.

The head loss in transitions is proportional to the change in the velocity head, and is traditionally expressed as

$$h_{L_t} = k_t \left| \frac{V_2^2}{2g} - \frac{V_1^2}{2g} \right| \tag{10.53}$$

Approximate values of the loss coefficient k_t for the transitions discussed are given in Table 10.5. Notice that the more efficient transitions are the ones that are more expensive to build, and that losses are greater with expanding flows, as usual. Experimenters have found that transition lengths need to be at least 2.25 times the change in the water surface width.

With supercritical flow ($\mathbf{F} > 1$), surface waves are formed as described in Sec. 10.20, and special procedures are required for transition design.

At a channel entrance from a reservoir or from a larger channel the head loss for a square-edged entrance is about 0.5 times the velocity head. By rounding the entrance, the head loss can be reduced to slightly less than 0.2 times the velocity head.

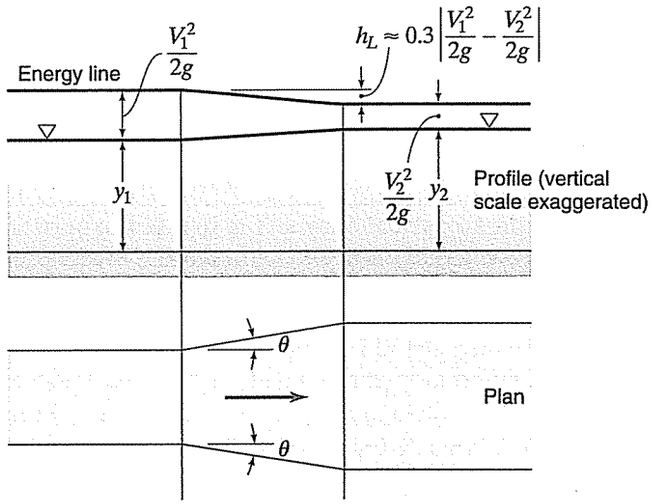


Figure 10.32
Simple open-channel wedge transition for decelerating flow.

TABLE 10.5 **Loss coefficients for channel transitions with subcritical flow**

Transition type	Loss coefficient k_t	
	Contracting	Expanding
Abrupt	0.4–0.5	0.75–1.00
Cylinder-quadrant	0.2	0.5
Wedge	0.1–0.2	0.3–0.5
Warped	0.1	0.3

SAMPLE PROBLEM 10.12 Refer to Fig. 10.32. A rectangular channel changes in width from 4 ft to 6 ft. Measurements indicate that $y_1 = 2.50$ ft and $Q = 50$ cfs. Determine the depth y_2 by (*a*) neglecting head loss, and (*b*) considering the head loss to be given as shown on the figure.

Solution

Eq. (10.23): $y_{c_1} = \left(\frac{(50/4)^2}{32.2}\right)^{1/3} = 1.693$ ft; $y_{c_2} = \left(\frac{(50/6)^2}{32.2}\right)^{1/3} = 1.292$ ft

$$V_1 = 50/(2.5 \times 4) = 5 \text{ fps}; \quad V_2 = 50/6y_2 = 8.33/y_2 \text{ fps}$$

(*a*) $h_L = 0$, so $E_1 = E_2$, i.e., $2.5 + \frac{5^2}{2(32.2)} = y_2 + \frac{(8.33/y_2)^2}{2(32.2)}$

We could solve this with an equation solver, a polynomial solver, or by trial and error (see Sample Prob. 10.8). We obtain $y_2 = 2.75$, 0.702, or −0.559 ft (meaningless).

As we know of nothing to cause supercritical flow, $y_2 = 2.75$ ft ***ANS***

(*b*) $$E_1 = E_2 + h_L = E_2 + 0.3\left(\frac{V_1^2}{2g} - \frac{V_2^2}{2g}\right)$$

$$y_1 + \frac{V_1^2}{2g} = y_2 + \frac{V_2^2}{2g} + 0.3\left(\frac{V_1^2}{2g} - \frac{V_2^2}{2g}\right), \text{ so } y_1 + 0.7\frac{V_1^2}{2g} = y_2 + 0.7\frac{V_2^2}{2g}$$

$$2.5 + 0.7\frac{5^2}{2(32.2)} = 2.77 = y_2 + \frac{0.755}{y_2^2}; \text{ by trial, } y_2 = 2.67, 0.584, \text{ or } -0.479 \text{ ft.}$$

As we know of nothing to cause supercritical flow, $y_2 = 2.67$ ft. ***ANS***

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

EXERCISES

10.22.1 Water flows at 180 cfs from a trapezoidal channel ($b = 10$ ft, side slopes = $2.5H{:}1V$) into a rectangular channel with the same bed width. If the upstream depth is 3.5 ft and the transition is abrupt ($k_t = 0.4$), find the downstream depth.

10.22.2 A rectangular, 3-m-wide channel is connected to a trapezoidal channel ($b = 1$ m, side slopes = $1.5H{:}1V$) by a warped transition. When 7.5 m³/s of water is flowing, the depth in the upstream channel is 1.5 m. (*a*) Find y_2. (*b*) Is the flow through the transition expanding or contracting?

10.23 HYDRAULICS OF CULVERTS

A culvert is a conduit passing under a road or highway. In section, culverts may be circular, rectangular, or oval. Culverts may operate with either a submerged entrance (Fig. 10.33) or a free (unsubmerged) entrance.

Submerged Entrance

In the case of a submerged entrance there are three possible regimes of flow as indicated in Fig. 10.33. Under conditions (*a*) and (*b*) of the figure the culvert is said to be flowing under ***outlet control,*** while condition (*c*) represents ***entrance control.*** In (*a*) the outlet is submerged, possibly because of inadequate channel capacity downstream or due to backwater from a connecting stream. In (*b*) the normal depth y_0 of the flow is greater than the culvert height D, causing the culvert to flow full.[11] Writing energy Eq. (5.28) from the upstream water surface (1) to the water surface at the outlet (2), taking the latter as datum, for both cases (*a*) and (*b*) we obtain

$$0 + \Delta h + 0 - h_L = 0 + 0 + \frac{V^2}{2g}$$

or

$$\Delta h = h_L + \frac{V^2}{2g}$$

where Δh is defined[12] in Figs. 10.33*a* and 10.33*b*, and $V^2/2g$ is the velocity head loss at submerged discharge in case (*a*) or the residual velocity head at discharge in case (*b*). We can find Δh from $\Delta h = y_1 + LS_0 - y_2$, where y is the water depth above invert, and L and S_0 are the barrel length and slope, respectively. The head loss h_L between the two points is equal to h_e, the entrance loss, plus h_f, the friction loss in the barrel. Therefore

$$\Delta h = h_e + h_f + \frac{V^2}{2g} \tag{10.54}$$

Entrance loss is a function of the velocity head in the culvert, while friction loss may be computed using Manning's equation (10.8). Thus, in BG units,[13]

$$\text{Outlet control, case } (a) \text{ or } (b): \quad \Delta h = k_e\frac{V^2}{2g} + \frac{n^2V^2L}{2.21R_h^{4/3}} + \frac{V^2}{2g} \tag{10.55}$$

This expression in BG units can be reduced to

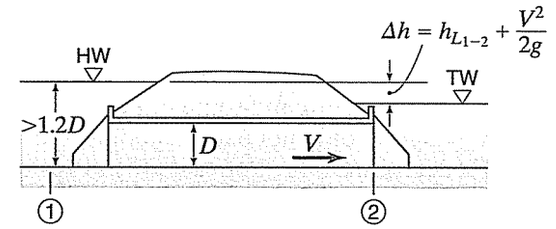
$$\text{Outlet control, case } (a) \text{ or } (b): \quad \Delta h = \left(k_e + \frac{29.2n^2L}{R_h^{4/3}} + 1\right)\frac{V^2}{2g} \tag{10.56}$$

▦: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

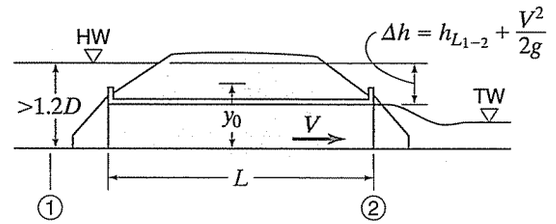
[11] If there is a contraction of flow at the culvert entrance, reexpansion will require about six diameters. Hence a very short culvert may not flow full even though $y_0 \geq D$.

[12] The energy grade line at the entrance and exit is above the actual water surface by the velocity head of the approaching or leaving water. With water ponded at the entrance or exit, this velocity head is usually negligible but should be included in computations if it is of significant magnitude.

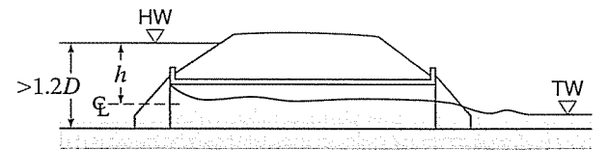
[13] In SI units, the 2.21 disappears from the denominator of the second term on the right-hand side of Eq. (10.55) and the constant 29.2 in the numerator of the second term in parentheses of Eq. (10.56) becomes 19.62.



(*a*) Submerged outlet



(*b*) Normal depth > barrel height



(*c*) Orifice flow, normal depth < barrel height

Figure 10.33
Flow conditions in culverts with submerged entrance.

The entrance coefficient k_e (Fig. 8.13) is about 0.5 for a square-edged entrance and about 0.05 if the entrance is well rounded. If the outlet is submerged, the head loss may be reduced somewhat by flaring the culvert outlet so that the outlet velocity is reduced and some of the velocity head recovered. Tests show that the flare angle should not exceed about 6° for maximum effectiveness.

To determine which of cases (*b*) or (*c*) occurs when the outlet is free (not submerged), we need to find if normal flow in the barrel will fill it. Usually the discharge is known or assumed. For rectangular culverts we can find the normal depth in the usual way from Manning's Eq. (10.8) by trial and error. For circular cross sections it is easier to use Manning's equation to find the diameter which would just flow full ($R_h = D/4$), and to compare that with the actual or proposed diameter. If we are considering alternative slopes with a given barrel diameter, we can rearrange the algebra to solve for the slope that just causes the barrel to flow full.

If normal depth in the culvert is less than the barrel height, with the inlet submerged and the outlet free, the condition (*c*) illustrated in Fig. 10.33*c* will normally result. This culvert is said to be flowing under ***entrance control,*** i.e., the entrance will not admit water fast enough to fill the barrel, and the discharge is determined by the entrance conditions. The inlet then functions like an orifice for which

Entrance control, case (*c*):
$$Q = C_d A\sqrt{2gh} \tag{10.57}$$

where h is the head on the center of the orifice[14] and C_d is the orifice coefficient of discharge. The head required for a given flow Q is therefore

Entrance control, case (*c*):
$$h = \frac{1}{C_d^2}\frac{Q^2}{2gA^2} \tag{10.58}$$

It is impractical to cite appropriate values of C_d, because of the wide variety of entrance conditions which may occur; for a specific design we must determine C_d from model tests or tests of similar entrances. For a sharp-edged entrance without suppression of the contraction $C_d = 0.62$, while for a well-rounded entrance C_d approaches 1.0. If the culvert is set with its invert at stream-bed level, the contraction is suppressed at the bottom. Flared wingwalls at the approach may also cause partial suppression of the side contractions.

Free Entrance

Some box culverts may be designed so that the top of the box forms the roadway. In this case the headwater should not submerge the inlet, and one of the flow conditions of Fig. 10.34 (free entrance) will exist. In cases (*a*) and (*b*) critical depth in the barrel controls the headwater elevation, while in case (*c*) the tailwater elevation is the control. In all cases we can compute the headwater elevation

[14] Equation (10.57) applies for an orifice on which the head h to the center of the orifice is large compared with the orifice height D. When the headwater depth is 1.20D (that is, $h = 0.7D$), an error of 2% results. In the light of other uncertainties in the design this error can be ignored.

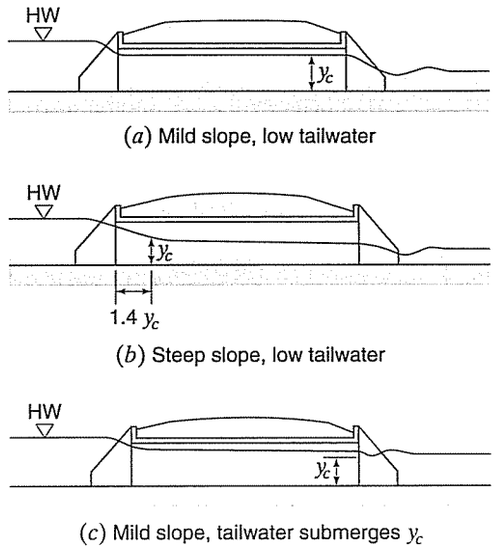


Figure 10.34 Flow conditions in culverts with free entrance.

using the principles of open-channel flow discussed in this chapter with an allowance for entrance and exit losses.

When the culvert is on a steep slope [case (*b*)], critical depth will occur at about 1.4y_c downstream from the entrance. The water surface will impinge on the headwall when the headwater depth is about 1.2D if y_c is 0.8D or more. Since it would be inefficient to design a culvert with y_c much less than 0.8D, a headwater depth of 1.2D is approximately the boundary between free-entrance conditions (Fig. 10.34) and submerged-entrance conditions (Fig. 10.33).

SAMPLE PROBLEM 10.13 A culvert under a road must be 30 m long, have a slope of 0.003, and carry 4.3 m³/s. If the maximum permissible headwater level is 3.6 m above the culvert invert, what size of corrugated-pipe culvert ($n = 0.025$) would you select? Neglect velocity of approach. Assume a square-edged inlet with $k_e = 0.5$ and $C_d = 0.65$. The outlet will discharge freely.

Solution

Sec. 10.23: Assume the headwater depth > 1.2D so that the entrance is submerged per Fig. 10.33. Therefore assume that $D <$ HW depth/1.2 = 3.6 m/1.2 = 3.0 m. Given the discharge is free, so Fig. 10.33*a* cannot apply. Conditions are those of either Fig. 10.33*b* or 10.33*c*.

Assume case (*b*), Fig. 10.33, i.e., that the barrel flows full.

$$V = \frac{Q}{A} = \frac{4.3}{\pi D^2/4} = \frac{5.47}{D^2}; \quad R = D/4$$

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

Fig. 10.33*b*: $$\Delta h = h_{L_{1-2}} + \frac{V^2}{2g} = (y_1 - y_2) + (z_1 - z_2) = y_1 - y_2 + S_0 L$$
$$= 3.6 - D + 0.003(30) = 3.69 - D$$

Eq. (10.56): $$\Delta h = \left[0.5 + \frac{19.62(0.025)^2 30}{(D/4)^{4/3}} + 1\right]\frac{5.47^2}{2(9.81)D^4}$$
$$= \left(1.5 + \frac{2.34}{D^{4/3}}\right)\frac{1.528}{D^4}$$

Equating the two Δh expressions and simplifying,

$$3.69 = D + \left(1.5 + \frac{2.34}{D^{4/3}}\right)\frac{1.528}{D^4}$$

By trial and error (see Sample Probs. 3.5 and 5.8) or by equation solver (Appendixes C.1–C.3, D), $D = 1.196$ m. Thus the first assumption (submerged entrance with $D < 3$ m) is correct.

Now determine if we have case (*b*) or case (*c*); to do this, find the maximum diameter d_0 that will just flow full with normal (uniform) flow:

Eq. (10.8*b*): $$4.3 = \frac{1}{0.025}\frac{\pi d_0^2}{4}\left(\frac{d_0}{4}\right)^{2/3}(0.003)^{1/2}; \quad d_0 = 1.994 \text{ m}$$

As $D < d_0$, the culvert runs full, we do have case (*b*), as assumed. These assumptions and analysis are valid. $D = 1.196$ m.

Use standard $D = 1.2$ m ***ANS***

EXERCISE

10.23.1 Repeat Sample Prob. 10.13 for the situation where the culvert must be 100 m long.

10.24 FURTHER TOPICS IN OPEN-CHANNEL FLOW

The basics of open-channel flow discussed in this chapter are just an introduction to many more advanced subjects.

With steady gradually varied flow, a variety of specialized techniques have been devised to solve for water-surface profiles in irregular, natural channels. Ways have been developed to treat the dividing of flow around islands, and the joining of flows at ***confluences.*** When a canal connects two reservoirs having varying levels, the discharge in the canal, which depends on these levels, is called its ***delivery,*** and we can develop ***delivery curves*** to relate the discharge to the depth

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

at each end. When flow from a reservoir or lake enters a channel with a mild slope, the discharge may depend on the length of the channel if there is a downstream control such as a contraction or a free overfall; we call this ***the discharge problem.*** Steady flow along a channel may vary with distance, such as in gutters and with side-channel spillways. We refer to this as ***spatially varied flow,*** for which the water-surface profiles are usually solved by trial-and-error procedures.

Near channel ***appurtenances*** like weirs (Secs. 11.11 and 11.12), spillways (Sec. 11.13), bridge piers, flumes, transition structures (Sec. 10.22), culverts (Sec. 10.23), and stilling basins (Sec. 10.18), an analysis of the resulting steady rapidly varied flow conditions is often quite involved. These are specialized subjects, which depend strongly on details of the appurtenances. ***Standing waves*** and ***cross waves,*** introduced in Sec. 10.20, are rapidly varying flow phenomena that often occur with supercritical flow. They can be very complex, and are strongly influenced by the channel shape.

With ***unsteady flow*** in open channels the depth and/or the velocity varies with time at a point. This is an advanced subject not addressed in this text. We can vary the flow gradually or rapidly. Gradually varied unsteady flow includes flood waves, tidal flows, and waves resulting from the gradual change of channel control structures such as gates. It is governed by differential equations that usually require numerical methods and computers to solve them. Rapidly varied unsteady flow includes such flows as ***pulsating flow (roll waves),*** which occurs in supercritical flow on very steep slopes, and ***surges*** or ***surge waves*** (moving hydraulic jumps) caused by the rapid operation of a control structure or the sudden failure of a dam. In some locations surges are caused by tidal action, when the surge is known as a ***tidal bore.*** Although we can solve some very simplified problems of these types analytically, in most cases more realistic problems must be solved by numerical methods using computers.

Phenomena governed by density include ***salt wedges*** at estuaries where fresh river water overlies saline ocean water, and similar ***cooling water wedges*** in rivers caused by waste heat from power plants. Density effects in lakes and reservoirs in many cases result in an annual overturning of the contents due to seasonal temperature changes. ***Density currents*** can also be caused by sediment loads, which in turn relate to the subjects of ***scour*** and ***sediment transport.***

Other more advanced phenomena of interest arise when we are analyzing the fate of pollutants, and include ***advection*** and mixing processes known as ***turbulent diffusion*** and ***dispersion.*** Again, solution of these processes usually requires numerical methods using computers.

Excellent books specializing in open-channel flow include references 20, 30, and 38 in Appendix E.

PROBLEMS

10.1 For the channel of Sample Prob. 10.1 compute the "open-channel Reynolds number" assuming that water at 50°F is flowing. Refer to Fig. 8.11 to verify whether or not the flow is fully rough. Determine e from Fig. 8.11 and compare it with the value computed in the example.

10.2 Assuming the values of f versus **R** and e/D given for pipes in Fig. 8.11 also apply to open channels, find the rate of discharge of water at 60°F in a 100-in-diameter smooth concrete pipe flowing half full, if the pipe is laid on a grade of 2 ft/mile. Why must D be replaced by $4R_h$?

10.3 Figure P10.3 shows a cross section of a canal that is to carry 2000 cfs. The canal is lined with concrete, for which n is 0.014. (a) What is the slope of this canal, and what is the drop in elevation per mile? (b) If the flow in the canal were to decrease to 1000 cfs, all other data, including the slope and n, being the same, what would be the depth of the water?

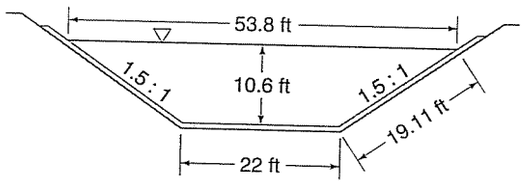


Figure P10.3

10.4 In Prob. 10.3 (a) find the corresponding value of e and compare it with values previously given for concrete pipe. Does it fall in the range given?

10.5 Figure P10.5 shows a cross section of a canal forming a portion of the Colorado River Aqueduct, which is to carry 44 m³/s. The canal is lined with concrete, for which n is 0.014. (a) What is the slope of this canal, and what is the drop in elevation per kilometer? (b) If the flow in the canal were to decrease to 22 m³/s, all other data, including the slope and n, being the same, what would be the depth of the water?

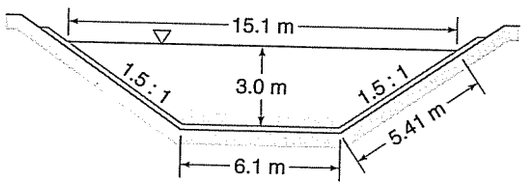


Figure P10.5

10.6 In Prob. 10.5 find the corresponding value of e and compare it with values previously given for concrete pipe. Does it fall in the range given?

10.7 A monolithic concrete inverted siphon is circular in cross section and is 20 ft in diameter. Obviously, it is completely filled with water. (a) If $n = 0.013$, find the slope of the hydraulic grade line for a flow of 2100 cfs. (b) Solve the same problem using the methods of Chap. 8 and assuming a mean value of e from Table 8.1 for concrete pipe. Compare the result with that of part (a).

10.8 A monolithic concrete inverted siphon on the Colorado River Aqueduct is circular in cross section and is 4.88 m in diameter. Obviously, it is completely filled with water. (a) If $n = 0.013$, find the slope of the hydraulic grade line for a flow of 45 m³/s. (b) Solve the same problem using the methods of Chap. 8 and assuming a mean value of e from Table 8.1 for concrete pipe. Compare the result with that of part (a).

10.9 A 30-in-diameter pipe is known to have a Manning's n of 0.014. What is Manning's n for a 96-in-diameter pipe having exactly the same e as the smaller pipe?

10.10 Refer to Fig. X10.3.6. Find the flow rate at water depths of 1, 2.5, 4, 5.5, and 7 ft if $n = 0.025$ and $S = 0.0020$. The dimensions are as follows: $a = 4$ ft, $b = 5$ ft, $d = 7$ ft, and $w = 35$ ft.

10.11 Using Eq. (10.12), determine the depth below the surface at which the velocity is equal to the mean velocity. Also find the average of the velocities at 0.2 and 0.8 depths. Let $y = 6$ ft, $S = 0.0005$, and $n = 0.024$.

10.12 Prove that the most efficient hydraulic trapezoidal cross section discussed in Sec. 10.6 envelopes a semicircle with its center in the water surface.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.13 Consider a variety of rectangular sections, all of which have a cross-sectional area of 20 ft². Plot the hydraulic radii versus channel widths for a range of channel widths from 2 to 20 ft and note the depth:width ratio when R_h is maximum.

10.14 Set up a general expression for the wetted perimeter P of a trapezoidal channel in terms of the cross-sectional area A, depth y, and angle of side slope ϕ (Fig. P10.14). Then differentiate P with respect to y with A and ϕ held constant. From this prove that $R_h = y/2$ for the section of greatest hydraulic efficiency (i.e., smallest P for a given A).

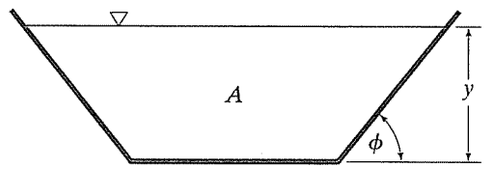


Figure P10.14

10.15 Using the results of Prob. 10.14, prove that the most efficient triangular section is the one with a 90° vertex angle.

10.16 The amount of water to be carried by a canal excavated in smooth earth ($n = 0.030$) is 370 cfs. It has side slopes of 2:1 (see Fig. P10.16), and a bed slope of 2.5 ft/mile. (a) If the depth of water y is to be 5 ft, what must be the bottom width b? (b) How does this compare with the depth and bottom width for the most efficient trapezoidal section for the given conditions?

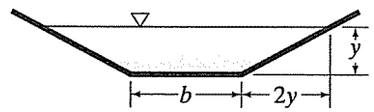


Figure P10.16

10.17 The amount of water to be carried by a canal excavated in smooth earth ($n = 0.030$) is 12 m³/s. It has side slopes of 2:1 (see Fig. P10.16), and a bed slope of 600 mm/km. (a) If the depth of water y is to be 2.0 m, what must be the bottom width b? (b) How does this compare with the depth and bottom width for the most efficient trapezoidal section for the given conditions?

10.18 Refer to Fig. P10.16. (a) If the discharge in the canal ($n = 0.030$) is to be 200 cfs while the depth y is 5 ft and the velocity is not to exceed 150 ft/min, what must be the minimum bottom width b and the maximum drop in elevation per mile? (b) Compare this with the depth and bottom width for maximum efficiency when using the same side slopes, roughness, bed slope, and discharge.

10.19 Refer to Fig. P10.16. (a) If the discharge in the canal ($n = 0.028$) is to be 6 m³/s while the depth y is 1.8 m and the velocity is not to exceed 50 m/min, what must be the minimum bottom width b and the maximum drop in elevation per km? (b) Compare this with the depth and bottom width for maximum efficiency when using the same side slopes, roughness, bed slope, and discharge.

10.20 Prove that the value of θ given in Sec. 10.7 for the point of maximum discharge (assuming constant n) is correct. After differentiating, a trial-and-error type of solution will be found most practical here.

10.21 Evaluate the friction factor f for laminar flow in a wide and shallow open channel in terms of the Reynolds number, and compare it with f for pipe flow using Eq. (8.29). (*Note*: Recall that for a wide and shallow channel the hydraulic radius is approximately equal to the depth.)

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.22 In Sec. 10.8 the velocity distribution in laminar sheet flow was found to be given by $u = (gS/2\nu)y_0^2[1 - (y/y_0)^2]$, where y in this case is the variable distance downward from the surface. Evaluate α, and compare it with the result of Sample Prob. 5.1.

10.23 Consider a wide and shallow rectangular channel on a given slope. With what power of the discharge does the depth vary? With what power of the discharge does the critical depth vary? As the flow increases, does the Froude number increase or decrease? Assume Manning's equation, with a constant value of n.

10.24 Differentiate Eq. (10.17) to obtain the expression for y_c given in Eq. (10.23).

10.25 A flow of 90 cfs is carried in a rectangular channel 10 ft wide at a depth of 1.6 ft. If the channel is made of smooth concrete ($n = 0.016$), find the slope necessary to sustain uniform flow at this depth. What roughness coefficient would be required to produce uniform critical flow for the given rate of discharge on this slope?

10.26 A rectangular channel 15 ft wide carries a flow of 320 cfs. Find the critical depth and the critical velocity for this flow. Find also the critical slope if $n = 0.015$.

10.27 Water flows with a velocity of 6 fps and at a depth of 3 ft in a wide rectangular channel. Is the flow subcritical or supercritical? Find the alternate depth for the same discharge and specific energy by two methods: (a) by direct solution of Eq. (10.17); (b) by use of Fig. 10.13.

10.28 Water is released from a sluice gate in a rectangular channel 6 ft wide such that the depth is 3 ft and the velocity is 18 fps (Fig. P10.28). Find (a) the critical depth for this specific energy; (b) the critical depth for this rate of discharge; (c) the type of flow and the alternate depth by either direct solution or the discharge curve.

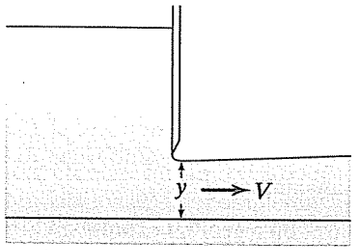


Figure P10.28

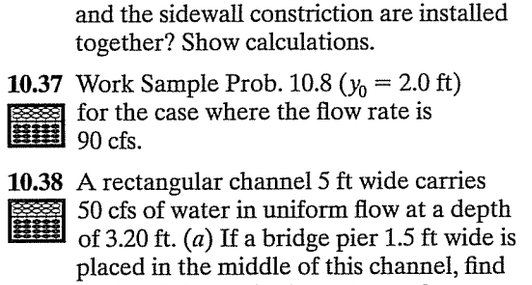
10.29 Water is released from a sluice gate in a rectangular channel 1.5 m wide such that the depth is 0.6 m and the velocity is 4.5 m/s (Fig. P10.28). Find (a) the critical depth for this specific energy; (b) the critical depth for this rate of discharge; (c) the type of flow and the alternate depth by either direct solution or the discharge curve.

10.30 A trapezoidal canal with side slopes of 2:1 ($H:V$) has a bottom width of 10 ft and carries a flow of 600 cfs. (a) Find the critical depth and critical velocity. (b) If the canal is lined with brick ($n = 0.015$), find the critical slope for the same rate of discharge.

10.31 A trapezoidal canal with side slopes of 1:1 has a bottom width of 4 m and carries a flow of 25 m³/s. (a) Find the critical depth and critical velocity. (b) If the canal is lined with brick ($n = 0.015$), find the critical slope for the same rate of discharge.

10.32 For a circular conduit with a diameter of 12 ft, compute the specific energy for a flow of 120 cfs at depths of 2, 4, 6, and 10 ft assuming $\alpha = 1.0$. At what depth is E the least? Check to see if Eq. (10.33) is satisfied at this depth.

10.33 Figure P10.33 describes the cross section of an open channel for which $S_0 = 0.03$ and $n = 0.020$. The sketch is drawn to the scale shown. When the flow rate is 120 cfs, find (a) the depth for uniform flow and (b) the critical depth. (*Hint:* The cross-sectional area may be found by planimetry or counting squares and the wetted perimeter by use of dividers.)



y, ft	x_L, ft	x_R, ft
0	0	0
1	1.5	1.1
2	2.6	2.0
3	3.25	3.3
4	3.6	5.5
5	3.85	8.2

Figure P10.33

10.34 Refer to the figure, table, and hint for Prob. 10.33, replacing feet dimensions with meters. Let the slope be 0.009 with $n = 0.020$. When the flow rate is 60 m³/s, find (a) the depth for uniform flow; (b) the critical depth.

10.35 Work Sample Prob. 10.8 ($y_0 = 2.0$ ft) for the case where the flow rate is 275 cfs.

10.36 A flow of 2.0 m³/s is carried in a rectangular channel 1.8 m wide at a depth of 1.0 m. Will critical depth occur at a section where (a) a frictionless hump 180 mm high is installed across the bed, (b) a frictionless sidewall constriction (with no hump) reduces the channel width to 1.4 m, and (c) the hump and the sidewall constriction are installed together? Show calculations.

10.37 Work Sample Prob. 10.8 ($y_0 = 2.0$ ft) for the case where the flow rate is 90 cfs.

10.38 A rectangular channel 5 ft wide carries 50 cfs of water in uniform flow at a depth of 3.20 ft. (a) If a bridge pier 1.5 ft wide is placed in the middle of this channel, find the local change in the water-surface elevation. (b) What is the minimum width of constricted channel which will not cause a rise in water surface upstream?

10.39 A rectangular channel 1.2 m wide carries 1.1 m³/s of water in uniform flow at a depth of 0.85 m. If a bridge pier 0.3 m wide is placed in the middle of this channel, find the local change in the water-surface elevation. What is the minimum width of constricted channel which will not cause a rise in water surface upstream?

10.40 A rectangular channel 12 ft wide carries 24 cfs in uniform flow at a depth of 0.32 ft. Find the local change in water-surface elevation caused by a 0.15-ft-high obstruction across the channel bed.

10.41 Fifty cubic feet per second of water flows uniformly in a 6-ft-wide rectangular channel at a depth of 2.5 ft (Fig. P10.41). What is the change in water-surface elevation at a section contracted to a 4-ft width with an 0.2-ft depression in the bottom?

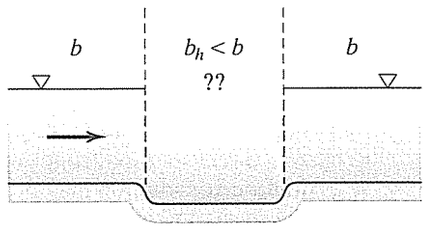


Figure P10.41

10.42 Water flows uniformly in a 2.0-m-wide rectangular channel at a rate of 1.6 m³/s and a depth of 0.75 m (Fig. P10.41). What is the change in water-surface elevation at a section contracted to a 1.4-m width with an 80-mm depression in the bottom?

10.43 A 5-ft-wide rectangular flume ($n = 0.012$) carries 60 cfs of water. At one point the water depth is found to be 4 ft; 1000 ft downstream it is measured at 3 ft. Calculate the bed slope of the flume, using one reach. Sketch the flume bed, water surface, and energy line, to check that the answer is reasonable.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.44 A 1.6-m-wide rectangular flume ($n = 0.013$) carries 1.9 m³/s of water. At one point the water depth is found to be 1.3 m; 320 m downstream it is measured at 1.0 m. Calculate the bed slope of the flume, using one reach. Sketch the flume bed, water surface, and energy line, to check that the answer is reasonable.

10.45 A 5-ft-wide rectangular flume ($S_0 = 0.01$, $n = 0.012$) carries 60 cfs of water. Find the depth 50 ft downstream from a section where the flow is 1.5 ft deep. Use only one reach. Is the flow subcritical or supercritical? [*Note:* In this case it will not be possible to make a direct solution from Eq. (10.37). A trial-and-error solution may best be set in the form of a table with the headings y_2, V_2, $\overline{V}$, A_2, P_2, R_2, $\overline{R}$, $\overline{R}^{2/3}$, S, etc.]

10.46 A 1.6-m-wide rectangular flume ($S_0 = 0.008$, $n = 0.013$) carries 1.9 m³/s of water. Find the depth 17 m downstream from a section where the flow is 0.50 m deep. Use only one reach. Is the flow subcritical or supercritical? [*Note:* In this case it will not be possible to make a direct solution from Eq. (10.37). A trial-and-error solution may best be set in the form of a table with the headings y_2, V_2, $\overline{V}$, A_2, P_2, R_2, $\overline{R}$, $\overline{R}^{2/3}$, S, etc.]

10.47 Repeat Prob. 10.45, but increase the 50-ft distance to 500 ft.

10.48 Repeat Prob. 10.46, but increase the 17-m distance to 170 m.

10.49 A rectangular flume 9 ft wide is built of planed timber ($n = 0.012$) on a bed slope of 0.3 ft per 1000 ft, ending in a free overfall (Fig. P10.49). If the measured depth at the fall is 1.67 ft, find (*a*) the rate of flow; (*b*) the distance upstream from the fall to where the depth is 3.6 ft. [*Note:* Assume that critical depth occurs at a distance of $4y_c$ upstream from the fall, and use reaches with end depths of 2.5, 2.8, 3.2, and 3.6 ft.]

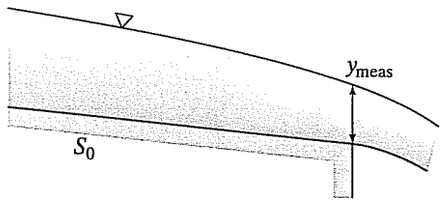


Figure P10.49

10.50 A rectangular flume 3 m wide is built of planed timber ($n = 0.012$) on a bed slope of 200 mm per km, ending in a free overfall (Fig. P10.49). If the measured depth at the fall is 0.55 m, find (*a*) the rate of flow; (*b*) the distance upstream from the fall to where the depth is 1.2 m. [*Note:* Assume that critical depth occurs at a distance of $4y_c$ upstream from the fall, and use reaches with end depths of 0.82, 0.9, 1.0, and 1.2 m.]

10.51 A wide and shallow rectangular channel dredged in earth ($n = 0.035$) is laid on a slope of 9 ft/mi and carries a flow of 90 cfs/ft of width. (*a*) Find the water depth 2 miles upstream from a section where the depth is 26.1 ft, using a single reach. (*b*) Compare the result with that obtained using three reaches of equal length.

10.52 The slope of a stream of rectangular cross section is $S_0 = 0.0003$, the width is 170 ft, and the value of the Chézy C is 78.3 ft$^{1/2}$/s. (*a*) Find the depth for a uniform flow of 101.54 cfs per foot of width of the stream. (*b*) If a dam raises the water level so that at a certain distance upstream the increase is 5 ft, how far from this latter section will the increase be only 1 ft? Use reaches with 1-ft depth increments.

10.53 A portion of an outfall sewer is approximately a circular conduit 5 ft in diameter and with a slope of 1 ft in 1100 ft. It is of brick, for which $n = 0.013$ (neglect variation with depth). (*a*) What would be its maximum capacity for uniform flow. (*b*) If it discharges 120 cfs with a depth at the end of 2.90 ft (Fig. P10.53), how far back from the end must it become a pressure conduit? Proceeding from the mouth upstream, find by tabular solution the length of sewer that is not flowing full. Use three reaches with equal depth increments. (*c*) Find the pressure at the top and bottom of the pipe at the section that is 600 ft upstream from the point at which the sewer no longer flows full. Sketch the energy line and hydraulic grade line from this point to the end of the sewer.

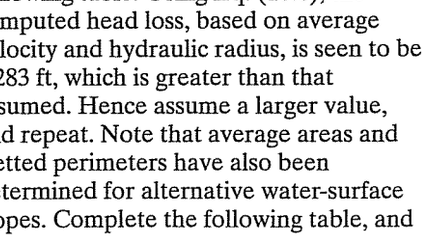


Figure P10.53

10.54 For the channel of Prob. 10.33 find the distance between a section where the depth is 3.5 ft to another where the depth is 3.0 ft. Which section is upstream? Use only one reach. See the hint of Prob. 10.33.

10.55 When the flow in a certain natural stream is 7600 cfs, it is required to find the elevation of the water surface at different sections upstream from a certain initial point. A survey of the channel shows that conditions are fairly similar for a length of 1500 ft upstream from the initial point, and then beyond that there is another stretch of 2200 ft, and so on. Assuming a rise in the water surface in the distance of 1500 ft to be 0.20 ft, a study of the stream bed shows the average values of the area and wetted perimeter to be as given in the following table. Using Eq. (10.6), the computed head loss, based on average velocity and hydraulic radius, is seen to be 0.283 ft, which is greater than that assumed. Hence assume a larger value, and repeat. Note that average areas and wetted perimeters have also been determined for alternative water-surface slopes. Complete the following table, and find the probable rise in elevation in the first 1500 ft. [In a similar manner, the rises in other lengths may be computed, and the sum of all of them up to the desired point will give the elevation at that point above the initial.] Assume $n = 0.036$.

Assumed rise SL, ft	Average A, ft²	Average P, ft	Average R, ft	Average V, fps	$SL = LV^2/C^2R$, ft
0.20	3100	350	8.86	2.45	0.283
0.25	3180	359	8.86		
0.26	3190	360	8.86		
0.27	3220	363	8.86		
0.28	3230	364	8.86		

10.56 Classify the water-surface profile of Prob. 10.45 as one of the forms shown in Fig. 10.20. Show all necessary calculations. Sketch the profile.

10.57 Classify the water-surface profile of Prob. 10.46 as one of the forms shown in Fig. 10.20. Show all necessary calculations. Sketch the profile.

10.58 A trapezoidal canal dredged in smooth earth ($n = 0.030$) has a bottom width of 15 ft, side slopes of 1 : 1, and a bed slope $S_0 = 0.0003$. With a flow of 800 cfs, $y_c = 4.05$ ft, and $y_0 = 10.8$ ft. Find the distance (along an M_2 curve) from a free overfall back to where the depth is 10 ft. Use reaches with end depths of 6, 8, and 10 ft.

10.59 A rectangular channel 10 ft wide carries 120 cfs in uniform flow at a depth of 2.00 ft. Suppose that an obstruction such as a submerged weir is placed across the channel, rising to a height of 8 in above the bottom. (*a*) Does this obstruction cause a hydraulic jump to form upstream? Why or why not? (*b*) Find the water depth over the obstruction, and just upstream of it. Classify the surface profile, if possible, to be found upstream from the weir. Sketch the resulting water-surface profile and energy line, showing y_c and y_0.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.60 Suppose that the slope and roughness of the channel in Prob. 10.59 are such that uniform flow of 120 cfs occurs at a depth of 1.20 ft. Consider an obstruction rising 3 in above the bottom of the channel. Will a hydraulic jump form upstream? Classify the surface profile found just upstream from the obstruction.

10.61 Analyze the water-surface profile in a long rectangular channel ($n = 0.014$). The channel is 12 ft wide, the flow rate is 480 cfs, and there is an abrupt change in slope from 0.0019 to 0.0180. Make a sketch showing normal depths, critical depths, and water-surface profile types.

10.62 Repeat Prob. 10.61 for the case where the flow rate is 180 cfs.

10.63 Repeat Prob. 10.61 for the case where the slope change is from 0.0019 to 0.0007, and the flow rate is 180 cfs. Compute the approximate distance upstream from the break to the point where normal depth occurs, using one reach.

10.64 In an 8-ft-wide rectangular channel ($n = 0.015$) water flows at 300 cfs. A low dam (broad-crested weir) placed in the channel raises the water to a depth of 8.0 ft. Analyze the water-surface profile upstream from the dam back to uniform depth if the channel slope is (*a*) 0.0006, (*b*) 0.0009, and (*c*) 0.006.

10.65 Derive Eq. (10.48) in the manner suggested in Sec. 10.18.

10.66 In a rectangular channel 10 ft wide with a flow of 200 cfs the depth is 1 ft. If a hydraulic jump is produced, (*a*) what will be the depth immediately after it? (*b*) What will be the loss of energy?

10.67 In a rectangular channel 4 m wide with a flow of 7.65 m^3/s the depth is 0.4 m. If a hydraulic jump is produced, (*a*) what will be the depth immediately after it? (*b*) What will be the loss of energy?

10.68 Repeat Prob. 10.66 for the case where the channel bed slopes at 10°. For this slope, jump length $\approx 4y_2$. Assume friction force = 400 lb/ft of width. Also find the horsepower loss.

10.69 Repeat Prob. 10.67 for the case where the channel bed slopes at 10°. For this slope, jump length $\approx 4y_2$. Assume friction force = 6 kN/m of width. Also find the power (kW) loss.

10.70 The tidal bore, which carries the tide into the estuary of a large river, is an example of an abrupt translatory wave, or moving hydraulic jump. Suppose such a bore rises to a height of 13 ft above the normal low-tide river depth of 10 ft. The speed of travel of the bore upstream is 13 mph. Find the velocity of the undisturbed river. Does this represent subcritical or supercritical flow? [*Note:* The theory developed in Sec. 10.18 is based on the hydraulic jump in a fixed position. In the case of a moving jump, all kinematic terms must be considered relative to the moving wave as a frame of reference.]

10.71 A hydraulic jump occurs in a triangular flume having side slopes at 1:1. The flow rate is 18 cfs and the depth before the jump is 1.2 ft. Find the depth after the jump and the power loss in the jump.

10.72 A hydraulic jump occurs in a triangular flume having side slopes at 1:1. The flow rate is 0.45 m^3/s and the depth before the jump is 0.30 m. Find the depth after the jump and the power loss in the jump.

10.73 A wide and shallow rectangular channel with bed slope $S_0 = 0.0004$ and roughness $n = 0.022$ carries a steady flow of 65 cfs/ft of width. If a sluice gate (Fig. 11.34) is so adjusted as to produce a minimum depth of 1.6 ft in the channel, determine whether a hydraulic jump will form downstream, and if so, find (using one reach) the distance from the gate to the jump.

10.74 Solve for and sketch the water-surface profile in a very long 10-ft-wide rectangular open channel of rough masonry ($n = 0.017$) when the flow rate is 400 cfs. The bed slope is 0.020, and a 5-ft-high streamlined hump is at the downstream end. Locate all hydraulic jumps, treating any gradually varied flow as a single reach. How does the water-surface *elevation* vary between key points?

10.75 At a point in a shallow lake, the wave from a passing boat is observed to rise 0.5 ft above the undisturbed water surface. The observed speed of the wave is 8 mph. Find the approximate depth of the lake at this point. Compute it three ways and comment on them.

10.76 At a point in a shallow lake, the wave from a passing boat is observed to rise 300 mm above the undisturbed water surface. The observed speed of the wave is 16 km/h. Find the approximate depth of the lake at this point. Compute it three ways and comment on them.

10.77 A rectangular channel 12 ft wide carries 360 cfs under uniform flow conditions. By how much should the outside wall be elevated above the inside wall for a bend of 45-ft radius to the centerline of the channel if (*a*) the normal depth is 4.4 ft and (*b*) the normal depth is 2.2 ft.

10.78 Refer to Fig. 10.32. A rectangular channel changes in width from 5 to 7.5 ft. Measurements indicate that $y_1 = 3.00$ ft and $Q = 62$ cfs. Determine the depth y_2 by (*a*) neglecting head loss; (*b*) considering the head loss to be given as shown on the figure.

10.79 A rectangular channel changes in width from 1.35 m to 2.0 m with the wedge-shaped transition shown in Fig. 10.32. Given that $y_1 = 0.75$ m and $Q = 1.5$ m^3/s, find the depth y_2 by (*a*) neglecting head loss and (*b*) using a loss coefficient k_t of 0.4. (*c*) Does the head loss increase or decrease the flow expansion?

10.80 Flow from a trapezoidal channel (20-ft bed width, 1:1 side slopes) enters a 20-ft-wide rectangular channel via a cylinder-quadrant transition (Fig. P10.80). (*a*) If the upstream depth is 4 ft when the flow rate is 530 cfs, find the resulting downstream depth. (*b*) If the flow increases to 550 cfs, causing the upstream depth to increase to 4.1 ft, does the downstream depth increase or decrease? By how much?

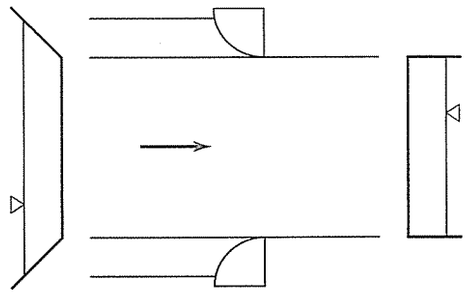


Figure P10.80

10.81 What is the capacity of a 5 ft by 5 ft concrete box culvert ($n = 0.013$) with a rounded entrance ($k_e = 0.05$, $C_d = 0.95$) if the culvert slope is 0.004, the length is 180 ft, and the headwater level is 7 ft above the culvert invert? Assume (*a*) free outlet conditions, (*b*) tailwater elevation 1 ft above top of box at outlet, and (*c*) tailwater elevation 2 ft above top of box at outlet. Neglect headwater and tailwater velocity heads.

10.82 Repeat Prob. 10.81(*a*) for the cases where the culvert slope is (*a*) 0.03 and (*b*) 0.07.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

10.83 A culvert under a road must carry 4.5 m^3/s. (*a*) If the culvert length is 32 m, the slope is 0.004, and the maximum permissible headwater level above the culvert invert is 3.8 m, what size of corrugated-pipe culvert ($n = 0.025$) would you select? The outlet will discharge freely. Neglect velocity of approach. Assume square-edged entrance with $k_e = 0.5$, $C_d = 0.65$. (*b*) Repeat for a culvert length of 110 m.

10.84 A 120-ft-long corrugated metal pipe ($n = 0.022$) of 30-in diameter is tested in a laboratory. The headwater is maintained at a level which is 5 ft above the pipe invert at entrance, and the outlet cannot be submerged. Assume a square-edged entrance with $k_e = 0.5$, $C_d = 0.68$, and neglect headwater and tailwater velocity heads. Compute values of Q for S_0 values of 0.0, 0.01, 0.03, and 0.08. As the slope is increased, at what slope does the flow change to condition (*c*) of Fig. 10.33?

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

CHAPTER 11
Fluid Measurements

Both the engineer and the scientific investigator are often faced with the problem of measuring various fluid properties such as density, viscosity, and surface tension. Also, measurements are often required of various fluid phenomena, as pressure, velocity, and flow rate. In this chapter only the principles and theory of such measurements will be discussed. Detailed information on the various measuring devices can be found elsewhere in the literature.[1] This chapter draws on many concepts presented in previous chapters; thus it also serves as a good review.

11.1 MEASUREMENT OF FLUID PROPERTIES

Most commonly we measure ***liquid density*** by weighing a known volume of the liquid to find its specific weight (Sec. 2.3), and then dividing by the acceleration due to gravity. Another technique makes use of hydrostatic weighing, where a nonporous solid object of known volume is weighed (*a*) in air and then (*b*) in the liquid whose density is to be determined. The ***hydrometer*** is a variation of this technique. We calculate the densities from fluid statics. Another, though not very accurate, way to determine liquid density is to place two immiscible liquids in a U tube, one of known density, the other of unknown density. From fluid statics we can find the unknown density. These various techniques of measuring liquid density are illustrated in Exers. 11.1.1–11.1.4 and Probs. 11.1 and 11.2.

The measurement of ***viscosity*** is generally made with a device known as a ***viscometer.*** Various types of viscometers are available. They all depend on the creation of laminar-flow conditions. We shall confine this discussion to the measurement of the viscosity of liquids. Since viscosity varies considerably with temperature, it is essential that the fluid be at a constant temperature when a measurement is being made. This is generally accomplished by immersing the device in a constant-temperature bath.

Several types of ***rotational viscometers*** are available. These generally consist of two concentric cylinders that are rotated with respect to one another.

[1] See, for example, *Fluid Meters: Their Theory and Application,* 6th ed., American Society of Mechanical Engineers, New York, 1971.

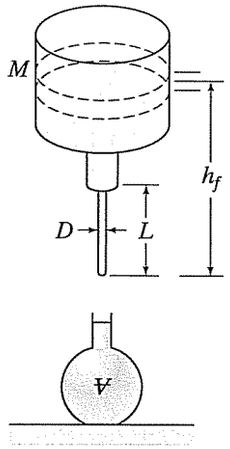


Figure 11.1
Falling-head tube-type viscometer.

We fill the narrow space between them with liquid whose viscosity is to be measured. The rate of rotation under the influence of a given torque is indicative of the viscosity of the liquid. One difficulty with this type of viscometer is that we must account for mechanical friction, and this is difficult to deal with accurately.

The ***tube-type viscometer*** is perhaps the most reliable. Figure 11.1 shows the Saybolt viscometer. In this device the liquid is originally at M, with the bottom of the tube plugged. The plug is removed, and the time required for a certain volume of liquid to pass through the tube is a measure of the kinematic viscosity of the liquid. In this device the flow is unsteady and the tube is of such small diameter that we can assume that the flow is laminar. Substituting the expression for mean velocity V for laminar flow in tubes (Eq. 8.28) into the continuity expression (Eq. 4.7) gives

$$Q = \frac{\pi D^4 \gamma h_f}{128 \mu L} \tag{11.1}$$

As an approximation, let h_f be the average imposed head during the flow period and let $Q \approx \forall/t$, where $\forall$ is the volume of liquid that flows out of the tube in time t. Substituting $Q = \forall/t$ into Eq. (11.1) and making use of Eq. (2.11), we get

$$\nu = \frac{\pi D^4 h_f}{128 \forall L} gt \tag{11.2}$$

Since D, L, $\forall$, and h_f are constants of the device, $\nu = Kgt$, and the kinematic viscosity is seen to be proportional to the measured time. Equation (11.2) gives good results if the tube is relatively long. However, for a short tube, as with the Saybolt viscometer, we need to apply a correction factor if the tube is too short for the establishment of laminar flow (Sec. 8.8).

There are several other types of tube viscometers, but they are all based on the same principle. Some come with a set of tubes of various diameters so that measurements can be made on liquids with a wide range of viscosities in a

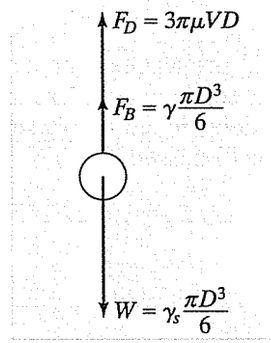


Figure 11.2
Free-body diagram of sphere falling at terminal velocity.

convenient time period. Because the dimensions of such fine tubes cannot be perfectly duplicated, each tube is individually calibrated by measuring the time for a liquid of known viscosity at a given temperature to discharge the standard volume.

The ***falling-sphere viscometer*** is a third type. In such a device we place the liquid in a tall transparent cylinder and drop in a sphere of known weight and diameter. If the sphere is small enough, Stokes' law (Sec. 9.7) will prevail and the fall velocity of the sphere will be approximately inversely proportional to the absolute viscosity of the liquid. We can see that this is so by examining the free-body diagram of such a falling sphere (Fig. 11.2). The forces acting include gravity, buoyancy, and drag. Stokes' law states that if $DV/\nu < 1$, the drag force on a sphere is given by $F_D = 3\pi\mu VD$, where V is the velocity of the sphere and D is its diameter. When the sphere is dropped in a liquid, it will quickly accelerate to terminal velocity, at which $\sum F_z = 0$. Then

$$W - F_B - F_D = \gamma_s \frac{\pi D^3}{6} - \gamma \frac{\pi D^3}{6} - 3\pi\mu VD = 0$$

where γ_s and γ represent the specific weight of the sphere and liquid, respectively. Solving this equation, we get

$$\mu = \frac{D^2(\gamma_s - \gamma)}{18V} \tag{11.3}$$

In the preceding development we assumed that the sphere was dropped into a liquid of infinite extent. In actuality, the liquid will be contained in a tube and a ***wall effect*** will influence the drag force and hence the fall velocity. Experimenters have found that the wall effect[2] can be expressed approximately as

$$\frac{V}{V_t} \approx 1 + \frac{9D}{4D_t} + \left(\frac{9D}{4D_t}\right)^2 \tag{11.4}$$

[2] J. S. McNown, H. M. Lee, M. B. McPherson, and S. M. Engez, Influence of Boundary Proximity on the Drag of Spheres, *Proc. 7th Intern. Congr. Appl. Mech.*, 1948.

where D_t is the tube diameter, and V_t represents the fall velocity in the tube. Equation (11.4) is reliable only if $D/D_t < \frac{1}{3}$.

Other fluid properties such as surface tension, elasticity, vapor pressure, specific heats at constant pressure and constant temperature, and gas constant are commonly determined by physicists, and the techniques for their measurement will not be discussed here.

EXERCISES

11.1.1 A small object has a volume of 0.0070 ft^3 and weighs 1.50 lb in air and 1.18 lb in a liquid. What is the density of the liquid?

11.1.2 A small object has a volume of 300 mL and weighs 15.50 N in air and 10.50 N in a liquid. What is the density of the liquid?

11.1.3 Carbon tetrachloride ($s = 1.59$) is placed in an open U tube. A liquid is poured into one of the legs of the tube. A column of this liquid 19.4 in high balances a carbon tetrachloride column 15.0 in high. What is the specific weight of the liquid? In Sec. 11.1 it is mentioned that this method will give only approximate values. Why is this so?

11.1.4 Carbon tetrachloride ($s = 1.59$) is placed in an open U tube. A liquid is poured into one of the legs of the tube. A column of this liquid 392 mm high balances a carbon tetrachloride column 300 mm high. What is the specific weight of the liquid? In Sec. 11.1 it is mentioned that this method will give only approximate values. Why is this so?

11.1.5 A rotational viscometer is constructed of two concentric cylinders of height 15.0 in. The OD of the inner cylinder is 4.00 in, and the ID of the outer cylinder is 4.10 in. When a torque of 6.0 ft·lb is applied to the outer cylinder, it was found to rotate at 1 revolution per 3.8 sec. Find the (absolute) viscosity of the fluid. Neglect mechanical friction.

11.1.6 A rotational viscometer is constructed of two concentric cylinders of height 300 mm. The OD of the inner cylinder is 100.0 mm, and the ID of the outer cylinder is 102.0 mm. When a torque of 8.0 N·m is applied to the outer cylinder, it was found to rotate at 1 revolution per 4.0 s. Find the (absolute) viscosity of the fluid. Neglect mechanical friction.

11.1.7 A tube viscometer similar to the one of Fig. 11.1 has a diameter of 0.0420 in and a length of 3.10 in. A volume of 60 mL flowed through the tube in 128.7 s, causing the vertical distance from the liquid surface in the reservoir to the tube outlet to change from 10.00 to 9.50 in. Find the kinematic viscosity of the liquid.

11.1.8 If 60°F water flows through a tube-type viscometer in 120 sec, how long will it take the same volume of 100°F water to pass through the same viscometer?

11.1.9 If 15°C water flows through a tube-type viscometer in 85 s, how long will it take the same volume of 40°C water to pass through the same viscometer?

11.1.10 A lead sphere ($s = 11.4$) with a diameter of 0.25 in falls through oil ($s = 0.86$) in a 2.25-in-diameter cylinder. If the sphere moves with a constant velocity of 0.150 fps, what is the absolute viscosity of the oil? Check **R** to see if it is less than 1.0.

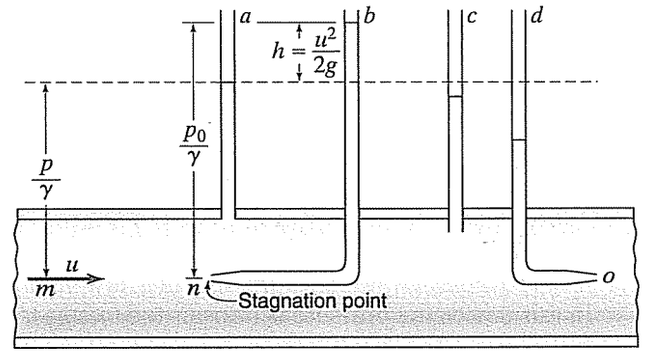


Figure 11.3

11.2 MEASUREMENT OF STATIC PRESSURE

We first defined the static pressure at a point in a flowing fluid in Secs. 5.4 and 5.11; it is the pressure in the fluid unchanged by the measuring instrument. To get an accurate measurement of static pressure in a flowing fluid, it is important that the measuring device fit the streamlines perfectly so as to create no disturbance to the flow. In a straight reach of conduit, the static pressure is ordinarily measured by attaching to the piezometer a pressure gage or a U-tube manometer, as described in Sec. 3.5. The piezometer opening in the side of the conduit should be normal to and flush with the surface. Any projection, such as (*c*) in Fig. 11.3, will result in error. It has been found, for example, that a projection of 0.10 in (2.5 mm) will cause a 16% change in the local velocity head. In this case the recorded pressure is depressed below the pressure in the undisturbed fluid because the disturbance of the streamline pattern increases the velocity, hence decreasing the pressure according to the Bernoulli equation.

When measuring the static pressure in a pipe, it is desirable to have two or more openings around the periphery of the section to account for possible imperfections of the wall. For this purpose a ***piezometer ring*** (Fig. 11.4) is used.

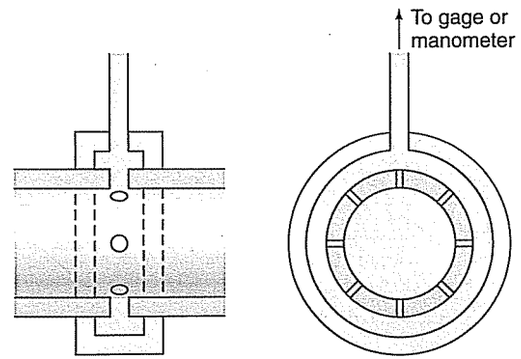


Figure 11.4
Piezometer ring.

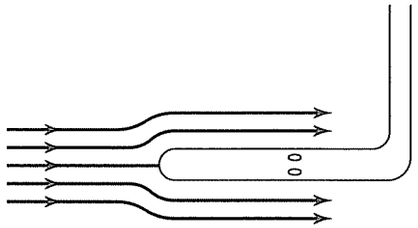


Figure 11.5
Static tube.

To measure the static pressure in a flow field, we use a ***static tube*** (Fig. 11.5). In this device the pressure is transmitted to a gage or manometer through piezometric holes that are evenly spaced around the circumference of the tube. This device will give good results if it is perfectly aligned with the flow. Actually, the mean velocity past the piezometer holes will be slightly larger than that of the undisturbed flow field; hence the pressure at the holes will generally be somewhat below the pressure of the undisturbed fluid. We can minimize this error by making the diameter of the tube as small as possible. If the direction of the flow is unknown for two-dimensional flows, we can use a ***direction-finding tube*** (Fig.11.6). This device is a cylindrical tube having two piezometer holes located as shown. Each piezometer is connected to its own measuring device. We can rotate the tube until each tube shows the same reading. Then, from symmetry, we can determine the direction of flow. If the piezometer openings are located as shown, the recorded pressures will correspond very closely to those of the undisturbed flow.

To obtain a reading of the fluid pressure, we can connect a piezometer tube to a Bourdon gage (Sec. 3.5) or to a pressure transducer (Fig. 3.9). The latter is sometimes connected to a strip-chart recorder or the pressure reading may be displayed on a panel in digital form.

11.3 MEASUREMENT OF VELOCITY WITH PITOT TUBES

One means of measuring the local velocity u in a flowing fluid is the pitot tube, named after its inventor Henri Pitot (1695–1771), a French physicist who used a bent tube in 1732 to measure velocities in the River Seine. In Sec. 5.4 we saw that the pressure at the forward stagnation point of a stationary body in a flowing fluid is $p_0 = p + \rho u^2/2$, where p and u are the pressure and velocity, respectively, in the undisturbed flow upstream from the body. If $p_0 - p$ can be measured, the velocity at a point is determined by this relation. The stagnation

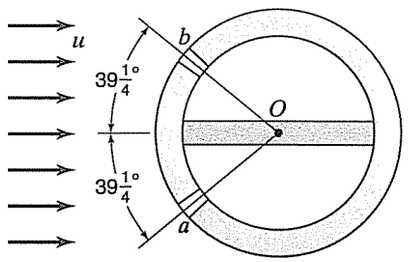


Figure 11.6
Direction-finding tube.

pressure can be measured by a tube facing upstream, such as (*b*) in Fig. 11.3. For a liquid jet or open stream with parallel streamlines, only this single tube is necessary, since the height h to which the liquid rises in the tube above the surrounding free surface is equal to the velocity head in the stream approaching the tip of the tube.

For a closed conduit under pressure we must measure the static pressure also, as shown by tube (*a*) in Fig. 11.3, and subtract this from the total pitot reading to secure the differential head h. We can measure the differential pressure with any suitable manometer arrangement. We can derive the formula for the pitot tube for incompressible flow by writing the (ideal) energy equation between points m and n of Fig. 11.3,

$$\frac{p}{\gamma} + \frac{u^2}{2g} = \frac{p_0}{\gamma} \tag{11.5}$$

from which

$$u_i = \sqrt{2g\left(\frac{p_0}{\gamma} - \frac{p}{\gamma}\right)}$$

This equation gives the ideal velocity of flow[3] u_i at the point in the stream where the pitot tube is located. In actuality, directional velocity fluctuations due to turbulence increase pitot-tube readings, so that we must multiply the right side of Eq. (11.6) by a factor C varying from 0.98 to 0.995 to give the true velocity,

$$u = Cu_i = C\sqrt{2g\left(\frac{p_0}{\gamma} - \frac{p}{\gamma}\right)} \tag{11.6}$$

Where conditions are such that it is impractical to measure static pressure at the wall, we can use a combined ***pitot-static tube,*** as in Fig. 11.7. We measure the static pressure through two or more holes drilled through an outer tube into an annular space. Rarely are the piezometer holes located in precisely the correct position to indicate the true value of p/γ. So we modify Eq. (11.6) into:

$$u = C_I\sqrt{2g\left(\frac{p_0}{\gamma} - \frac{p}{\gamma}\right)} \tag{11.7}$$

[3] Equations (11.5)–(11.7) as well as those presented in Secs. 11.6–11.9 apply strictly to incompressible fluids. However, these equations will all give very good results when applied to compressible fluids if $\mathbf{M} < 0.1$. At high values of $\mathbf{M}$ we must consider the effects of compressibility as we will in Sec. 11.10. See also Secs. 7.4, 13.2, and 13.3.

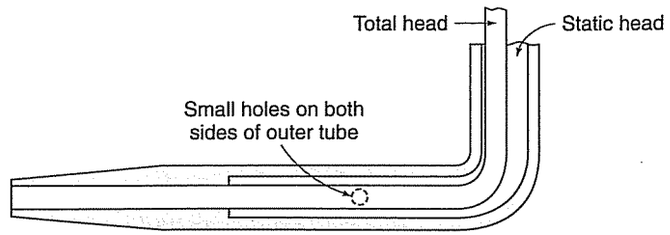


Figure 11.7
Pitot-static tube.

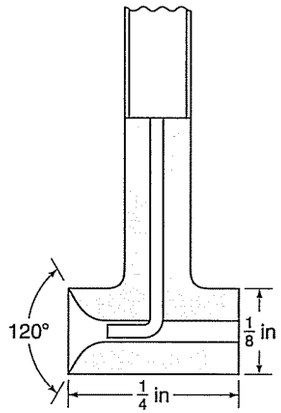


Figure 11.8
Kiel probe.

where we introduce C_I, a coefficient of instrument, to account for this discrepancy. We can use either BG or SI units with this equation, since C_I is dimensionless. However, when a coefficient possesses dimensions [see Eq. (11.27), for example], we must modify an equation developed for BG units for application to SI units, and vice versa. A particular type of pitot-static tube with a blunt nose, the ***Prandtl tube,*** is designed so that $C_I = 1$. For other pitot-static tubes, we must determine the coefficient C_I by calibration in the laboratory.

Another instrument, the ***pitometer,*** consists of two tubes, one pointing upstream and the other downstream, such as tubes (b) and (d) of Fig. 11.3. The reading for tube (d) will be considerably below the level of the static head. The equation applicable to a pitometer is identical to Eq. (11.7), except that p/γ is replaced by the pressure head sensed by the downstream tube.

Most of these devices will give reasonably accurate results even if the tube is as much as $\pm 15°$ out of alignment with the direction of flow.

Still greater insensitivity to angularity is possible if we guide the flow past the pitot tube by means of a shroud, as shown in Fig. 11.8. Such an arrangement, called a ***Kiel probe,*** is used extensively in aeronautics. The stagnation-pressure measurement with this device is accurate to within 1% of the dynamic pressure for yaw angles up to $\pm 54°$. A disadvantage is that the static pressure must be measured independently.

We can use the direction-finding tube (Fig. 11.6) to determine velocity. The procedure is to orient it properly so that both piezometers give the same reading. This reading is the static head. Then turn the tube through $39\frac{1}{4}°$ to obtain the stagnation pressure head. The difference in the two readings is the velocity head. This device has been used extensively in wind tunnels and in investigations of hydraulic machinery.

Specialized, very small pitot tubes are used in experimental work to measure velocities very close to a boundary. The ***Stanton tube*** (Fig. 11.9a) uses the wall to form one side of the tube, and we can use it only within the viscous sublayer of turbulent flows (Fig. 8.8) or with laminar flow. The ***Preston tube*** (Fig. 11.9b) is designed for use in the transition zone (see Fig. 8.8), not submerged in the viscous sublayer, and we can use it for turbulent flow over smooth

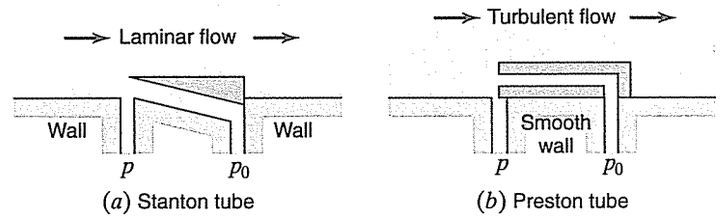


Figure 11.9
Wall pitot tubes.

surfaces. Engineers use both to collect data for the determination of the boundary shear stress τ_0, which has been found to depend on $p_0 - p$. We calculate the boundary shear stress by substituting the measured velocities into the theory of velocity profiles in circular pipes. We can assume that this theory holds for flat boundaries as well as curved boundaries in the region very close to the boundary.

SAMPLE PROBLEM 11.1 Air at 20°C is flowing through the pipe shown in Fig. S11.1, resulting in pressure gage readings of 70.2 kPa at A and 71.1 kPa at B. Atmospheric pressure is 684 mmHg. (a) Find the air velocity u. (b) What is the largest pressure difference (kPa) between the two gages for which compressibility effects can be safely neglected?

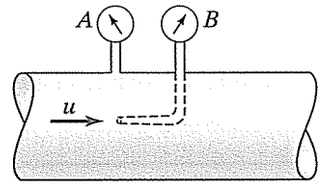


Figure S11.1

Solution

Table A.5 for air: $R = 287\ \text{m}^2/(\text{s}^2 \cdot \text{K})$

(a) $p_{\text{abs}} = p_{\text{at}} + p_{\text{gage}} = 101.32(684/760) + 70.2 = 161.4\ \text{kPa}$

Eq. (2.5): $$\gamma = \frac{gp}{RT} = \frac{9.81(161.4)}{287(273 + 20)} = 0.01883\ \text{kN/m}^3$$

Eq. (11.6): $$u = \sqrt{2(9.81)\left(\frac{71.1 - 70.2}{0.01883}\right)} = 30.6\ \text{m/s} \qquad \textit{ANS}$$

(b) Sec. 11.3, footnote 3: We need $\mathbf{M} < 0.1$

Sec. 5.4: $\mathbf{M} = V/c$; at 20°C, $c = 345$ m/s [or use Eq. (13.15)].

So we need $V < 0.1c = u_{\text{max}} = 34.5$ m/s

From Eq. (11.5): $$\Delta p = p_0 - p = \gamma \frac{u^2}{2g}$$

So $$\Delta p_{\max} = \frac{\gamma(u_{\max})^2}{2g} = \frac{0.01883(34.5)^2}{2(9.81)} = 1.142 \text{ kPa} \qquad \textit{ANS}$$

Note: This $\Delta p_{\max}$ compares with the observed Δp of $71.1 - 70.2 = 0.9$ kPa. Therefore it was safe to neglect compressibility effects in part (a).

EXERCISES

11.3.1 In Fig. S11.1 kerosene ($s = 0.81$) is flowing. The pressure gages at A and B read 70 and 150 N/m^2. Find the velocity u, assuming $C_I = 1.0$.

11.3.2 A pitot tube and a wall piezometer tube are installed in a pipe carrying 60°F water. The tubes are connected to a water-mercury manometer, which registers a differential of 3.4 in. Given that $C_I = 0.98$, find the flow velocity approaching the tube.

11.3.3 A pitot tube and a wall piezometer tube are installed in a pipe carrying 15°C water. The tubes are connected to a water-mercury manometer, which registers a differential of 78 mm. Given that $C_I = 0.97$, find the flow velocity approaching the tube.

11.3.4 The fluids of Exer. 11.3.2 are reversed so that mercury ($s = 13.56$) is flowing in the pipe and water is the gage fluid (with the manometer now inverted). With the same gage differential, find the flow velocity of the mercury.

11.3.5 The fluids of Exer. 11.3.3 are reversed so that mercury ($s = 13.56$) is flowing in the pipe and water is the gage fluid (with the manometer now inverted). With the same gage differential, find the flow velocity of the mercury.

11.3.6 A pitot-static tube ($C_I = 0.985$) is connected to an inverted U tube containing oil ($s = 0.875$). Find the velocity of flowing water if the manometer reading is 3.6 in.

11.3.7 A pitot-static tube ($C_I = 0.992$) is connected to an inverted U tube containing oil ($s = 0.91$). Find the velocity of flowing water if the manometer reading is 127 mm.

11.4 MEASUREMENT OF VELOCITY BY OTHER METHODS

Other methods for measuring local velocity will be discussed in this section.

Current Meter and Rotating Anemometer

These two instruments, which are the same in principle, determine the velocity as a function of the speed at which a series of cups or vanes rotate about an axis either parallel or normal to the flow. The instrument used in water is called a ***current meter,*** and when designed for use in air, we call it an ***anemometer.*** As the force exerted depends on the density of the fluid as well as on its velocity, the anemometer must be so made as to operate with less friction than the current meter.

If the meter is made with cups that move in a circular path about an axis perpendicular to the flow, it always rotates in the same direction and at the same rate regardless of the direction of the velocity, whether positive or negative, and it even rotates when the velocity is at right angles to its plane of rotation. Thus this type is not suitable where there are eddies or other irregularities in the flow. If the meter is constructed of vanes rotating about an axis parallel to the flow, like a propeller, it will register the component of velocity along its axis, especially if it is surrounded by a shielding cylinder. It will rotate in an opposite direction for negative flow, and is therefore a more dependable type of meter.

Hot-Wire and Hot-Film Anemometer

The ***hot-wire anemometer*** measures the instantaneous velocity at a point. It consists of a small sensing element that is placed in the flow field at the point where the velocity is to be measured. The sensing element is a short thin wire, which is generally of plantinum or tungsten, connected to a suitable electronic circuit. The operation depends on the fact that the electrical resistance of a wire is a function of its temperature; that the temperature, in turn, depends on the heat transfer to the surrounding fluid; and that the rate of heat transfer increases with increasing velocity of flow past the wire.

In one type of hot-wire anemometer the wire is maintained at a constant temperature by a variable voltage, which changes the current through the wire. Thus, when an increase in velocity tends to cool the wire, a balancing device creates an increase in voltage to increase the current through the wire. This tends to heat up the wire to counteract the cooling and thus maintain it at constant temperature. The voltage provides a measure of the velocity of the fluid. The hot-wire anemometer is a very sensitive instrument particularly adapted to the measurement of turbulent velocity fluctuations as in Fig. 4.6. A ***hot-film anemometer,*** though similar to the hot-wire, is more rugged in that its sensing element consists of a metal film laid over a glass rod and provided with a protecting coating.

Float Measurements

A crude technique for estimating the average velocity of flow in a river or stream is to observe the velocity at which a float will travel down a stream. To get good results the reach of stream should be straight and uniform with a minimum of surface disturbances. The average velocity of flow V will generally be about (0.85 ± 0.05) times the float velocity.

Photographic and Optical Methods

The camera is one of the most valuable tools in a fluid-mechanics research laboratory. In studying the motion of water, for example, researchers can introduce a series of small spheres consisting of a mixture of benzene and carbon tetrachloride adjusted to the same specific gravity as the water into the flow through suitable nozzles. When illuminated from the direction of the camera, these spheres will stand out in a picture. If successive exposures are taken on the

same film, the velocities and accelerations of the particles can be determined. A similar technique involves the use of hydrogen bubbles generated through use of a fine wire which serves as the negative electrode of a dc electric circuit. By pulsing the voltage across the wire, the water is electrolyzed, thus releasing hydrogen bubbles. Short uninsulated sections of wire will permit the bubbles to be emitted at fixed points along the wire. This, when combined with intermittent pulsing, will aid in flow visualization.

More recently developed instruments employ lasers. ***Laser Doppler velocimetry*** (LDV), also called ***laser Doppler anemometry*** (LDA), directs laser light into a fixed, very small zone, practically a point, within moving fluid. When small particles (about micron sized) or bubbles moving with the flow pass through the measuring zone, the LDV measures the Doppler shift of light they scatter. This enables accurate determination of the flow velocity, or its three components, without disturbing the flow. If there are sufficient particles, a practically continuous time history is achievable.

Somewhat similar, ***particle image velocimetry*** (PIV) uses laser light emitted in rapid pulses to illuminate a plane or volume through which flow seeded with neutral-density, micron-sized particles or bubbles passes. Photographs or images of the successive positions of the particles can provide good visualization of the flow field, and enable the calculation of the particle (fluid) velocity vectors. PIV has been used for single-phase and two-phase flows. It is accurate and does not disturb the flow, but because three-dimensional applications are complex, they are still in early stages of development. The main advantage of PIV over LDV is that a much larger region can be measured.

In the study of compressible fluids many techniques have been devised to measure optically the variations in density, as given by the ***interferometer,*** or the rate at which density changes in space, as determined in the ***shadowgraph*** and ***schlieren*** methods (Fig. 13.9).[4] From such measurements of density and density gradient, it is possible to locate shock waves. Although of great importance, these photographic methods are too complex to warrant further description here.

Other Methods

Other devices for measuring velocity of flow include magnetic flowmeters and acoustic flowmeters. ***Magnetic flowmeters*** are used to measure velocity of flow in liquids. The liquid serves as a conductor and develops a voltage as it travels through a magnetic field. With proper calibration, this device can be used to measure the average velocity in pipes. Small magnetic flowmeters can be used to measure local velocities in a flowing liquid. However, their accuracy drops off in the vicinity of boundaries.

Acoustic flowmeters depend on the effect of the moving fluid on sound waves. These devices are expensive, and are used primarily for research. One of their advantages is that they can be used without disturbing the flow.

[4] For an excellent discussion of optical methods used in the study of fluid flow, see Sec. 11.6 of Ref. 75 in Appendix E.

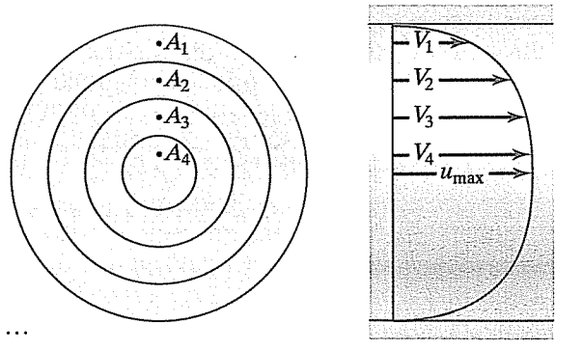


Figure 11.10
Determination of pipe discharge.

$$Q = \sum A_i V_i$$
$$= A_1V_1 + A_2V_2 + \cdots$$

11.5 Measurement of Discharge

There are various ways of measuring discharge. In a pipe, for example, we can determine the velocity at various radii using a pitot-static tube or a pitot tube in combination with a wall piezometer. We can then consider the cross section of a pipe as a series of concentric rings, each with a known velocity. The flow through these rings is summed up, as in Fig. 11.10, to determine the total flow rate.

To determine the flow in a river or stream, we use a similar technique. We divide the stream into a number of convenient sections, as in Fig. 11.11, and measure the average velocity in each section. A pitot tube could be used for such measurements, but a current meter is more commonly used. It has been found that the average velocity occurs at about $0.6 \times$ depth (Sec. 10.4), so the velocity is generally measured at that level. As a result, in Fig. 11.11

$$Q_{2\text{-}3} = \left(\frac{y_2 + y_3}{2}\right) b_{2\text{-}3} \frac{V_2 + V_3}{2}$$

and $Q_{\text{total}} = \sum Q$. Another widely used method is to replace the velocity at $0.6 \times$ depth with the average of the velocities at $0.2 \times$ depth and $0.8 \times$ depth. A crude estimate of the flow in a river or stream can be made by multiplying ($0.85 \times$ float velocity) times the area of the average cross section in the reach of stream over which the float measurement was made.

Devices for the direct measurement of discharge can be divided into two categories, those that measure by weight or positive displacement a certain quantity of fluid and those that employ some aspect of fluid mechanics. An example of the first type of device is the household water meter in which a nutating disk

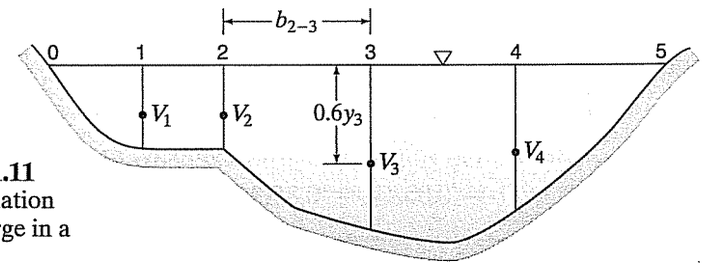


Figure 11.11
Determination of discharge in a stream.

oscillates in a chamber. On each oscillation a known quantity of water passes through the meter. We will discuss the second type of flow-measuring device, which depends on basic principles of fluid mechanics combined with empirical data, in Secs. 11.6–11.14.

SAMPLE PROBLEM 11.2 Measurements made on the cross section of Fig. 11.11 are:

Position	y, ft	b, ft	Velocity V, fps, at Surface	0.2y	0.6y	0.8y
0	—		—	—	—	—
		4.09				
1	3.60		—	2.88	2.80	1.50
		3.53				
2	3.44		—	3.15	3.02	2.50
		6.55				
3	7.20		4.05	4.18	3.48	3.10
		6.45				
4	6.72		—	3.40	3.15	2.85
		6.58				
5	—		—	—	—	—

Calculate the total discharge in the stream by the three different methods described.

Solution
Using $Q = AV$:

Method: Position	$A = b\bar{y}$ ft^2	(1) $V_{0.6}$ fps	(1) $Q_{0.6}$ cfs	(2) $\frac{V_{0.2} + V_{0.8}}{2}$ fps	(2) $\overline{V}_{0.2,0.8}$ fps	(2) $Q_{0.2,0.8}$ cfs	
0				—			
	7.36	1.40	10.31		1.10	8.06	
1				2.19			
	12.43	2.91	36.16		2.51	31.16	
2				2.83			
	34.85	3.25	113.25		3.23	112.64	
3				3.64			
	43.28	3.32	143.47		3.38	146.39	
4				3.13			
	20.46	1.58	32.23		1.56	31.97	
5				—			
Total	118.38		335.4	***ANS***		330.2	***ANS***

(3) Assume float velocity = surface velocity at position 3:

$$Q = 0.85AV_s = 0.85(118.38)4.05 = 407.5 \text{ cfs} \quad \textbf{\textit{ANS}}$$

11.6 ORIFICES, NOZZLES, AND TUBES

Among the devices used for the measurement of discharge are orifices and nozzles. Tubes are rarely used in this way but we include them here because their theory is the same and experiments on tubes provide information about entrance losses from reservoirs into pipelines. An ***orifice*** is an opening (usually circular) in the wall of a tank or in a plate normal to the axis of a pipe, the plate being either at the end of the pipe or in some intermediate location. An orifice is characterized by the fact that the thickness of the wall or plate is very small relative to the size of the opening. A ***standard orifice*** is one with a sharp edge as in Fig. 11.12*a* or an absolutely square shoulder as in Fig. 11.12*b*, so that there is only a line contact with the fluid. Those shown in Figs. 11.12*c* and *d* are not standard, because the flow through them is affected by the thickness of the plate, the roughness of the surface, and, for *d*, the radius of curvature. Hence such orifices should be calibrated if high accuracy is desired.

A ***nozzle*** is a tube of changing diameter, usually converging as in Fig. 11.13 if it is used for liquids; but for a gas or a vapor a nozzle may first converge and then diverge (Sec. 13.9) to produce supersonic flow. In addition to possible use as a flow measuring device a nozzle has other important uses, such as providing a high-velocity stream for fire fighting or for power in a steam turbine or an impulse turbine (Pelton wheel, see Sec. 16.2). The ***base*** of a nozzle is the point (or strictly the plane, perpendicular to its centerline) at which the diameter starts to change from that of the supporting pipe. It is the nozzle entry point, and usually the point of attachment to the supporting pipe.

A ***tube*** is a short pipe whose length is not more than two or three diameters. There is no sharp distinction between a tube and the thick-walled orifices of Figs. 11.12*c* and *d*. A tube may be of uniform diameter, or it may diverge.

A ***jet*** is a stream issuing from an orifice, nozzle, or tube. It is not enclosed by solid boundary walls but is surrounded by a fluid whose velocity is less than

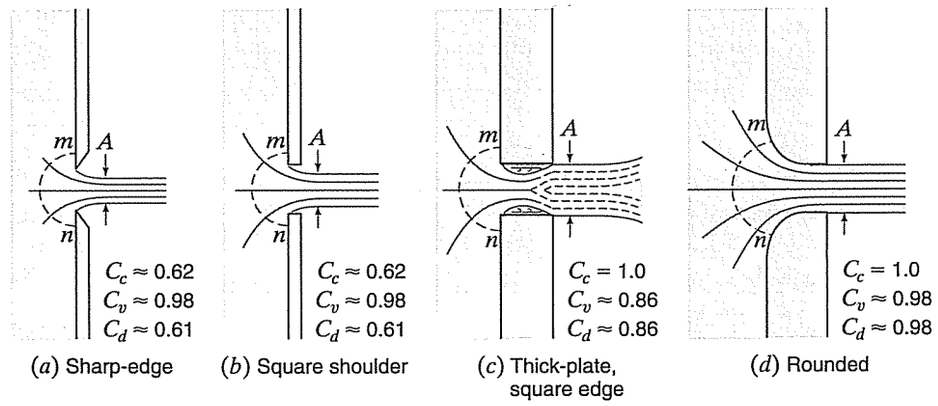


Figure 11.12
Orifices.

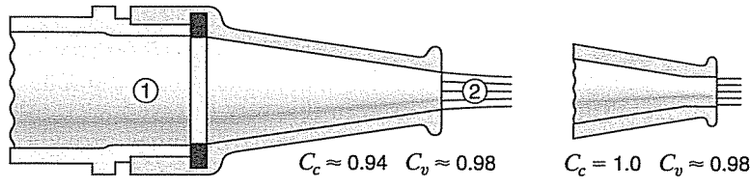


Figure 11.13
Nozzles.

its own. The two fluids may be different or they may be of the same kind. A ***free jet*** is a stream of *liquid* surrounded by a *gas* and is therefore directly under the influence of gravity. A ***submerged jet*** is a stream of any fluid surrounded by a fluid of the same type, that is, a gas jet discharging into a gas or a liquid jet discharging into a liquid. A submerged jet is buoyed up by the surrounding fluid and is not directly under the action of gravity.

Jet Contraction

Where the streamlines converge in approaching an orifice, as shown in Fig. 11.14, they continue to converge beyond the upstream section of the orifice until they reach the section *xy*, where they become parallel. Commonly this section is about $0.5D_o$ from the upstream edge of the opening, where D_o is the diameter of the orifice. The section *xy* is then a section of minimum area, and is called the ***vena contracta.*** Beyond the vena contracta the streamlines commonly diverge because of

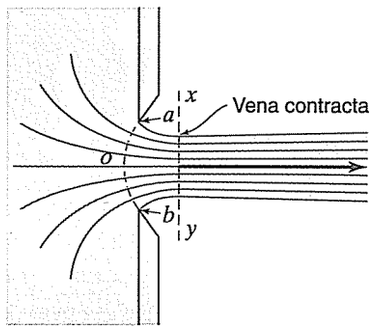


Figure 11.14
Jet contraction.

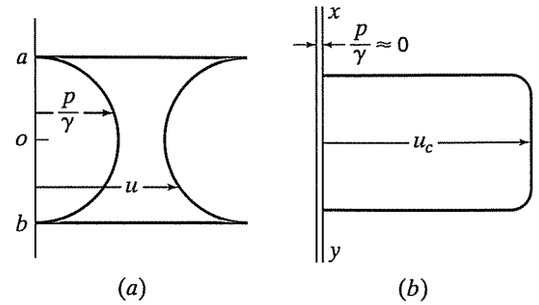


Figure 11.15
Pressure and velocity variations in a jet. (*a*) At section *aob* of Fig. 11.14. (*b*) At vena contracta (section *xy*) in Fig. 11.14.

frictional effects.[5] In Fig. 11.12*c* the minimum section is referred to as a ***submerged vena contracta,*** since it is surrounded by its own fluid. In Fig. 11.12*d* there is no vena contracta as the rounded entry to the opening permits the streamlines to gradually converge to the cross-sectional area of the orifice.

Jet Velocity and Pressure

Jet velocity is defined as the average velocity at the vena contracta in Figs. 11.12*a* and *b*, and at the downstream edge of the orifices in Figs. 11.12*c* and *d*. The velocity at these sections is practically constant across the section except for a small annular region around the outside (Fig. 11.15*b*). In all four of the jets of Fig. 11.12 the pressure is practically constant across the diameter of the jet wherever the streamlines are parallel, and this pressure must be equal to that in the medium surrounding the jet at that section. At sections *mn* in Fig. 11.12 where the streamlines are curved, the effective cross-sectional area of the flow (at right angles to streamlines) is greater than at the minimum section, and hence the average velocities at sections *mn* are considerably less than the jet velocities. The same is true of section *aob* of Fig. 11.14. In Fig. 11.15*a* the velocity and pressure distributions at section *aob* of Fig. 11.14 are shown. These variations are the result of the curvature of the streamlines and centrifugal effects (Sec. 5.16).

Coefficient of Contraction C_c

The ratio of the area A of a jet (Fig. 11.12), to the area A_o of the orifice or other opening, is called the ***coefficient of contraction.*** Thus $A = C_c A_o$.

[5] Of course, if a free jet is discharged vertically downward, the acceleration due to gravity will cause its velocity to increase and the area to decrease continuously, so that there may be no apparent section of minimum area. In such special cases, we should take the vena contracta as the place where marked contraction ceases and before the place where gravity has increased the velocity to any appreciable extent above the true jet velocity.

Coefficient of Velocity C_v

The velocity that would be attained in the jet if friction did not exist may be termed the ideal velocity V_i.[6] It is practically the value of u_c in Fig. 11.15. Because of friction, the actual average velocity V is less than the ideal velocity, and the ratio V/V_i is called the ***coefficient of velocity.*** Thus $V = C_v V_i$.

Coefficient of Discharge C_d

The ratio of the actual rate of discharge Q to the ideal rate of discharge Q_i (the flow that would occur if there were no friction and no contraction) is defined as the coefficient of discharge. Thus $Q = C_d Q_i$. By observing that $Q = AV$ and $Q_i = A_o V_i$, it is seen that $C_d = C_c C_v$.

Determining the Coefficients

The coefficient of contraction can be determined by using outside calipers to measure the jet diameter at the vena contracta and then comparing the jet area with the orifice area. The contraction coefficient is very sensitive to small variations in the edge of the orifice or in the upstream face of the plate. Thus slightly rounding the edge of the orifice in Fig. 11.12*b* or roughening the orifice plate will increase the contraction coefficient materially.

The average velocity V of a free jet may be determined by a velocity traverse of the jet with a fine pitot tube or it may be obtained by measuring the flow rate and dividing by the cross-sectional area of the jet. We can also compute the velocity approximately from the coordinates of the trajectory of the jet, as discussed in Sec. 5.15. The ideal velocity V_i is computed by the Bernoulli theorem. So we can compute C_v for an orifice, nozzle, or tube by dividing V by V_i.

The coefficient of discharge is the one that can most readily be obtained and with a high degree of accuracy. It is also the one that is of the most practical value. For a liquid, we can determine the actual Q by some standard method such as a volume or a weight measurement over a known time. For a gas we can note the change in pressure and temperature in a container of known volume from which the gas may flow. Obviously, if we measure any two of the coefficients, we can compute the third from them. Thus, in equation form,

Ideal flow rate:
$$Q_i = A_i V_i = A_o \sqrt{2g(\Delta H)} \tag{11.8}$$

Actual flow rate:
$$Q = AV = C_c A_o (C_v \sqrt{2g(\Delta H)}) \tag{11.9}$$

or
$$Q = C_d Q_i = C_d A_o \sqrt{2g(\Delta H)} \tag{11.10}$$

and
$$C_d = \frac{Q}{Q_i} = C_c C_v \tag{11.11}$$

[6] We frequently call this the ***theoretical velocity,*** but this may be a misuse of the word "theoretical." Any correct theory should allow for the fact that friction exists and affects the result. Otherwise it is not correct theory but merely an incorrect hypothesis.

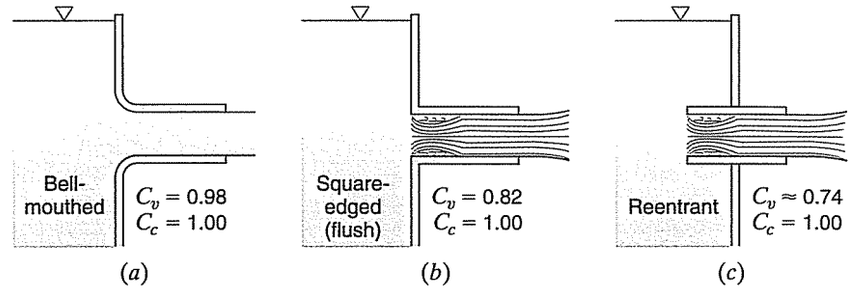


Figure 11.16
Coefficients for tubes.

where ΔH is the total difference in energy head between the upstream section and the minimum section of the jet (section A of Fig. 11.12). Recall that the total energy head $H = z + p/\gamma + V^2/2g$. If the flow is from a tank, the velocity of approach is negligible and can be neglected. If the discharge is to the atmosphere (free jet), the downstream pressure head is zero, whereas if the jet is submerged, the downstream pressure head is equal to the depth of submergence[7] (Fig. 11.18) in the case of a liquid or to the pressure head surrounding the jet in the case of a gas.

Typical values of the coefficients for orifices, nozzles, and tubes are as indicated in Figs. 11.12,[8] 11.13, and 11.16, respectively. It is apparent from Fig. 11.16 that rounding the entrance to a tube increases the coefficient of velocity. Any device that provides a uniform diameter for a long enough distance before exit, such as the tubes of Fig. 11.16 or the nozzle tip of Fig. 11.13*b*, will usually create a $C_c = 1.0$. Although this increases the size of the jet from the given area, it also tends to produce more friction.

If the geometry of the orifice, nozzle or tube is standard such as those of Figs. 11.12, 11.13, and 11.16, the coefficients should be very close to the values indicated on the figures. However, the best way to determine the coefficients of a device, particularly one of unusual shape, is by experiment in the laboratory. Also, we can make a fair estimate of the contraction by sketching the flow net. If we wish to estimate the coefficient of discharge of an orifice, nozzle, or tube it is usually best to estimate velocity and contraction coefficients separately and calculate the discharge coefficient from them.

Borda Tube

Tubes (*b*) and (*c*) in Fig. 11.16 are shown as flowing full, and because of the turbulence, the jets issuing from them will have a "broomy" appearance. Because

[7] If the jet discharges into a different liquid, we must convert the depth of submergence to an equivalent depth of the *flowing liquid.*

[8] Surface tension can become important when orifices operate under low heads. The coefficient of contraction of small, sharp-edged, and square-edged orifices such as those of Figs. 11.12*a* and *b* have values of C_c as high as 0.72 rather than the usual 0.62 when operating under heads less than about 0.5 ft.

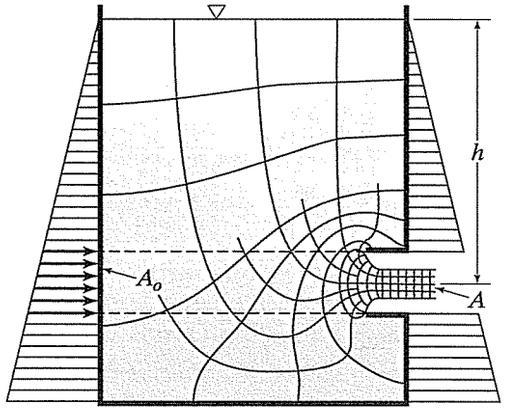


Figure 11.17
Borda tube.

of the contraction of the jet at entrance to these tubes the local velocity in the central portion of the stream will be higher than that at exit from the tubes, and hence the pressure will be lower. If we lower the pressure to that of the vapor pressure of the liquid, the streamlines will then no longer follow the walls, and the jet springs clear. In such a case tube (*b*) of Fig. 11.16 becomes equivalent to orifice (*b*) in Fig. 11.12, while tube (*c*) of Fig. 11.16 behaves as shown in Fig. 11.17. If its length is less than its diameter, the reentrant tube is called a ***Borda mouthpiece.*** Because of the greater curvature of the streamlines for a reentrant tube, if the tube flows full the velocity coefficient is lower (Fig. 11.16*c*) than for any other type of entry. But if the jet springs clear as in Fig. 11.17, the velocity coefficient is as high as for a sharp-edged orifice.

The Borda mouthpiece is of interest because it is one device for which the contraction coefficient can be very simply calculated. For all other orifices and tubes there is a reduction of the pressure on the walls adjacent to the opening, but the exact pressure values are unknown. But for the reentrant tube the velocity along the wall of the tank is almost zero at all points, and hence the pressure is essentially hydrostatic. In the case of a Borda tube with the jet springing clear, the only unbalanced pressure is that on an equal area A_o opposite to the tube (Fig. 11.17), and the force due to this pressure is $\gamma h A_o$. The time rate of change of momentum due to the flow out of the tube is $\rho Q V = \gamma A V^2/g$, where A is the area of the jet. Equating force to time rate of change of momentum, $\gamma h A_o = \gamma A V^2/g$, and thus $V^2 = gh A_o/A$. Ideally, $V^2 = 2gh$, and thus, ideally, $C_c = A/A_o = 0.5$. The actual values of the coefficients for a Borda tube with the jet springing clear are $C_c = 0.52$, $C_v = 0.98$, and $C_d = 0.51$.

Head Loss

We can find the relationship between the head loss and the coefficient of velocity of an orifice, nozzle, or tube by comparing the ideal energy equation with the actual (or real) energy equation between points 1 and 2 in Fig. 11.13*a*. The ideal energy equation (5.29) is

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}$$

In the case of a free jet $p_2 = 0$, while for the most general case of a submerged jet $p_2 \neq 0$. From continuity, $A_1 V_1 = A_2 V_2$; hence we can write

$$\frac{p_1}{\gamma} + z_1 + \left(\frac{A_2}{A_1}\right)^2 \frac{V_2^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}$$

which leads to

$$(V_2)_{\text{ideal}} = \frac{1}{\sqrt{1 - (A_2/A_1)^2}} \sqrt{2g\left[\left(\frac{p_1}{\gamma} + z_1\right) - \left(\frac{p_2}{\gamma} + z_2\right)\right]} \qquad (11.12)$$

The real energy equation (5.28) accounts for head loss and is expressed as

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} - h_{L_{1-2}} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}$$

which leads to the actual velocity

$$V_2 = \frac{1}{\sqrt{1 - (A_2/A_1)^2}} \sqrt{2g\left[\left(\frac{p_1}{\gamma} + z_1\right) - \left(\frac{p_2}{\gamma} + z_2\right) - h_{L_{1-2}}\right]} \qquad (11.13)$$

Recalling that the actual $V = C_v V_{\text{ideal}}$, and combining this with the above expressions for V_{ideal} and actual V gives

$$h_{L_{1-2}} = \left(\frac{1}{C_v^2} - 1\right)\left[1 - \left(\frac{A_2}{A_1}\right)^2\right]\frac{V_2^2}{2g} \qquad (11.14)$$

where V_2 is an actual velocity. This equation is perfectly general; it expresses the head loss between a section upstream of an orifice and the jet (section A in Fig. 11.12) or between sections 1 and 2 in Fig. 11.13*a*, etc. If the orifice or nozzle takes off directly from a tank where $A_1 \gg A_2$, the velocity of approach is negligible and Eq. (11.14) reduces to

$$h_{L_{1-2}} = \left(\frac{1}{C_v^2} - 1\right)\frac{V_2^2}{2g} \qquad (11.15)$$

Note that for the tubes of Fig. 11.16 with $C_v = 0.98$, 0.82, and 0.74, Eq. (11.15) yields $h_L = 0.04V_2^2/2g$, $0.5V_2^2/2g$, and $0.8V_2^2/2g$, respectively. These correspond to the values for minor loss at entrance shown in Fig. 8.13.

Submerged Jet

For the case of a submerged jet, as in Fig. 11.18, the ideal energy equation is written between 1 and 2, realizing that the pressure head on the jet at 2 is equal to h_3. Thus

$$h_1 = h_3 + \frac{V_i^2}{2g}$$

or

$$V_i = \sqrt{2g(h_1 - h_3)} = \sqrt{2g(\Delta H)}$$

where V_i is the ideal velocity at the vena contracta of the submerged jet and ΔH is the net head differential expressed in terms of the flowing liquid. Hence $Q = C_c C_v A_o \sqrt{2g(\Delta H)}$ as in Eq. (11.9).

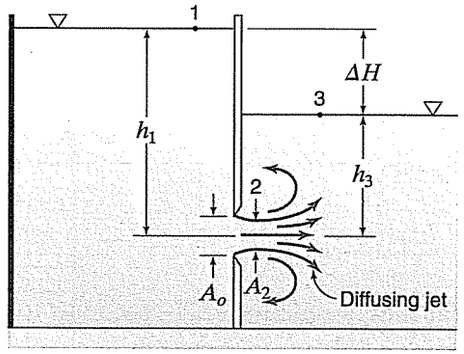


Figure 11.18
Submerged jet.

For a submerged orifice, nozzle, or tube the coefficients are practically the same as for a free jet, except that, for heads less than 10 ft (3 m) and for very small openings, the discharge coefficient may be slightly less. It is of interest to observe that, if the energy equation is written between 1 and 3, the result is $h_{L_{1-3}} = h_1 - h_3 = \Delta H$. Actually, the head loss in this case is that of Eq. (11.15) plus that of a submerged discharge, as described in Sec. 8.22. Hence, for submerged orifices, nozzles, and tubes,

$$h_{L_{1-3}} = \left(\frac{1}{C_v^2} - 1\right)\frac{V_2^2}{2g} + \frac{V_2^2}{2g} = \frac{1}{C_v^2}\frac{V_2^2}{2g} = \frac{V_i^2}{2g} = \Delta H$$

where the actual $V_2 = C_v V_i$, the velocity at the vena contracta.

SAMPLE PROBLEM 11.3 A 2-in circular orifice (not standard) at the end of the 3-in-diameter pipe shown in Fig. S11.3 discharges into the atmosphere a measured flow of 0.60 cfs of water when the pressure in the pipe is 10.0 psi. The jet velocity is determined by a pitot tube to be 39.2 fps. Find the values of the coefficients C_v, C_c, and C_d. Find also the head loss from inlet to vena contracta.

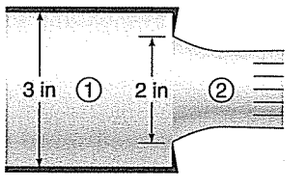


Figure S11.3

Solution
Define the inlet as section 1 and the throat as section 2.

$$\frac{p_1}{\gamma} = 10\left(\frac{144}{62.4}\right) = 23.1 \text{ ft}$$

$$V_1 = \frac{Q}{A_1} = \frac{0.60}{\pi(1.5/12)^2} = 12.22 \text{ fps}, \quad \frac{V_1^2}{2g} = 2.32 \text{ ft}$$

Express the ideal energy equation from 1 to 2 to determine the ideal velocity V_{2i} at 2

$$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} = \frac{V_{2i}^2}{2g}$$

$$\frac{V_{2i}^2}{2g} = 23.1 + 2.32 = 25.4; \quad V_{2i} = 40.4 \text{ fps}$$

$$C_v = \frac{V_2}{V_{2i}} = \frac{39.2}{40.4} = 0.969 \qquad \textbf{\textit{ANS}}$$

Area of jet $$A_2 = \frac{Q}{V_2} = \frac{0.60}{39.2} = 0.01531 \text{ ft}^2$$

$$C_c = \frac{A_2}{A_o} = \frac{0.01531}{\pi(1/12)^2} = 0.702 \qquad \textbf{\textit{ANS}}$$

Hence $$C_d = C_c C_v = 0.680 \qquad \textbf{\textit{ANS}}$$

From Eq. (11.14):

$$h_{L_{1-2}} = \left(\frac{1}{(0.969)^2} - 1\right)\left[1 - \left(\frac{2}{3}\right)^4\right]\frac{V_2^2}{2g} = 0.0517\frac{V_2^2}{2g}$$

$$= 0.0517\frac{(39.2)^2}{2(32.2)} = 1.233 \text{ ft} \qquad \textbf{\textit{ANS}}$$

Check: Calculate the actual velocity V_2 at 2 by expressing the real energy equation from 1 to 2:

$$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} - h_{L_{1-2}} = \frac{V_2^2}{2g}$$

$$23.1 + 2.32 - 1.233 = \frac{V_2^2}{2g}; \quad \text{actual } V_2 = 39.4 \text{ fps}$$

This checks well with the measured velocity of 39.2 fps.

EXERCISES

11.6.1 Water issues from a circular 5-in-diameter orifice under a head of 45 ft. If 534 ft^3 are discharged in 2 min, what is the coefficient of discharge? If the diameter at the vena contracta is measured to be 3.93 in, what is the coefficient of contraction and what is the coefficient of velocity?

11.6.2 A jet discharges 0.131 cfs from a 1.25-in-diameter orifice in a vertical plane under a head of 10 ft. The jet centerline passes through the point 11.47 ft horizontally from the vena contracta and 3.5 ft below the center of the orifice. Find the coefficients of (*a*) discharge, (*b*) velocity, and (*c*) contraction.

11.6.3 A jet discharges 5.19 L/s from a 35-mm-diameter orifice in a vertical plane under a head of 4 m. The jet centerline passes through the point 4.28 m horizontally from the vena contracta and 1.2 m below the center of the orifice. Find the coefficients of (*a*) discharge, (*b*) velocity, and (*c*) contraction.

11.6.4 A 4-in-diameter water jet discharges from a nozzle whose velocity coefficient is 0.96. The pressure in the 9-in-diameter pipe is 11 psi. Assuming the jet does not contract, what is the velocity at the tip of the nozzle? What is the flow rate?

11.6.5 In Fig. X11.6.5 the area A is twice the area A_o. If the diverging tube discharges water when $h = 6$ ft, find (*a*) the velocity at the throat; (*b*) the pressure head at the throat. Neglect all friction losses.

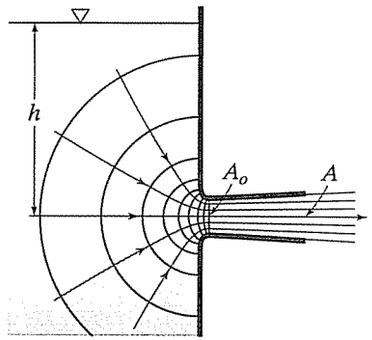


Figure X11.6.5

11.6.6 For the same data as Exer. 11.6.5, with a barometric pressure of 14.8 psia and a water temperature of 90°F, what is the maximum value of h at which the tube will flow full? What will happen if h is made greater than this?

11.6.7 For the rounded entrance and tube flowing full as in Fig. X11.6.5, $C_c = 1.0$ for both the throat and the exit, and thus $C_v = C_d$ for both sections. For the throat, assume the value of C_v as given for Fig. 11.16*a*, and assume that for the tube as a whole the discharge coefficient applied to the exit end is 0.72. Find the velocity at the throat and the pressure head at the throat if $h = 6$ ft. Compare your answer with Exer. 11.6.5, which neglected friction.

11.6.8 Given that the diverging tube shown in Fig. X11.6.5 has $A = 1.75A_0$. Find the throat velocity and pressure head when $h = 3$ m of water. Neglect all friction losses.

11.6.9 Given that the diverging tube shown in Fig. X11.6.5 has $A = 1.75A_0$. If it is operating at 1000 m elevation under standard atmospheric conditions, what will be the maximum value of h for which the tube will flow full?

11.6.10 Water flows through a 60-mm-diameter sharp-edged orifice which connects two adjacent tanks. The head on one side of the orifice is 2.5 m and 0.5 m on the other. Given $C_c = 0.62$ and $C_v = 0.95$, calculate the flow rate.

11.7 VENTURI METER

The converging tube is an efficient device for converting pressure head to velocity head, while the diverging tube converts velocity head to pressure head. The two may be combined to form a ***venturi tube,*** named after Giovanni B. Venturi (1746–1822), an Italian physicist who investigated its principle about 1791. It was applied to the measurement of water by an American engineer, Clemens Herschel, in 1886. As shown in Fig. 11.19, it consists of a tube with a constricted ***throat,*** which produces an increased velocity accompanied by a reduction in pressure, followed by a gradually diverging portion in which the velocity is transformed back into pressure with slight friction loss. As there is a definite relation between the pressure differential and the rate of flow, the tube may be made to serve as a metering device known as a ***venturi meter.*** The venturi meter is used for measuring the rate of flow of both compressible and incompressible fluids.[9] In this section we shall consider the application of the venturi meter to incompressible fluids; its application to compressible fluids will be discussed in Sec. 11.10.

Writing the Bernoulli equation between sections (1) (inlet) and (2) (throat) of Fig. 11.19, we have, for the ideal case,

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}$$

Substituting the continuity equation, $V_1 = (A_2/A_1)V_2$, we get for the ideal throat velocity

$$V_{2i} = \sqrt{\frac{1}{1 - (A_2/A_1)^2}}\sqrt{2g\left[\left(\frac{p_1}{\gamma} + z_1\right) - \left(\frac{p_2}{\gamma} + z_2\right)\right]}$$

Recalling, from Eq. (3.12*b*), that we often find it convenient to write

$$\left(\frac{p_1}{\gamma} + z_1\right) - \left(\frac{p_2}{\gamma} + z_2\right) = \Delta\left(\frac{p}{\gamma} + z\right)$$

then here we have

$$V_{2i} = \sqrt{\frac{1}{1 - (A_2/A_1)^2}}\sqrt{2g\Delta\left(\frac{p}{\gamma} + z\right)}$$

As there is some friction loss between (1) and (2), the true velocity V_2 is slightly less than the ideal value given by this expression. Therefore we introduce a discharge coefficient C, so that the flow is given by CQ_i or

$$Q = A_2V_2 = CA_2V_{2i} = \frac{CA_2}{\sqrt{1 - (D_2/D_1)^4}}\sqrt{2g\Delta\left(\frac{p}{\gamma} + z\right)} \qquad (11.16)$$

[9] As mentioned earlier, if $\mathbf{M} < 0.1$, a compressible fluid can be treated as if it were incompressible without introducing much error.

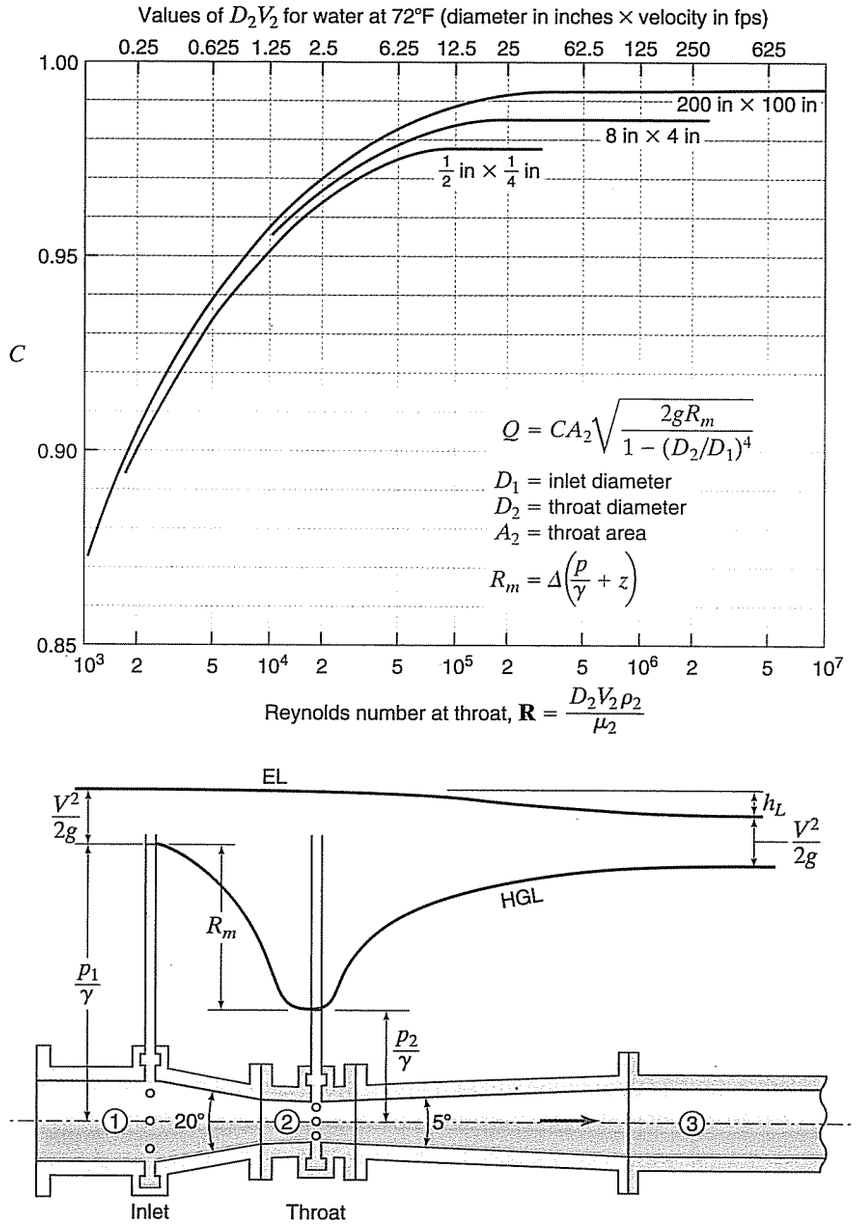


Figure 11.19
Venturi meter with conical entrance and flow coefficients for $D_2/D_1 = 0.5$.

In this situation the discharge coefficient C is identical to the velocity coefficient C_v since the coefficient of contraction $C_c = 1.0$. If we use a differential manometer with piezometric connections at sections (1) and (2), then from Fig. 3.14 and Eq. (3.12*b*) we note that

$$\Delta\left(\frac{p}{\gamma}+z\right) = R_m\left(\frac{s_M}{s_F}-1\right) \tag{11.17}$$

where R_m is the manometer reading, and s_M and s_F are the specific gravities of the manometer and flowing fluids, respectively.

The venturi tube provides an accurate means for measuring flow in pipelines. With a suitable recording device, we can integrate the flow rate so as to give the total quantity of flow. Aside from the installation cost, the only disadvantage of the venturi meter is that it introduces a permanent frictional resistance in the pipeline. Practically all this loss occurs in the diverging part between sections (2) and (3), as noted in Fig. 5.12, and is ordinarily from $0.1h$ to $0.2h$, where h is the static-head differential between the upstream section and the throat, as indicated in Fig. 11.19.

Values of D_2/D_1 may vary from $\frac{1}{4}$ to $\frac{3}{4}$, but a common ratio is $\frac{1}{2}$. A small ratio gives increased accuracy of the gage reading, but is accompanied by a higher friction loss and may produce an undesirably low pressure at the throat, sufficient in some cases to cause liberation of dissolved air or even vaporization of the liquid at this point. This phenomenon, called ***cavitation,*** has been described in Sec. 5.10. The angles of convergence and divergence indicated in Fig. 11.19 are those considered optimum, though somewhat larger angles are sometimes used to reduce the length and cost of the tube.

For accuracy in use, the venturi meter should be preceded by a straight pipe whose length is at least 5 to 10 pipe diameters. The approach section becomes more important as the diameter ratio increases, and the required length of straight pipe depends on the conditions preceding it. For example, the vortex formed from two short-radius elbows in planes at right angles is not eliminated within 30 pipe diameters (see Sec. 8.26). We can alleviate such a condition by installing straightening vanes preceding the meter. The pressure differential should be obtained from piezometer rings (Fig. 11.4) surrounding the pipe, with a number of suitable openings in the two sections. In fact, these openings are sometimes replaced by very narrow slots extending most of the way around the circumference.

Unless specific information is available for a given venturi tube, we can assume the value of C is about 0.99 for large tubes and about 0.97 or 0.98 for small ones, provided the flow is such as to give Reynolds numbers greater than about 10^5 (Fig. 11.19). A roughening of the surface of the converging section from age or scale deposit will reduce the coefficient slightly. Venturi tubes in service for many years have shown a decrease in C of the order of 1–2%. Dimensional analysis of a venturi tube indicates that the coefficient C should be a function of Reynolds number and of the geometric parameters D_1 and D_2. Values of venturi-tube coefficients are shown in Fig. 11.19. This diagram is for a diameter ratio of $D_2/D_1 = 0.5$, but is reasonably valid for smaller ratios also. For best results we should calibrate a venturi meter by conducting a series of tests in which the flow rate is measured over a wide range of Reynolds numbers.

Occasionally, the precise calibration of a venturi tube has given a value of C greater than 1. Such an abnormal result is sometimes due to improper piezometer openings. But another possible explanation is that the α's at sections (1) and (2) are such that this is really so.

SAMPLE PROBLEM 11.4 Find the discharge rate of 20°C water through the venturi tube shown in Fig. S11.4 if $D_1 = 800$ mm, $D_2 = 400$ mm, $\Delta z = 2.00$ m, and $R_m = 150$ mmHg. Assume Fig. 11.19 is applicable.

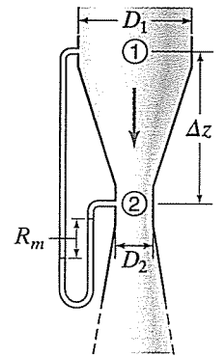


Figure S11.4

Solution

Table A.1 for water at 20°C: $\nu = 1.003 \times 10^{-6}$ m²/s

Venturi size is 800 mm × 400 mm = 31.5 in × 15.75 in.

This is about midway between the 8 in × 4 in and 200 in × 100 in curves on Fig. 11.19. So, from Fig. 11.19: Maximum $C \approx 0.988$; assume this value. Eqs. (11.16) and (11.17):

$$Q = \frac{0.988\pi(0.40/2)^2}{\sqrt{1-(400/800)^4}}\sqrt{2(9.81)0.15\left(\frac{13.55}{1}-1\right)} = 0.779 \text{ m}^3/\text{s}$$

$$V_2 = \frac{Q}{A_2} = \frac{0.779}{\pi(0.40/2)^2} = 6.20 \text{ m/s}$$

Eq. (7.6): $$\mathbf{R} = \frac{D_2V_2}{\nu} = \frac{0.40(6.20)}{1.003\times10^{-6}} = 2.47\times10^6$$

Check Fig. 11.19 for this **R**: $C = 0.988$. Thus $Q = 0.779$ m³/s ***ANS***

EXERCISES

11.7.1 The venturi meter of Fig. S11.4 has $D_1 = 8$ in, $D_2 = 4$ in, and $\Delta z = 1.5$ ft. Assume the discharge coefficients of Fig. 11.19 are applicable. Find the flow rate of 72°F water when the manometer containing carbon tetrachloride ($s = 1.59$) reads $R_m = 4$ in.

11.7.2 The venturi meter of Fig. S11.4 has $D_1 = 200$ mm, $D_2 = 100$ mm, and $\Delta z = 450$ mm. Assume the discharge coefficients of Fig. 11.19 are applicable. Find the flow rate of 20°C water when the manometer containing carbon tetrachloride ($s = 1.59$) reads $R_m = 100$ mm.

11.7.3 The venturi meter of Fig. S11.4 has $D_1 = 900$ mm, $D_2 = 450$ mm, and $\Delta z = 2.25$ m. Assume the discharge coefficients of Fig. 11.19 are applicable. Find the flow rate of 20°C water when the mercury ($s = 13.56$) manometer reads $R_m = 160$ mm.

11.7.4 The venturi meter of Fig. S11.4 has $D_1 = 4$ in, $D_2 = 2$ in, and $\Delta z = 12$ in. Assume the discharge coefficients of Fig. 11.19 are applicable. Find the reading R_m of the mercury ($s = 13.56$) manometer when oil ($s = 0.92$) with a kinematic viscosity of 0.0005 ft²/sec is flowing at a rate of 0.40 cfs.

11.8 FLOW NOZZLE

If the diverging discharge cone of a venturi tube is omitted, the result is a ***flow nozzle*** of the type shown in Fig. 11.20. This is simpler than the venturi tube and can be installed between the flanges of a pipeline. It will serve the same purpose, though at the expense of an increased frictional loss in the pipe. Although we could use venturi-meter Eq. (11.16) for the flow nozzle, it is more convenient and customary to include the correction for velocity of approach with the coefficient of discharge, so that

$$Q = KA_2\sqrt{2g\Delta\left(\frac{p}{\gamma}+z\right)} \tag{11.18}$$

where we call K the ***flow coefficient*** and A_2 the area of the nozzle throat. Comparison with Eq. (11.16) establishes the relation

$$K = \frac{C}{\sqrt{1-(D_2/D_1)^4}} \tag{11.19}$$

Although there are many designs of flow nozzles, the ISA (International Standards Association) nozzle (Fig. 11.21) has become an accepted standard form in many countries. The quoted "nozzle diameter" is the throat diameter D_2.

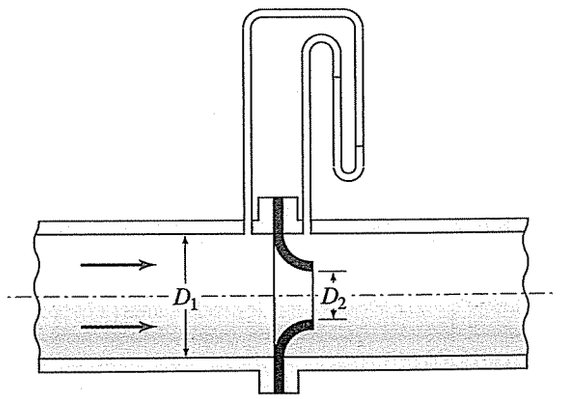


Figure 11.20
Flow nozzle.

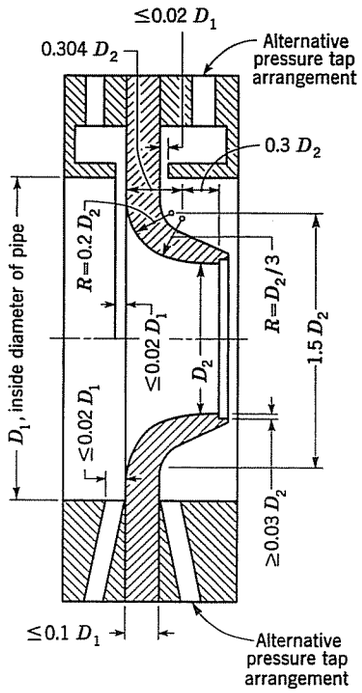


Figure 11.21
ISA flow nozzle.

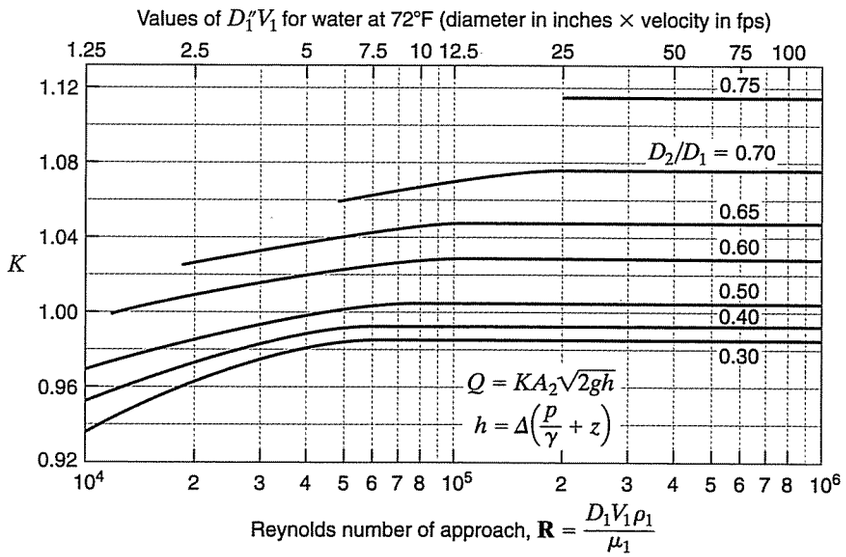


Figure 11.22
Flow coefficients for ISA nozzle. (Adapted from *ASME Flow Measurement*, 1959.)

Values of K for various diameter ratios of the ISA nozzle are given in Fig. 11.22 as a function of Reynolds number. Note that in this case the Reynolds number is computed for the approach pipe rather than for the nozzle throat, which is a convenience since **R** in the pipe is frequently needed for other computations also.

As shown in Fig. 11.22, many of the values of K are greater than unity, which results from including the correction for approach velocity with the conventional coefficient of discharge. Many attempts have been made to design a nozzle for which the velocity-of-approach correction would just compensate for the discharge coefficient, leaving a value of the flow coefficient equal to unity, principally using so-called ***long-radius nozzles.*** Usually such a coefficient of unity is approached over only a limited range.

As in the case of the venturi meter, the flow nozzle should be preceded by at least 10 diameters of straight pipe for accurate measurement. Two alternative arrangements for the pressure taps are shown in Fig. 11.21.

SAMPLE PROBLEM 11.5 A 2-in ISA flow nozzle is installed in a 3-in pipe carrying water at 72°F. If a water-air manometer shows a differential of 2 in, find the flow.

Solution

This requires a trial-and-error type of solution. First assume a reasonable value of K. From Fig. 11.22, for $D_2/D_1 = 0.67$, and for the level part of the curve, $K = 1.06$.

$$A_1 = \frac{\pi}{4}\left(\frac{3}{12}\right)^2 = 0.0491 \text{ ft}^2; \quad A_2 = \frac{\pi}{4}\left(\frac{2}{12}\right)^2 = 0.0218 \text{ ft}^2$$

This air-water manometer is like Fig. 3.14*b* with $z_B - z_A = 0$. For air and water, $s_M/s_F \approx 0.001$, so it can be neglected in Eq. (3.13), and

$$\Delta\left(\frac{p}{\gamma} + z\right) = R_m = \frac{2}{12} = 0.1667 \text{ ft}$$

Eq. (11.18): $Q = 1.06 \times 0.0218\sqrt{2(32.2) \times 0.1667} = 0.0746$ cfs

With this first determination of Q,

$$V_1 = \frac{Q}{A} = \frac{0.0746}{0.0491} = 1.519 \text{ fps}$$

Then $$D_1''V_1 = 3 \times 1.519 = 4.56$$

From Fig. 11.22, $K = 1.04$ and

$$Q = \frac{1.04}{1.06} \times 0.0746 = 0.0732 \text{ cfs} \qquad \textbf{\textit{ANS}}$$

No further correction is necessary.

EXERCISE

11.8.1 A 100-mm ISA flow nozzle (Figs. 11.21 and 11.22) is used to measure the flow of 40°C water through a 200-mm pipe. What would be the reading on a mercury manometer for the following flow rates; (*a*) 1.5 L/s; (*b*) 15 L/s; (*c*) 150 L/s.

11.9 ORIFICE METER

We can use an orifice in a pipeline, as in Fig. 11.23, as a meter in the same manner as the venturi tube or the flow nozzle. We can also place it on the end of the pipe so as to discharge a free jet. The flow rate through an orifice meter is commonly expressed as

$$Q = KA_o\sqrt{2g\Delta\left(\frac{p}{\gamma} + z\right)} \tag{11.20}$$

This is the same form as Eq. (11.18), except that A_2 is replaced by A_o, the cross-sectional area of the orifice opening. Typical values of K for a standard orifice meter are given in Fig. 11.24. The variation of K with Reynolds number is quite different from the trend of the flow coefficients for venturi tubes and flow nozzles. At high Reynolds numbers K is essentially constant, but as the Reynolds number is lowered, the value of K for the orifice increases, with the maximum value of K occurring at Reynolds numbers between 200 and 600, depending on the D_o/D_1 ratio of the orifice. The lowering of the Reynolds number increases viscous action, which causes a decrease in C_v and an increase in C_c. The latter apparently predominates over the former until C_c reaches a maximum value of about 1.0. With a further decrease in Reynolds number, K then becomes smaller because C_v continues to decrease.

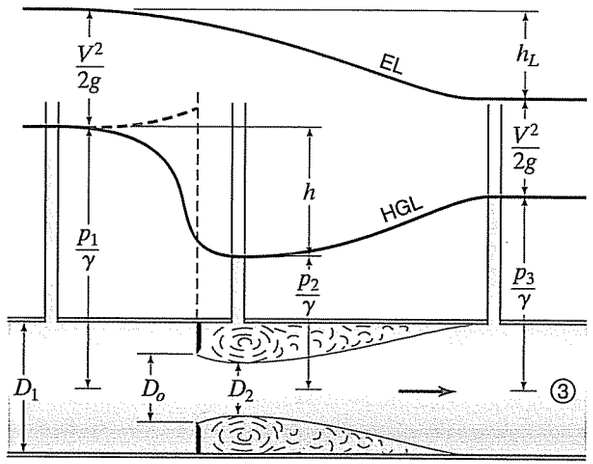
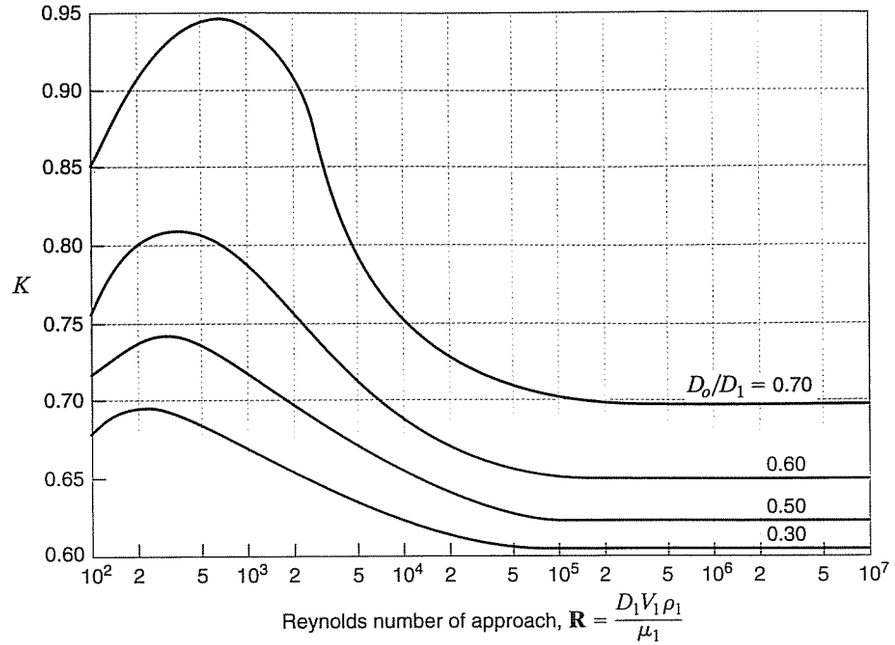


Figure 11.23
Thin-plate orifice in a pipe. (Scale distorted: the region of eddying turbulence will usually extend $4D_1$ to $8D_1$ downstream, depending on the Reynolds number.)



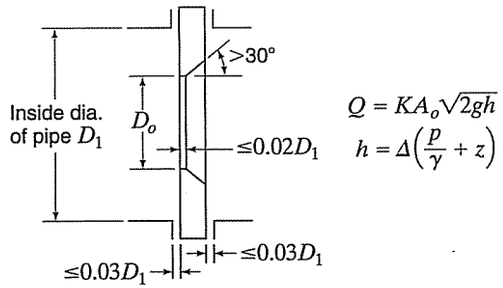


Figure 11.24
VDI orifice meter and flow coefficients for flange taps. (Adapted from *NACA Tech. Mem.* 952.)

The difference between an orifice meter and a venturi tube or flow nozzle is that for both of the latter there is no contraction, so that A_2 is also the area of the throat and is fixed, while for the orifice A_2 is the area of the jet and is a variable and is less than A_o, the area of the orifice. For the venturi tube or flow nozzle the discharge coefficient is practically a velocity coefficient, while for the orifice it is much more affected by variations in C_c than it is by variations in C_v.

The pressure differential may be measured between a point about one pipe diameter upstream of the orifice and the vena contracta, approximately one-half the pipe diameter downstream. The distance to the vena contracta is not constant, but decreases as D_o/D_1 increases. The differential can also be measured between the two corners on each side of the orifice plate. These ***flange taps***

have the advantage that the orifice meter is self-contained; the plate may be slipped into a pipeline without the necessity of making piezometer connections in the pipe.

The particular advantage of an orifice as a measuring device is that it may be installed in a pipeline with a minimum of trouble and expense. Its principal disadvantage is the greater frictional loss it causes as compared with the venturi meter or the flow nozzle.

EXERCISE

11.9.1 Repeat Exer. 11.8.1 for a VDI orifice.

11.10 FLOW MEASUREMENT OF COMPRESSIBLE FLUIDS

Strictly speaking, most of the equations that we have presented earlier in this chapter apply only to incompressible fluids, but, practically, they can be used for all liquids and even for gases and vapors where the pressure differential is small relative to the total pressure. As this is the condition usually encountered in the metering of all fluids, even compressible ones, the preceding treatment has extensive application. However, there are conditions in metering fluids where we must consider compressibility.

As for the case of incompressible fluids, we can derive equations for ideal frictionless flow and then introduce a coefficient to obtain a correct result. The ideal condition that we will impose on the compressible fluid is that the flow be isentropic, i.e., a frictionless adiabatic process (no transfer of heat, see Sec. 2.7). The latter is practically true for metering devices, as the time for the fluid to pass through is so short that very little heat transfer can take place.

Pitot Tubes

An expression applicable to pitot tubes for subsonic flow of compressible fluids can be derived by introducing the conditions at the upstream tip of the tube (that is, $V_2 = 0$ and $p_2 = p_0$) into Eq. (13.35), substituting the first expression for R from Eq. (13.4), and making use of the perfect gas relationship, Eq. (2.5). Doing so gives for pitot tubes

$$\frac{V_1^2}{2} = c_p T_1\left[\left(\frac{p_2}{p_1}\right)^{(k-1)/k} - 1\right] = c_p T_2\left[1 - \left(\frac{p_1}{p_2}\right)^{(k-1)/k}\right] \qquad (11.21)$$

We can obtain the static pressure p_1 (see Sec. 11.2) from the side openings of the pitot tube or from a regular piezometer, and the stagnation pressure p_0 ($= p_2$, Sec. 11.3) is indicated by the pitot tube itself. We must apply a coefficient if the side openings do not measure the true static pressure. Equation (11.21) does not apply to supersonic conditions, because a shock wave would form upstream of the stagnation point. In such a case a special analysis considering the effect of the shock wave is required.

Venturi Meters

To develop an expression applicable to compressible flow through venturi tubes, we take Eq. (13.35) and combine it with continuity ($g\dot{m} = G = \gamma_1 A_1 V_1 = \gamma_2 A_2 V_2$) to get

$$g\dot{m}_{\text{ideal}} = G_{\text{ideal}} = A_2\sqrt{2g\frac{k}{k-1}p_1\gamma_1\left(\frac{p_2}{p_1}\right)^{2/k}\frac{1-(p_2/p_1)^{(k-1)/k}}{1-(A_2/A_1)^2(p_2/p_1)^{2/k}}} \qquad (11.22)$$

This equation can be transformed into an equation for the actual weight rate of flow through venturi tubes by introducing the discharge coefficient C (Fig. 11.19) and an expansion factor Y. The resulting equation is

$$g\dot{m} = G = CYA_2\sqrt{2g\gamma_1\frac{p_1 - p_2}{1-(D_2/D_1)^4}} \qquad (11.23)$$

where C has the same value as for an incompressible fluid at the same Reynolds number and γ_1 can be replaced by p_1/RT_1 if desired. Values of Y for $k = 1.4$ are plotted in Fig. 11.25. For a venturi or nozzle throat where $C_c = 1$,

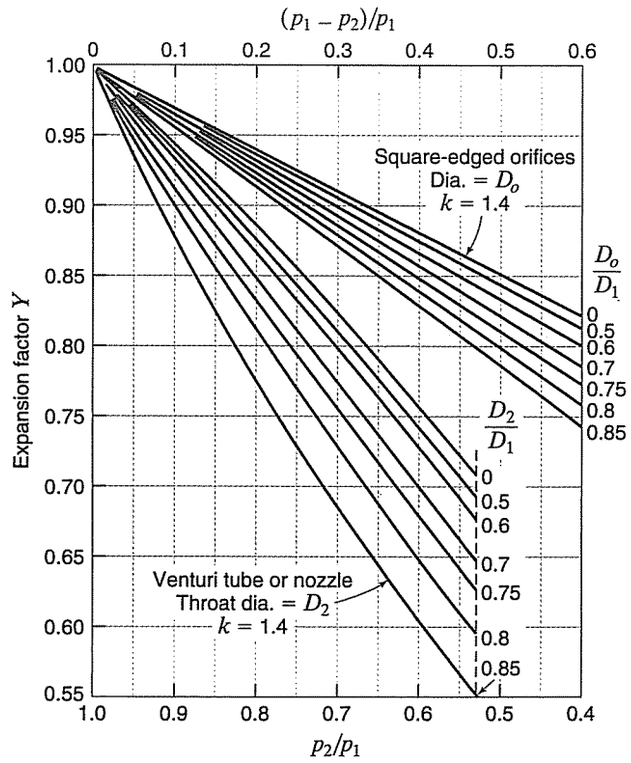


Figure 11.25
Expansion factors.

$$Y = \sqrt{\frac{[k/(k-1)](p_2/p_1)^{2/k}[1-(p_2/p_1)^{(k-1)/k}]}{1-(p_2/p_1)}}\sqrt{\frac{1-(D_2/D_1)^4}{1-(D_2/D_1)^4(p_2/p_1)^{2/k}}}$$

In Fig. 11.25 we see that for the venturi meter no values for Y are shown for p_2/p_1 ratios less than 0.528. This is so because, for air and other gases having adiabatic constant $k = 1.4$, the p_2/p_1 ratio will always be greater than 0.528 if the flow is subsonic, as is explained in Sec. 13.8.

Equation (11.23) is directly applicable to the flow of compressible fluids through venturi tubes where $C_c = 1.0$, provided the flow is subsonic.

Flow Nozzles and Orifice Meters

We can also use Eq. (11.23) for flow nozzles and orifice meters, though for flow nozzles C should be replaced by $K\sqrt{1-(D_2/D_1)^4}$ (from Eq. 11.19), so that we can use Fig. 11.22 directly. For orifice meters we should replace the C of Eq. (11.23) by $K\sqrt{1-(D_o/D_1)^4}$, we should replace D_2 by D_o, and we should replace A_2 by A_o where D_o is the diameter of the orifice opening and A_o is its area.

For compressible fluids the C_c of an orifice meter depends on the ratio p_2/p_1; hence Y varies in a different manner than in the case of a venturi. Values of Y for orifice meters are shown in Fig. 11.25. In the case of an orifice meter the maximum jet velocity is the sonic velocity c, but this does not impose a limit on the rate of discharge because the jet area continues to increase with decreasing values of p_2/p_1. For this reason the values of Y for the orifice are extended in Fig. 11.25 to lower values of p_2/p_1.

Supersonic Conditions

The general case of flow measurement under supersonic conditions will not be discussed in this text. If supersonic flow occurs in a converging or converging-diverging nozzle attached to the end of a pipe or to a tank, we can use Eqs. (13.43), (13.44), and (13.45) to compute ideal flow rates where the velocity of approach is negligible. This can be transformed into actual flow rates by introducing a proper flow coefficient.

SAMPLE PROBLEM 11.6 Determine the weight flow rate when air at 20°C and 700 kN/m^2 abs flows through a venturi meter if the pressure at the throat of the meter is 400 kN/m^2 abs. The diameters at inlet and throat are 250 and 125 mm, respectively. Assume that $C = 0.985$.

Solution

$p_2/p_1 = \dfrac{400}{700} = 0.571$; $D_2/D_1 = 0.50$. Fig. 11.25: $Y \approx 0.72$

Eq. (2.5): $\gamma_1 = \dfrac{gp}{RT} = \dfrac{(9.81\ \text{m/s}^2)(700\ \text{kN/m}^2)}{[287\ \text{m}^2/(\text{s}^2\cdot\text{K})](273+20)\ \text{K}} = 0.0817\ \text{kN/m}^3$

Eq. (11.23): $G = 0.985(0.72)\dfrac{\pi(0.125)^2}{4}\sqrt{2(9.81)0.0817\dfrac{700-400}{1-(0.5)^4}}$

$$G = 0.1971\ \text{kN/s} = 197.1\ \text{N/s} \qquad \textit{ANS}$$

If the relation between C and $\mathbf{R}_2$ for this meter is known, the value of $\mathbf{R}_2$ for the computed value of G can be determined. If the assumed value of C does not correspond with this value of $\mathbf{R}_2$, a slight adjustment in the value of C can be made to give a more accurate answer.

EXERCISES

11.10.1 Natural gas, for which $R = 3100$ ft·lb/(slug·°R) and $k = 1.3$, flows through a venturi tube. The pipe diameter is 18 in and the throat diameter is 9 in. The initial pressure of the gas is 160 psia at 64°F. What is the weight flow rate of the gas if the throat pressure is 104 psia and the meter coefficient is 0.98?

11.10.2 Helium [$R = 12{,}420$ ft·lb/(slug·°R), $k = 1.66$] is stored in a tank at 60 psia and 76°F. The gas flows out through a $\frac{3}{4}$-in-diameter orifice for which $C_v = 0.98$ and $C_c = 0.62$ for liquids. Find the weight flow rate if $Y = 0.95$ and the pressure outside the tank is 44 psia.

11.10.3 A tank contains air at 200 psia and 100°F. It flows out through an orifice with an area of 1.5 in^2, and enters a space where the pressure is 80 psia. Assuming $C_d = 0.60$, find the weight flow rate.

11.10.4 A tank contains air at 1460 kPa abs and 44°C. It flows out through an orifice with an area of 1250 mm^2, and enters a space where the pressure is 713 kPa abs. Assuming $C_d = 0.62$, find the weight flow rate.

11.10.5 Repeat Exer. 11.10.3, but discharging into a space where the pressure is 15 psia.

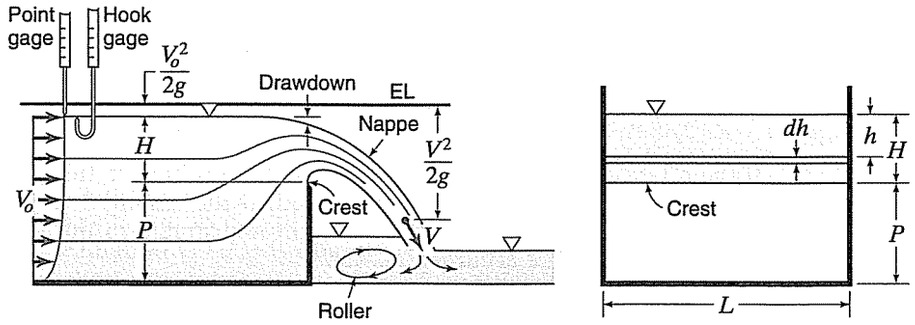
11.10.6 Repeat Exer. 11.10.4, but discharging into a space where the pressure is 105 kPa abs.

11.10.7 Find (*a*) the critical pressure and the corresponding throat velocity in a suitable nozzle, assuming air at 70°F and $p_1 = 100$ psia. Neglect the velocity of approach. Refer to Secs. 13.3, 13.5, and 13.6. (*b*) What will the values be if $D_2/D_1 = 0.80$?

11.10.8 Find (*a*) the critical pressure and the corresponding throat velocity in a suitable nozzle, assuming air at 20°C and $p_1 = 700$ kPa abs. Neglect the velocity of approach. Refer to Secs. 13.3, 13.5, and 13.6. (*b*) What will the values be if $D_2/D_1 = 0.80$?

11.11 THIN-PLATE WEIRS

Weirs have long been standard devices for the measurement of water flow. One category, known as ***thin-plate weirs,*** includes those constructed from a thin plate, usually of metal and erected perpendicular to the flow. The upstream face of the weir plate should be smooth, and the plate must be strictly vertical. All thin-plate weirs are sharp-crested; the upstream edge is formed by a horizontal



(*a*) Side view (*b*) Looking upstream

Figure 11.26
Flow over sharp-crested weir.

top surface usually less than $\frac{1}{16}$ in (1.6 mm) long in the flow direction, followed by a bevel on the downstream edge. Such a design causes the ***nappe*** (sheet of overflowing water) to spring clear as in Fig. 11.26*a*, with only a line contact at the crest, for all but the very lowest heads. If the nappe does not spring clear, we cannot consider the flow as true weir flow and the experimentally determined coefficients do not apply. The approach channel should be long enough so that a normal velocity distribution exists, and the water surface should be as free from waves as possible.

The weir crest can be straight and horizontal, or it may have a variety of shapes; a number of these are discussed below.

Suppressed Rectangular Weir

In its simplest form, water flows over the top of a plate with a straight and horizontal crest as shown in Fig. 11.26. The weir is as wide as the channel, so the width of the nappe is the same as the length of the crest. As there are no contractions of the stream at the sides, we say that end contractions are ***suppressed.*** It is essential that the sides of the channel upstream be smooth and regular. It is common to extend the sides of the channel downstream beyond the crest so that the nappe is confined laterally. The flowing water tends to entrain air from this enclosed space under the nappe, and unless this space is adequately ventilated, there will be a partial vacuum and perhaps all the air may eventually be swept out. The water will then cling to the downstream face of the plate, and the discharge will be greater for a given head than when the space is vented. Therefore venting of a suppressed weir is necessary if the standard formulas are to be applied.

We determine the rate of flow by measuring the water height H (head), above the crest, at a distance upstream from the crest equal to at least four times the maximum head to be employed. The amount of surface drawdown at the crest is typically about $0.15H$. The velocity at any point in the nappe is related to the energy line as shown in Fig. 11.26*a*.

To derive the flow equation for a rectangular weir having a crest of length L, consider an elementary area $dA = L\,dh$ in the plane of the crest, as shown in

Fig. 11.26*b*. This elementary area is in effect a horizontal slot of length L and height dh. Neglecting velocity of approach, the ideal velocity of flow through this area will be equal to $\sqrt{2gh}$. The apparent flow through this area is

$$dQ_i = L\,dh\sqrt{2gh} = L\sqrt{2g}h^{1/2}dh$$

Because each element is at a different h, the flow through it has a different velocity, so we must integrate over the whole area, from $h = 0$ to $h = H$, to obtain the total ideal flow Q_i, i.e.,

$$Q_i = \sqrt{2g}L\int_0^H h^{1/2}dh = \tfrac{2}{3}\sqrt{2g}LH^{3/2}$$

The actual flow over the weir will be less than the ideal flow, because the effective flow area is considerably smaller than LH due to drawdown from the top and contraction of the nappe from the crest below (see Fig. 11.26*a*). Introducing a coefficient of discharge C_d to account for this,

$$Q = C_d\tfrac{2}{3}\sqrt{2g}LH^{3/2} \tag{11.24}$$

Dimensional analysis of weir flow leads to some interesting conclusions that provide a basis for an understanding of the factors that influence the coefficient of discharge. The physical variables that influence the flow Q over the weir of Fig. 11.26 include L, H, P, g, μ, σ, and ρ. Using the Buckingham Π theorem (Sec. 7.7), and without going into the details, we obtain

$$Q = \phi\left(\mathbf{W}, \mathbf{R}, \frac{H}{P}\right)L\sqrt{g}H^{3/2}$$

Thus, comparing this expression with Eq. (11.24), we conclude that C_d depends on $\mathbf{W}$, $\mathbf{R}$, and H/P. It has been found that H/P is the most important of these (Probs. 11.41 and 11.46). The Weber number $\mathbf{W}$, which accounts for surface-tension effects, is important only at low heads. In the flow of water over weirs the Reynolds number is generally quite high, so viscous effects are generally insignificant. If one were to calibrate a weir for the flow of oil, however, $\mathbf{R}$ would undoubtedly affect C_d substantially. Typical values of C_d for sharp-crested weirs with water flowing range from about 0.62 for $H/P = 0.10$ to about 0.75 for $H/P = 2.0$.

Small-scale but precise experiments covering a wide range of conditions led T. Rehbock of the Karlsruhe Hydraulic Laboratory in Germany to the following expressions for C_d in Eq. (11.24):

BG units, H and P in ft:
$$C_d = 0.605 + \frac{1}{305H} + 0.08\frac{H}{P} \tag{11.25a}$$

SI units, H and P in m:
$$C_d = 0.605 + \frac{1}{1000H} + 0.08\frac{H}{P} \tag{11.25b}$$

These equations were obtained by fitting a curve to the plotted values of C_d for a great many experiments and are purely empirical. Capillarity is accounted for

by the second term, while velocity of approach (assumed to be uniform) is responsible for the last term. Rehbock's formula has been found to be accurate within 0.5% for values of P from 0.33 to 3.3 ft (0.10 to 1.0 m) and for values of H from 0.08 to 2.0 ft (0.025 to 0.60 m) with the ratio H/P not greater than 1.0. It is even valid for ratios greater than 1.0 if the bottom of the discharge channel is lower than that of the approach channel so that backwater does not affect the head.

It is convenient to express Eq. (11.24) as

$$Q = C_W L H^{3/2} \tag{11.26}$$

where C_W, the ***weir coefficient,***[10] replaces $C_d \frac{2}{3}\sqrt{2g}$.

Using a value of 0.62 for C_d in Eq. (11.24), we can write

$$Q \approx \begin{cases} 3.32 L H^{3/2} & \text{in BG units} \\ 1.83 L H^{3/2} & \text{in SI units} \end{cases} \tag{11.27}$$

These equations give good results if $H/P < 0.4$, which is well within the usual operating range. If the velocity of approach V_0 is appreciable, a correction must be applied to the preceding equations either by changing the form of the equation or, more commonly, by changing the value of the coefficient.

Rectangular Weir with End Contractions

When the length L of the crest of a rectangular weir is less than the width of the channel, the nappe will have ***end contractions*** so that its width is less than L. Experiments have indicated that under the conditions depicted in Fig. 11.27 the effect of each side contraction is to reduce the effective width of the nappe by $0.1H$. Hence for such a situation the flow rate may be computed by employing any of Eqs. (11.26) and (11.27) and substituting $(L - 0.1nH)$ for L, where n is the number of end contractions, normally 2 but sometimes 1. We call the results ***Francis formulas.***

[10] Since C_W is not dimensionless, its value in BG units is different from that in SI units, as indicated in Eq. (11.27).

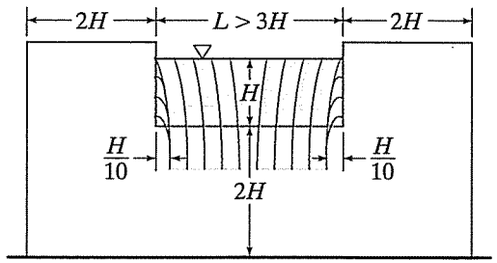


Figure 11.27
Limiting proportions of standard contracted weirs.

Cipolletti Weir

In order to avoid correcting for end contractions we often use a Cipolletti weir. It has a trapezoidal shape with side slopes of four vertical on one horizontal. The additional area adds approximately enough to the effective width of the stream to offset the lateral contractions.

V-notch, or Triangular, Weir

For relatively small flows the rectangular weir must be very narrow and thus of limited maximum capacity, or else the value of H will be so small that the nappe will not spring clear but will cling to the plate. For such a case the V-notch or triangular weir has the advantage that it can function for a very small flow and also measure reasonably large flows as well. The vertex angle is usually between 10° and 90° but rarely larger.

In Fig. 11.28 is a V-notch weir with a vertex angle θ. The ideal rate of discharge through an elementary area dA is $dQ = \sqrt{2gh}\,dA$. Now $dA = 2x\,dh$, and $x/(H - h) = \tan(\theta/2)$. Substituting in the foregoing, and introducing a coefficient of discharge C_d, we get this result for the entire notch:

$$Q = C_d 2\sqrt{2g} \tan\frac{\theta}{2} \int_0^H (H - h) h^{1/2}\, dh$$

Integrating between limits and reducing, the fundamental equation for all V-notch weirs is:

$$Q = C_d \frac{8}{15}\sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \tag{11.28}$$

For a given angle θ and assuming C_d is constant, this may be reduced to

$$Q = K H^{5/2} \tag{11.29}$$

The value of the constant K in English units will be different from that in SI units.

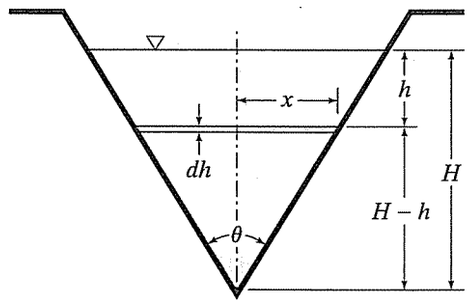


Figure 11.28
V-notch weir.

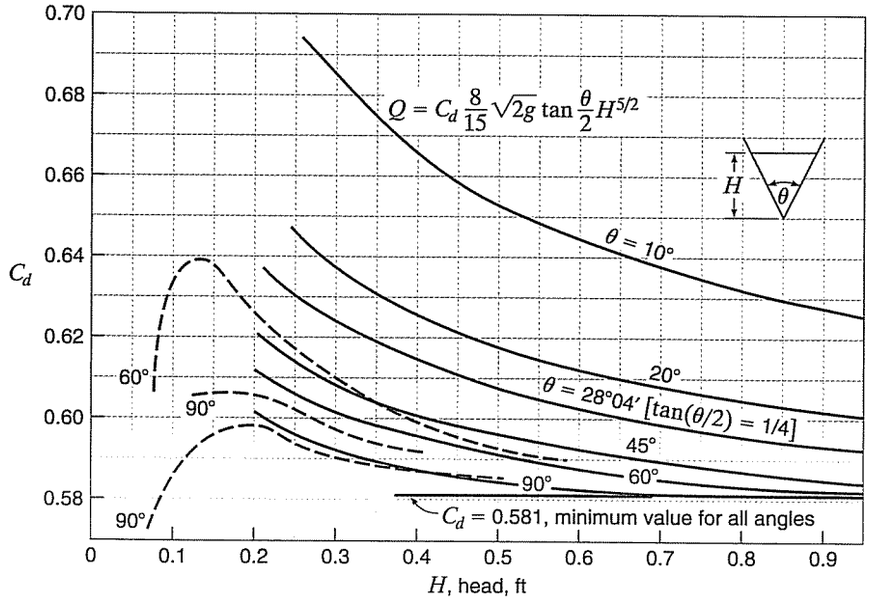


Figure 11.29
Coefficients for V-notch weirs obtained by three different experimenters.

In Fig. 11.29 we present experimental values for C_d for water flowing over V-notch weirs with central angles varying from 10° to 90°. The rise in C_d at heads less than 0.5 ft is due to incomplete contraction. At lower heads the frictional effects reduce the coefficient. At very low heads, when the nappe clings to the weir plate, the phenomenon can no longer be classed as weir flow and Eqs. (11.28) and (11.29) are inapplicable.

Proportional Weirs

While, for given simple geometric shapes of thin-plate weirs, head-discharge (H–Q) relations can easily be obtained using elementary calculus, the reverse problem of working out the shapes of weirs that produce desired H–Q relations is more challenging and generally requires the solution of integral equations. Research[11] has led to the development of new types of nonlinear weirs like the logarithmic weir ($Q \propto \log H$) and the quadratic weir ($Q \propto H^{1/2}$), called ***proportional weirs.*** Many of these proportional weirs measure discharge more accurately than traditional weirs, and we use them as efficient velocity controllers in sediment settling chambers and in industrial dosing facilities. An example of a linear proportional weir known as a Sutro Weir is given in Prob. 11.47.

[11] K. Keshava Murthy and D. P. Giridhar, Improved Inverted V-Notch or Chimney Weir, *J. Irrigation and Drainage Engineering, ASCE,* vol. 116, no. 3, 1990; K. Keshava Murthy and M. N. S. Prakash, Practical Constant Accuracy Linear Weir, *J. Irrigation and Drainage Engineering, ASCE,* vol. 120, no. 3, 1994.

SAMPLE PROBLEM 11.7 Water is flowing in a rectangular channel at a velocity of 3 fps and depth of 1.0 ft. Neglecting the effect of velocity of approach and employing Eq. (11.27), determine the height of a sharp-crested suppressed weir that must be installed to raise the water depth upstream of the weir to 4 ft.

Solution

$$L = \text{length of weir crest} = \text{width of channel}$$

Eq. (11.27): $$Q = AV = LyV = L(1)(3) = 3.33LH^{3/2}$$

$$H^{3/2} = \frac{3.0}{3.33} = 0.901, \quad H = 0.933 \text{ ft}$$

Fig. 11.26: $P = \text{height of weir required} = 4.00 - 0.933 = 3.07 \text{ ft}$ ***ANS***

EXERCISES

11.11.1 A 2.5-ft-high rectangular sharp-crested weir extends across a 6-ft-wide rectangular channel. When the head is 1.12 ft, determine the flow rate by neglecting the velocity of approach.

11.11.2 A 0.75-m-high rectangular sharp-crested weir extends across a 2.2-m-wide rectangular channel. When the head is 320 mm, determine the flow rate by neglecting the velocity of approach.

11.11.3 Suppose the rectangular weir of Exer. 11.11.1 is contracted at both ends. (*a*) Find the flow rate for a head of 1.12 ft by the Francis formula. (*b*) What would be the maximum value of H for which the Francis formula could be used?

11.11.4 Suppose the rectangular weir of Exer. 11.11.2 is contracted at both ends. (*a*) Find the flow rate for a head of 320 mm by the Francis formula. (*b*) What would be the maximum value of H for which the Francis formula could be used?

11.11.5 (*a*) What is the rate of discharge of water over a 45° triangular weir when the head is 0.6 ft? (*b*) With the same head, what would be the increase in discharge obtained by doubling the notch angle, i.e., for a 90° weir? (*c*) What would be the head for a discharge of 1.5 cfs of water over a 60° triangular weir?

11.12 STREAMLINED WEIRS AND FREE OVERFALL

In contrast to thin-plate weirs, streamlined weirs have large dimensions in the flow direction and they must have rounded crests and/or approach edges (they are not sharp-crested). Shapes of streamlined weir cross sections include rectangular with a rounded upstream edge, triangular with a rounded crest, and hydrofoil shapes (similar to airfoils).

Streamlined weirs are usually built of concrete, giving them the advantage of being rugged and able to stand up well under field conditions.

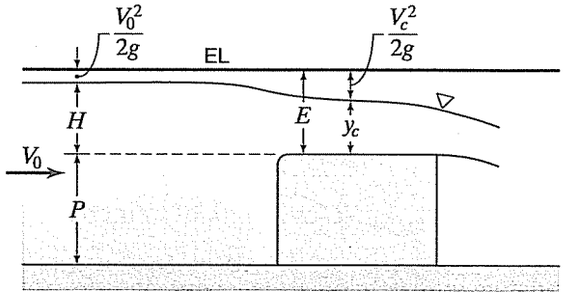


Figure 11.30
Broad-crested rectangular weir.

Broad-Crested Rectangular Weir

The broad-crested weir (Fig. 11.30), as noted in Sample Prob. 10.8, is a critical-depth meter; that is, if the weir is high enough (Sec. 10.13) critical depth occurs on the crest of the weir. In Eq. (10.22) we saw that for a rectangular channel $V_c = \sqrt{gy_c}$, while Eq. (10.26) stated that when the flow is critical $y_c = \frac{2}{3}E$. Using these relations, we can write for the flow over a broad-crested weir,

$$Q = AV = (Ly_c)\sqrt{gy_c} = L\sqrt{g}y_c^{3/2} = L\sqrt{g}\left(\tfrac{2}{3}\right)^{3/2}E^{3/2} \qquad (11.30)$$

where L is the width of the weir (length of crest), and E is the total head over the weir, equal to $H + V_0^2/2g$, in which H is the measured head and V_0 is the approach velocity from upstream. Let us now substitute Eq. (11.30) into Eq. (11.24), which is applicable to broad-crested weirs as well as sharp-crested suppressed rectangular weirs, since both have rectangular flow cross sections. This yields

$$C_d = \frac{1}{\sqrt{3}}\left(\frac{E}{H}\right)^{3/2} \qquad (11.31)$$

For very high weirs (that is, H/P small) the velocity of approach becomes small, so that $E \to H$ (Fig. 11.30) and thus $E/H \to 1$ and $C_d \to 1/\sqrt{3} = 0.577$. With a lower weir and the same flow rate the velocity of approach becomes larger and hence E/H increases in magnitude, resulting in larger values of C_d for lower broad-crested weirs. The actual value of C_d for a broad-crested weirs depends on the length of the weir and whether or not the upstream corner (edge) of the weir is rounded. The foregoing discussion, of course, assumes critical flow (Sec. 10.9) on the weir. If the approaching flow is supercritical, the presence of surface waves causes broad-crested rectangular weirs to be rather impractical for use as metering devices.

Other Streamlined Weirs

Many different streamlined weirs have been tested[12] and found not only to have high discharge coefficients but also to function efficiently under conditions of high submergence. Weir ***submergence*** σ is defined as the downstream water

[12] N. S. Lakshmana Rao, Theory of Weirs, *Advances in Hydroscience,* V. T. Chow, ed., Academic Press, N.Y., vol. 10, 1975.

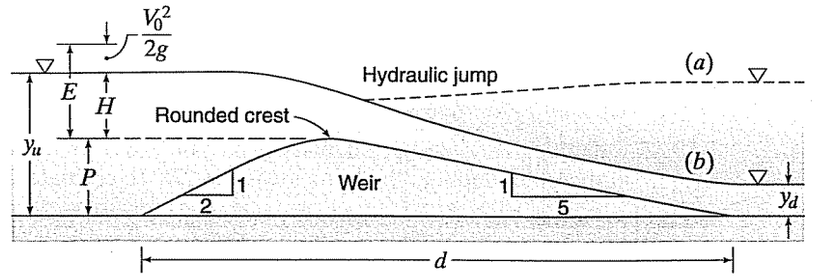


Figure 11.31
Streamlined triangular weir, $d/P = 7.5$, $C_d = 1.11$. Free surface at (a) critical submergence, $\sigma_c = 0.82$, and (b) $\sigma = 0.2$.

depth divided by the upstream water depth, i.e., $\sigma = y_d/y_u$. For sufficiently low values of σ the discharge over the weir is unaffected by the downstream depth; for increasing σ, when the discharge is first reduced by a prescribed small amount like 2 or 5% the submergence is defined to be the ***critical submergence*** σ_c.

The coefficient of discharge C_d of a streamlined weir, according to the International Standards Organization (ISO), is defined by the equation

$$Q = C_d\left(\tfrac{2}{3}\right)^{3/2}\sqrt{g}LE^{3/2} \qquad (11.32)$$

where the variables are as for Eq. (11.30) (see also Figs. 11.30 and 10.13). Note that this equation is the same as Eq. (11.30) with a discharge coefficient added.

Over 25 streamlined weirs have been classified into nine categories, depending on (a) whether σ_c is low (<0.65), medium (0.65–0.80), or high (>0.80), and (b) whether the weir discharge coefficient C_d is low (<1.0), medium (1.0–1.3), or high (>1.3). For example, several triangular-shaped streamlined weirs, and what we call hydrofoil weirs, fall into the category of medium C_d and high σ_c types. The streamlined triangular weir of Fig. 11.31 has a medium C_d and a high σ_c of 0.82 when we specify a discharge reduction of 6%.

Free Overfall

Another method for determining the flow rate in a rectangular channel is to measure the depth of flow y_b at the brink of a free overfall (Fig. 10.17). Substituting $y_c = y_b/0.72$ in Eq. (10.23) permits an approximate determination of the rate of flow per foot of channel width. This method can be used only if the flow is subcritical.

EXERCISES

11.12.1 What is the flow rate per unit width over a broad-crested weir that rises 1.0 ft above the bed of a horizontal channel when the head is 3.0 ft above the crest?

11.12.2 What is the flow rate per unit width over a broad-crested weir that rises 0.3 m above the bed of a horizontal channel when the head is 0.9 m above the crest?

11.12.3 Find the water depth just upstream of a 1.5-ft-high broad-crested weir in a channel 6 ft wide when the flow is 18 cfs.

11.12.4 Find the water depth just upstream of a 0.4-m-high broad-crested weir in a channel 1.8 m wide when the flow is 0.54 m^3/s.

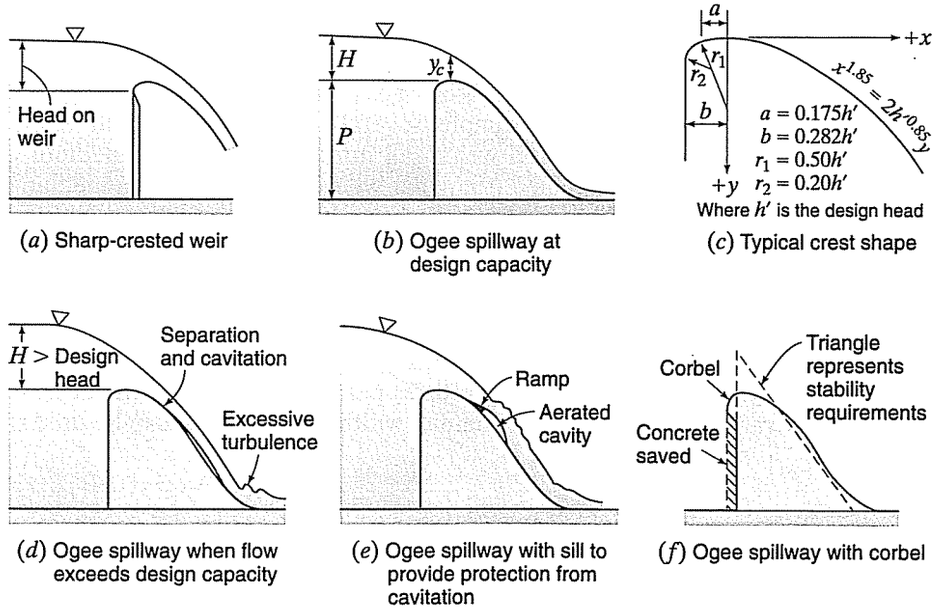
11.12.5 Subcritical flow in a rectangular open channel 6 ft wide ends in a free overfall. If the water depth at the brink measures 2.70 ft, estimate the flow rate.

11.12.6 Subcritical flow in a rectangular open channel 2 m wide ends in a free overfall. If the water depth at the brink measures 0.75 m, estimate the flow rate.

11.13 OVERFLOW SPILLWAY

An overflow spillway is a section of dam designed to permit water to pass over its crest. Overflow spillways are widely used on gravity, arch, and buttress dams. The ideal spillway should take the form of the underside of the nappe of a sharp-crested weir (Fig. 11.32*a*) when the flow rate corresponds to the maximum design capacity of the spillway (Fig. 11.32*b*). Figure 11.32*c* defines the shape of an ***ogee spillway,*** which closely approximates the ideal[13]; ogee means having an

[13] Hydraulic Models as an Aid to the Development of Design Criteria, U.S. Waterways Expt. Sta., Bull. 37, Vicksburg, Miss., 1951.



(*a*) Sharp-crested weir (*b*) Ogee spillway at design capacity (*c*) Typical crest shape

(*d*) Ogee spillway when flow exceeds design capacity (*e*) Ogee spillway with sill to provide protection from cavitation (*f*) Ogee spillway with corbel

Figure 11.32
Characteristics of an ogee spillway.

S-shaped curve. The reverse curve on the lower downstream face of the spillway should be smooth and gradual; a radius of about one-fourth of the spillway height has proved satisfactory.

The discharge of an overflow spillway is given by the weir equation, (11.26):

$$Q = C_w L H^{3/2} \tag{11.33}$$

where Q = discharge, cfs or m^3/s
C_w = weir coefficient
L = length of the crest, ft or m
H = head on the spillway (vertical distance from the crest of the spillway to the reservoir level), ft or m

The coefficient C_w varies with the design and head. For the standard overflow crest of Fig. 11.32*c* the variation of C_w is given in Fig. 11.33. Experimental models are often used to determine spillway coefficients. End contractions on a spillway reduce the effective length below the actual length L. Square-cornered piers disturb the flow considerably and reduce the effective length by the width of the piers plus about $0.2H$ for each pier. Streamlining the piers or flaring the spillway entrance minimizes the flow disturbance. If the cross-sectional area of the reservoir just upstream from the spillway is less than five times the area of flow over the spillway, the approach velocity will increase the discharge a noticeable amount. The effect of approach velocity can be accounted for by

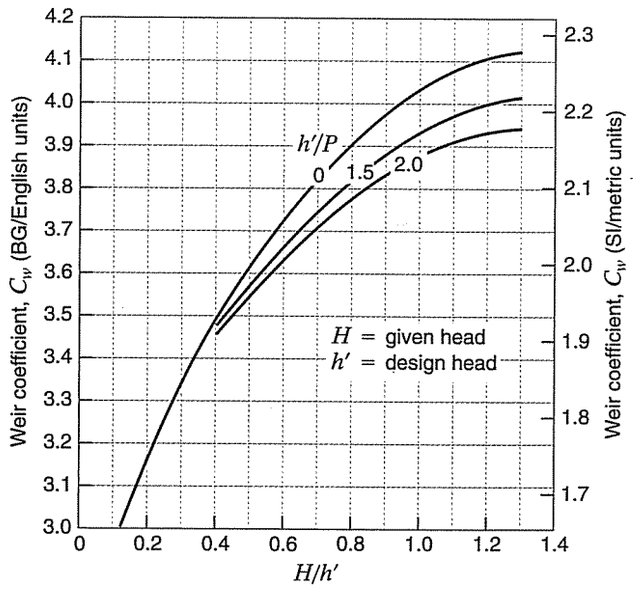


Figure 11.33
Variation of discharge coefficient with head for an ogee spillway crest such as shown in Fig. 11.32.

the equation

$$Q = C_v L\left(H + \frac{V_0^2}{2g}\right)^{3/2} \tag{11.34}$$

where V_0 is the approach velocity.

On high spillways, if the overflowing water breaks contact with the spillway surface, a vacuum will form at the point of separation (Fig. 11.32*d*) and cavitation may occur. Cavitation and vibration from the alternate making and breaking of contact between the water and the face of the spillway may result in serious structural damage. A ramp of proper shape and size when properly located (Fig. 11.32*e*) will direct the water away from the spillway surface to form a cavity.[14] To be effective, air must be freely admitted to the cavity. The result is that air is entrained in the water, the water bulks up, and when it returns to the spillway surface there is no problem with cavitation. On very high spillways these ramps may be used in tandem.

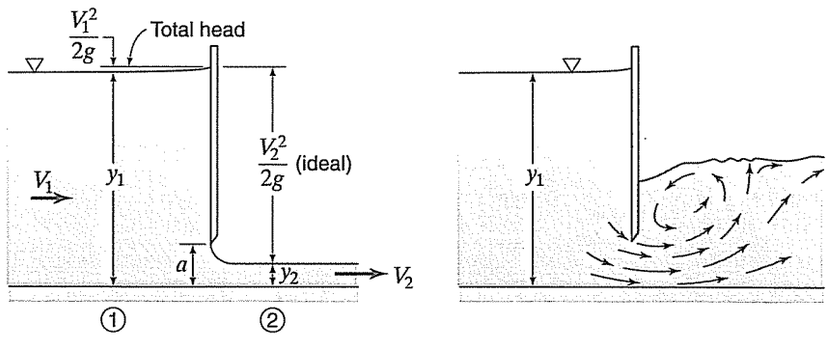
Other types of spillways include chute, side-channel, ski-jump, and shaft spillways.[15]

11.14 SLUICE GATE

The sluice gate shown in Fig. 11.34 is a device used to control the passage of water in an open channel. When properly calibrated, it may also serve as a means of flow measurement. As the lower edge of the gate opening is flush with the floor of the channel, contraction of the bottom surface of the issuing stream

[14] K. Zagustin and N. Castillejo, Model-Prototype Correlation for Flow Aeration in Guru Dam Spillway, *Proceedings International Association for Hydraulic Research,* vol. 3, 1983.

[15] For a brief discussion of the different types of spillways see Ref. 58 in Appendix E.



(*a*) Free flow ($y_2 = C_c a$) (*b*) Submerged flow

Figure 11.34
Flow through sluice gate.

is entirely suppressed. Side contractions will of course depend on the extent to which the opening spans the width of the channel. The complete contraction on the top side, however, because of the larger velocity components parallel to the face of the gate, will offset the suppressed bottom contraction, resulting in a coefficient of contraction nearly the same as for a slot with contractions at top and bottom.

Flow through a sluice gate differs fundamentally from flow through a slot in that the jet is not free but guided by a horizontal floor. Consequently, the final jet pressure is not atmospheric, but distributed hydrostatically in the vertical section. Writing the energy equation with respect to the stream bed as datum from point 1 to point 2 in the free-flow case (Fig. 11.34*a*) and neglecting head loss,

$$\frac{V_1^2}{2g} + y_1 = \frac{V_2^2}{2g} + y_2$$

from which, introducing continuity, the ideal velocity downstream is

$$V_{2i} = \frac{1}{\sqrt{1 - (A_2/A_1)^2}}\sqrt{2g(y_1 - y_2)} \tag{11.35}$$

The actual flow rate $Q = C_d Q_i = C_c C_v (A V_{2i})$, where $A = aB$ is the area of the gate opening and B is the width of the opening. For free flow, $y_2 = C_c a$.

Absorbing the effects of flow contraction, friction, velocity of approach, and the downstream depth y_2 into an experimental flow coefficient, we obtain a simple discharge equation for flow under a sluice gate:

$$Q = K_s A\sqrt{2gy_1} \tag{11.36}$$

where K_s is defined as the ***sluice coefficient.***[16]

Values of K_s depend on a and y_1 (Fig. 11.34*a*) and are usually between 0.55 and 0.60 for free flow, but they are significantly reduced when the flow conditions downstream cause submerged flow, as shown in Fig. 11.34*b*.

EXERCISES

11.14.1 A rectangular channel contains a sluice gate that extends across the full width of the channel with an opening of 0.24 ft. Given $C_d = 0.60$ and free flow, find the flow rate per unit width and the sluice coefficient when the upstream and downstream depths are 2.05 ft and 0.15 ft, respectively.

11.14.2 A rectangular channel contains a sluice gate that extends across the full width of the channel with an opening of 100 mm. Given $C_d = 0.60$ and free flow, find the flow rate per unit width and the sluice coefficient when the upstream and downstream depths are 850 mm and 60 mm, respectively.

[16] Values of discharge coefficients for sluice and other types of gates may be found in Hunter Rouse (ed.), *Engineering Hydraulics,* pp. 536–543, John Wiley & Sons, Inc., New York, 1950.

11.15 MEASUREMENT OF LIQUID-SURFACE ELEVATION

We can determine the elevation of the surface of a liquid at rest through use of a piezometer column, manometer, or pressure gage (Sec. 3.5). These will also give accurate results when applied to a stationary liquid contained in a tank that is moving, provided the tank is not undergoing an acceleration (Sec. 3.10). Staff gages, such as those used at reservoirs, provide approximate liquid surface elevation data.

Various methods are used to measure the surface elevation of moving liquids. To determine the head H on a weir, we must measure the elevation difference between the crest of the weir and the liquid surface. In the field the elevation of the liquid surface is often determined through use of a ***stilling well*** connected by a pipe to the main liquid body. A ***float*** in the well is used to actuate a clock-driven liquid-level recorder so that a continuous record of the liquid-surface elevation is obtained. In the laboratory a ***hook gage*** or ***point gage*** (Fig. 11.26) is commonly employed for liquid-surface level determinations. The point gage is particularly suitable for fast-moving liquids where a hook gage would create a local disturbance in the liquid surface. In all liquid-surface level determinations care should be taken to make the measurements in regions where there is no curvature of streamlines; otherwise centrifugal effects will give a false reading of the piezometric head.

Other methods for determining the elevation of a liquid surface include the use of the sonic devices, electric gages, and bubblers. A ***sonic device*** is mounted at some convenient location above the liquid surface and the time required for a sound pulse to travel vertically downward to the liquid surface and return is indicative of the relationship between the elevation of the liquid surface and the device. ***Electrical gages*** include those with capacitive sensors and those with resistive sensors. In the resistive type two parallel bare-wire conductors are partially immersed in the liquid as a component of an electrical system. The electrical resistance between them is a function of the liquid depth. With proper circuitry and calibration, the device can be used to provide data on liquid-surface elevation. The sonic devices and the electrical gages are equally applicable to liquids at rest or in motion. The ***bubbler*** is used primarily for liquids at rest. In the bubbler system, the minimum pressure required to drive a gas into a liquid (i.e., to form bubbles) at a depth is a measure of the depth of the liquid. If we know the elevation at which the bubbles are emitted we can determine the elevation of the liquid surface.

11.16 OTHER METHODS OF MEASURING DISCHARGE

In addition to the foregoing "standard" devices for measuring the flow of fluids, a number of supplementary devices are less amenable to exact theoretical analysis but are worthy of brief mention. One of the simplest for measuring flow in a pipeline is the ***elbow meter,*** which consists of nothing more than piezometer taps at the inner and outer walls of a 90° elbow in the line (see Sec. 8.26). The pressure difference, due to the centrifugal effects at the bend, will vary

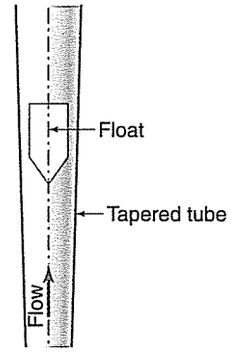


Figure 11.35
Rotameter type of flow meter.

approximately as the velocity head in the pipe. Like other meters, the elbow should have sections of a straight pipe upstream and downstream and should be calibrated in place.

The ***rotameter*** (Fig. 11.35) consists of a vertical glass tube that is slightly tapered, in which the metering ***float*** is suspended by the upward motion of the fluid around it. Directional notches cut in the float keep it rotating and thus free of wall friction. The rate of flow determines the equilibrium height of the float, and the tube is graduated to read the flow directly. We also use the rotameter for gas flow, but the weight of the float and the graduation must be changed accordingly.

The ***inferential meter*** or ***turbine meter*** consists of a propeller or wheel with curved blades, shaped so that the flow of gas or liquid passing through it causes it to rotate. The rotational speed indicates the flow rate, after calibration. Such meters are often provided with guide vanes that we can adjust to change the calibration.

Other techniques for measuring flow rate include the ***salt-velocity*** method. In this method a charge of concentrated salt is injected into the flow at an upstream station. Its arrival at a downward station is detected from conductivity measurements. In flowing between the two stations, the salt disperses and its arrival at the downstream station is spread out over a considerable period of time. The time of travel between the two stations is taken as the time from the instant of injection at the upstream station to the time at which the centroid of the conductivity-time curve passes the downstream station. Knowing the travel time and the distance, the velocity can be calculated, and then by multiplying by the cross-sectional area, we get the flow rate. In special circumstances other substances such as dye or tritium may be used instead of salt.

PROBLEMS

11.1 A hydrometer (Figs. 3.21 and 3.22*a*) is made in the form of an 8-in-long cylinder with a diameter of $\frac{3}{8}$ in. Attached to the end of the cylinder is a 1.2-in-diameter sphere. The entire device weighs 0.9 oz. What range of specific gravities can be measured with this device? When floating in a liquid with a density of 1.80 slugs/ft^3, how much of the cylinder will project above the surface?

11.2 A hydrometer (Figs. 3.21 and 3.22*a*) is made in the form of a 200-mm-long cylinder with a diameter of 6 mm. Attached to the end of the cylinder is a 22-mm-diameter sphere. The mass of the entire device is 8 g. What range of specific gravities can be measured with this device? When floating in a liquid having a density of 900 kg/m^3, how much of the cylinder will project above the surface?

11.3 Fifty milliliters of 70°F water flows through a certain tube-type viscometer in 53.7 sec, whereas the same volume of 50°F oil ($s = 0.90$) flows through it in 850 sec. Find the absolute viscosity of the oil.

11.4 A liquid ($\rho = 889$ kg/m^3) under a head of 500 mm flows at a steady rate of 30 mL/min through a glass tube of diameter 2.0 mm and length 4.5 m. What is the absolute viscosity and the kinematic viscosity of the liquid? Express the answers in stokes and poises. Confirm by calculation that the flow is laminar.

11.5 A glass bead ($s = 2.60$) with a diameter of 16 mm falls through a liquid ($s = 1.59$) in a 100-mm-diameter tube. If the bead moves with a constant velocity of 145 mm/min, what is the absolute viscosity and the kinematic viscosity of the liquid?

11.6 A 3.6-in-diameter tube contains oil ($s = 0.93$) with an absolute viscosity of 0.0065 lb·sec/ft^2. What is the maximum size of steel sphere ($s = 7.84$) that will satisfy Stokes' law when released into this oil? What will be the fall velocity of this sphere?

11.7 A 100-mm-diameter tube contains oil ($s = 0.9$) with a viscosity of 0.25 N·s/m^2. What is the maximum size of steel sphere ($s = 7.8$) that will satisfy Stokes' law when released into this oil? What will be the fall velocity of this sphere?

11.8 An 0.10-in-diameter sphere has a velocity of 0.005 fps when falling through liquid in a 1.0-in-diameter tube. What will be its fall velocities through the same liquid in tubes of diameter 0.50, 2.0, 4.0, and 10.0 in? Plot fall velocity vs tube diameter.

11.9 In Fig. S11.1, pressure gages A and B read 11.0 psi and 12.0 psi, respectively. If 60°F air is flowing, what is the velocity u? Atmospheric pressure is 27.0 inHg. Assume $C_I = 1.0$ and neglect compressibility effects.

11.10 Suppose the two pressure gages of Prob. 11.9 are replaced by a differential manometer containing water. What is the reading on the manometer?

11.11 The pitometers in Fig. P11.11 are connected to a mercury manometer, which reads 5.0 in. The velocity of flowing carbon tetrachloride ($s = 1.59$) is known to be 9.37 fps. What is C_I for this instrument?

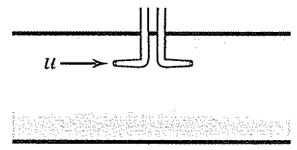


Figure P11.11

11.12 The pitometers in Fig. P11.11 are connected to a mercury manometer, which reads 120 mm. The velocity of flowing carbon tetrachloride ($s = 1.59$) is known to be 2.76 m/s. What is C_I for this instrument?

11.13 Given air at 60°C and standard atmospheric pressure is flowing in Fig. P11.11. The pitometers are attached to a manometer containing a liquid with $s = 0.90$. Plot the velocity u versus the manometer reading assuming $C_I = 0.93$.

11.14 Water at 90°F flows through a smooth, 12-in-diameter pipe. A Prandtl tube is placed on the centerline of the pipe. The reading on a differential manometer attached to this Prandtl tube is 8 in of carbon tetrachloride ($s = 1.59$). What is the flow rate?

11.15 Assume the pipe diameter in Fig. 11.10 is 16 in, and the flow is laminar, i.e., $u = u_{max} - kr^2$, when $u_{max} = 0.1$ fps. Divide the circle into concentric rings with radii of 2, 4, 6, and 8 in, and compute the flow rate by the method of Fig. 11.10 by taking the velocities at radii of 1, 3, 5, and 7 in as representative of the rings. Compare the result with that obtained by integration.

11.16 Assume the pipe diameter in Fig. 11.10 is 480 mm, and the flow is laminar, i.e., $u = u_{max} - kr^2$, when $u_{max} = 30$ mm/s. Divide the circle into concentric rings with radii of 60, 120, 180, and 240 mm, and compute the flow rate by the method of Fig. 11.10 by taking the velocities at radii of 30, 90, 150, and 210 mm as representative of the rings. Compare the result with that obtained by integration.

11.17 Water flows at 12 fps through a horizontal 3-in-diameter pipe. A nozzle at the end of the pipe has a velocity coefficient of 0.98. If the pressure in the pipe is 7.5 psi, find (*a*) the velocity in the jet; (*b*) the rate of discharge; (*c*) the diameter of the jet; and (*d*) the head loss through the nozzle.

11.18 Water flows at 3.5 m/s through a horizontal 120-mm-diameter pipe. A nozzle at the end of the pipe has a velocity coefficient of 0.98. If the pressure in the pipe is 45 kPa, find (*a*) the initial velocity of the jet; (*b*) the rate of discharge; (*c*) the diameter of the jet; and (*d*) the head loss through the nozzle.

11.19 In Fig. P11.19 the pressure at the base of the nozzle, point 1, is 21.0 psi, and the power available in the jet at point 2 is 3.42 hp. Find (*a*) the theoretical height to which the jet will rise; (*b*) the coefficient of velocity for this nozzle; (*c*) the head loss between points 1 and 2; and (*d*) the theoretical diameter of the jet at a point 20 ft above point 2.

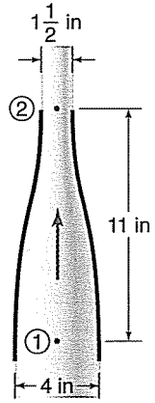


Figure P11.19

11.20 In Fig. P11.19 the pressure at the base of the nozzle, point 1, is 145 kPa, and the power available in the jet at point 2 is 2.55 kW. Find (*a*) the theoretical height to which the jet will rise; (*b*) the coefficient of velocity for this nozzle; (*c*) the head loss between points 1 and 2; and (*d*) the theoretical diameter of the jet at a point 6 m above point 2.

11.21 We can express the loss of head due to friction in a nozzle, tube, or orifice as $h_L = kV^2/2g$, where V is the actual velocity of the jet. (*a*) What is k for the three tubes in Fig. 11.16? (*b*) If these tubes discharge water under a head of 6 ft, what is the loss of head for each?

11.22 We can express the loss of head due to friction in a nozzle, tube, or orifice as $h_L = kV^2/2g$, where V is the actual velocity of the jet. (*a*) What is k for the three tubes in Fig. 11.16? (*b*) If these tubes discharge water under a head of 1.8 m, what is the loss of head for each?

11.23 Find the maximum theoretical head h at which the Borda tube of Fig. 11.17 will flow full if the liquid is 90°F water and the barometer reads 29.2 inHg. Assume $C_d = 0.72$ when the tube is flowing full.

11.24 Find the maximum theoretical head h at which the Borda tube of Fig. 11.17 will flow full if the liquid is 30°C water and the barometer reads 735 mmHg. Assume $C_d = 0.72$ when the tube is flowing full.

11.25 In Fig. P11.25 the orifice at the bottom of the large open tank has a diameter of 1.27 in. If the flow rate is 0.19 cfs, and the pitot tube registers a pressure of 16.8 psi, find the C_c and the C_v of the orifice. Neglect air resistance.

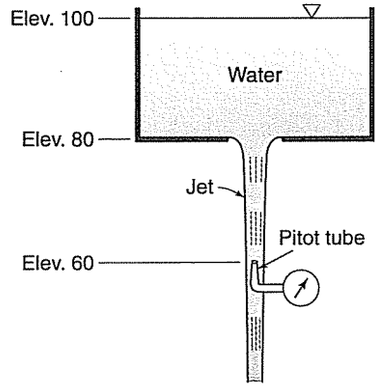


Figure P11.25

11.26 The venturi meter of Fig. S11.4 has $D_1 = 10$ in, $D_2 = 5$ in, and $\Delta z = 2$ ft. Assume the discharge coefficients of Fig. 11.19 are applicable and the diverging cone angle is 12°. (*a*) Find the flow rate of 72°F water when the mercury ($s = 13.55$) manometer reads $R_m = 5$ in. Find also the head loss (*b*) from inlet to throat, and (*c*) across the entire meter.

11.27 The venturi meter of Fig. S11.4 has $D_1 = 250$ mm, $D_2 = 125$ mm, and $\Delta z = 600$ mm. Assume the discharge coefficients of Fig. 11.19 are applicable and the diverging cone angle is 12°. (*a*) Find the flow rate of 20°C water when the mercury ($s = 13.55$) manometer reads $R_m = 12.8$ mm. Find also the head loss (*b*) from inlet to throat, and (*c*) across the entire meter.

11.28 Repeat Prob. 11.26 when the venturi meter is set horizontally (i.e., $\Delta z = 0$), with all other data the same.

11.29 Repeat Prob. 11.27 when the venturi meter is set horizontally (i.e., $\Delta z = 0$), with all other data the same.

11.30 An 80-mm ISA flow nozzle (Figs. 11.21 and 11.22) is used to measure the flow of 40°C water through a 160-mm pipe. What would be the reading on a mercury manometer for the following flow rates: (*a*) 3.8 L/s; (*b*) 38 L/s; (*c*) 380 L/s? (*d*) For which of the preceding three flows is the mercury manometer practical?

11.31 A 7-in ISA flow nozzle is used to measure the flow of 20°F crude oil ($s = 0.855$) through a 10-in-diameter pipe (Fig. P11.31). If a mercury manometer shows a reading of 6.9 in, find the flow rate.

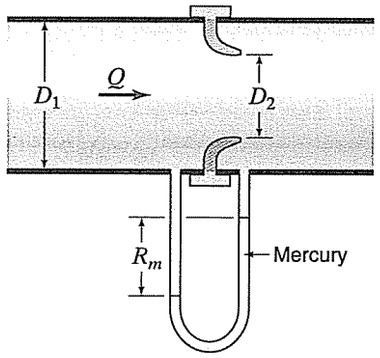


Figure P11.31

11.32 Repeat Prob. 11.30 for a VDI orifice.

11.33 Repeat Prob. 11.31 for a VDI orifice.

11.34 An 8-in-diameter pipe carries 2.7 cfs of water at 72°F. Find the differential head across the following types of meters: (*a*) an 8-in by 4-in VDI orifice; (*b*) an 8-in by 4-in ISA flow nozzle; (*c*) an 8-in by 4-in venturi meter.

11.35 Air at 70°F and 104 psia flows through a venturi tube with a coefficient of 0.98. The pressure at the throat is 67.6 psia, the inlet area is 0.60 ft^2, and the throat area is 0.15 ft^2. Given k for the air is 1.40. (*a*) Using Eq. (11.22), find the ideal weight flow rate. (*b*) Evaluate Y from Fig. 11.25, and use it to find the actual weight flow rate. (*c*) What is the throat velocity?

11.36 Air at 20°C and 700 kPa abs flows through a venturi tube with a coefficient of 0.98. The pressure at the throat is 420 kPa abs, the inlet area is 0.060 m^2, and the throat area is 0.015 m^2. Given k for the air is 1.40. (*a*) Using Eq. (11.22), find the ideal weight flow rate. (*b*) Evaluate Y from Fig. 11.25, and use it to find the actual weight flow rate. (*c*) What is the throat velocity?

11.37 Using the same data as Prob. 11.35, what would be the value of Y and the weight flow rate for a square-edged orifice with $C = 0.59$?

11.38 Using the same data as Prob. 11.36, what would be the value of Y and the weight flow rate for a square-edged orifice with $C = 0.62$?

11.39 Air flows through a 200-mm by 100-mm venturi meter. At inlet the air temperature is 15°C and the gage pressure is 150 kPa. Determine the flow rate if a mercury manometer reads 180 mm. Assume an atmospheric pressure of 101.3 kPa abs.

11.40 Air flows through a 50-mm-diameter orifice from a tank at 1500 kPa abs and 40°C into a space where the pressure is 500 kPa abs. (*a*) Compute the weight flow rate assuming $C_d = 0.60$. Refer to Sec. 13.8. Repeat for external pressures of (*b*) 750, (*c*) 1000, and (*d*) 1250 kPa abs.

11.41 Using the Rehbock formula, plot a family of curves of C_d versus H/P with H as a parameter. These curves give a complete picture of the variation of C_d for sharp-crested rectangular weirs. Include H/P values of 0.1, 0.2, 0.5, 1.0, and 2.0, and H values of 0.2, 1.0, and 5.0 ft.

11.42 For the Cipolletti weir of Sec. 11.11, confirm that the side slopes of the trapezoid are $4V{:}1H$ by setting the reduction in discharge due to contraction equal to the increase in discharge due to the triangular area added.

11.43 Develop in general terms an expression for the percent of error in Q over a triangular weir if there is a small error in the measurement of the vertex angle. Assume there is no error in the weir coefficient. Compute the percent error in Q if there is a 1° error in the measurement of the total vertex angle of a triangular weir having a total vertex angle of 60°.

11.44 A 60° V-notch weir and a rectangular weir with suppressed end contractions (crest length = 2 ft) are both used to measure a flow rate of approximately 0.25 cfs. Assuming C_d is known to be minimum for each weir, compute the percentage of error in Q that would result from an error of 0.01 ft in the respective head measurements.

11.45 A 60° V-notch weir and a rectangular weir with end contractions (crest length = 0.6 m) are both used to measure a flow rate of approximately 7 L/s. Assuming C_d is known to be minimum for each weir, compute the percentage of error in Q that would result from an error of 5 mm in the respective head measurements.

11.46 Plot C_d versus H/P for broad-crested weirs using Eq. (11.31). Include H/P values of 0.1, 0.2, 0.5, 1.0, and 2.0, and H values of 0.2, 1.0, and 5.0 ft.

11.47 All the weir crests discussed in this chapter produce flow rates that vary as the head to some power greater than 1. In certain cases, such as in the outlet of a constant-velocity sedimentation chamber, it is desirable to employ a weir form in which Q varies directly with H. The proportional-flow weir is set flush with the bottom of the channel, as shown in Fig. P11.47, while the sides taper inward, following the hyperbola $x\sqrt{y} = k$, a constant. Commencing with the head $h = H - y$, on the element of area $dA = 2x\,dy = 2(k/\sqrt{y})dy$, prove that the discharge equation for such a weir may be written as $Q = C_d \pi k \sqrt{2g} H$, and evaluate k in terms of the width B and the velocity V in the rectangular approach channel.

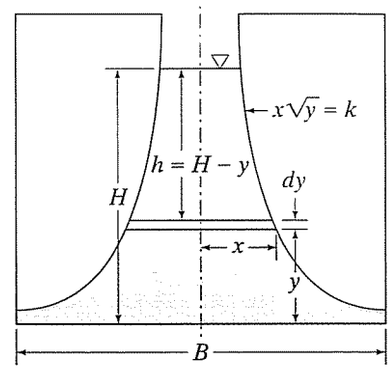


Figure P11.47

11.48 An 8-ft-wide rectangular channel contains a sluice gate which extends across the full width of the channel with an opening of 0.55 ft. Assuming $C_d = 0.60$, $C_c = 0.62$, and free flow, find the flow rate and the sluice coefficient when the upstream depth is 4.8 ft.

11.49 A 2.0-m-wide rectangular channel contains a sluice gate that extends across the full width of the channel with an opening of 0.15 m. Assuming $C_d = 0.60$, $C_c = 0.62$, and free flow, find the flow rate and the sluice coefficient when the upstream depth is 1.25 m.

11.50 Refer to Sample Prob. 5.12. (*a*) If $C_v = 0.98$, what is the flow rate? (*b*) If $C_c = 0.62$, what is the height of the opening? (*c*) What is K_s?

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

CHAPTER 12
Unsteady-Flow Problems

12.1 INTRODUCTION

This text deals mostly with steady flow, since the majority of cases of engineering interest are of this nature.[1] However, a number of cases of unsteady flow are very important, and some of them are discussed in this chapter. Earlier we learned that turbulent flow is unsteady in the strictest sense of the word, but if the mean temporal values are constant over a period of time, it is called mean steady flow. Here we shall consider cases where the mean temporal values continuously vary.

We shall investigate two main types of unsteady flow here. The first is where the water level in a reservoir or pressure tank is steadily rising or falling, so that the flow rate varies continuously, but where change takes place slowly. The second is where the velocity in a pipeline is changed rapidly by the fast closing or opening of a valve.

In the first case, of slow change, the flow is subject to the same forces as have previously been considered. Fast changes, of the second type, require the consideration of elastic forces.

Unsteady flow also includes such topics as oscillations in connected reservoirs and in U tubes and such phenomena as tidal motion and flood waves in open channels. Likewise, the field of machinery regulation by servomechanisms is intimately connected with unsteady motion. However, we shall not consider any of these topics here.

12.2 DISCHARGE WITH VARYING HEAD

When flow occurs under varying head, the rate of discharge will continuously vary. Let us consider the situation depicted in Fig. 12.1 in which $\mathcal{V}$ represents the volume of liquid contained in the tank at a particular instant of time. There is inflow at the rate Q_i and outflow at rate Q_o. The change in volume during a small time interval dt can be expressed as

$$d\mathcal{V} = Q_i dt - Q_o dt$$

[1] Where the unsteadiness is not too rapid, unsteady flow can usually be approximated by assuming the flow is steady at different rates over successive time periods of short duration.

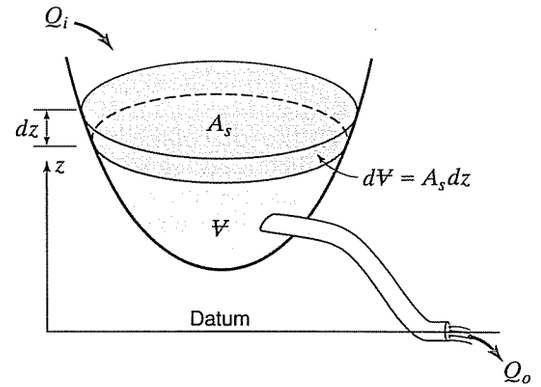


Figure 12.1

If A_s = area of the surface of the volume while dz is the change in level of the surface then $d\mathcal{V} = A_s dz$. Equating these two expressions for $d\mathcal{V}$,

$$A_s dz = Q_i dt - Q_o dt \tag{12.1}$$

Either Q_i or Q_o or both may be variable. The outflow Q_o is usually a function of z. For example, if liquid is discharged through an orifice or a pipe of area A under a differential head z, $Q_o = C_d A\sqrt{2gz}$, where C_d is a numerical discharge coefficient and z is a variable. If the liquid flows out over a weir or a spillway of length L, $Q_o = CLh^{3/2}$, where C is the appropriate coefficient and h is the head on the weir or spillway (Secs. 11.11–11.13). In either case z or h is the variable height of the liquid surface above the appropriate datum. The inflow Q_i commonly varies with time; however, such problems will not be considered here. We shall consider only the cases where $Q_i = 0$ or where Q_i = constant.

Rewriting Eq. (12.1) and integrating gives an expression for t, the time for the water level to change from z_1 to z_2. Thus

$$t = \int_{z_1}^{z_2} \frac{A_s dz}{Q_i - Q_o} \tag{12.2}$$

The right side of this expression can be integrated if Q_i is zero or constant and if A_s and Q_o can be expressed as functions of z. In the case of natural reservoirs, we cannot express the surface area as a simple mathematical function of z, but can obtain values of it from a topographic map. In such a case, we can solve Eq. (12.2) graphically by plotting values of $A_s/(Q_i - Q_o)$ against simultaneous values of z. The area under such a curve to some scale is the numerical value of the integral.

We must note here that instantaneous values for Q_o have been expressed in the same manner as for steady flow. This is not strictly correct, since for unsteady flow the energy equation should also include an acceleration head (see Eq. 12.6). The introduction of such a term renders the solution much more difficult. In cases where the value of z does not vary rapidly, no appreciable error will result if this acceleration term is disregarded. Therefore the equations can be written as for steady flow.

SAMPLE PROBLEM 12.1 The open wedge-shaped tank in Fig. S12.1 has a length of 15 ft perpendicular to the sketch. It is drained through a 3-in-diameter pipe 10 ft long whose discharge end is at elevation zero. The coefficient of loss at pipe entrance is 0.50, the total of the bend loss coefficients is 0.20, and f for the pipe is 0.018. Find the time required to lower the water surface in the tank from elevation 8 to 5 ft. Neglect the possible change of f with **R**, and assume that the acceleration effects in the pipe are negligible.

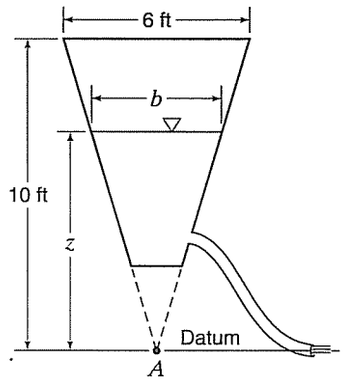


Figure S12.1

Solution

Energy Eq. (5.28) from water surface to jet at discharge:

$$0 + z + 0 - \left[0.5 + 0.2 + 0.018\left(\frac{10}{0.25}\right)\right]\frac{V^2}{2g} = 0 + 0 + \frac{V^2}{2g}$$

$$z - 1.420\frac{V^2}{2g} = \frac{V^2}{2g}; \quad V = 5.16z^{1/2}$$

$$Q_o = AV = \frac{\pi}{4}(0.25)^2 5.16z^{1/2} = 0.253z^{1/2}$$

Similar triangles: $$\frac{b}{z} = \frac{6}{10}; \quad \text{so} \quad b = 0.6z$$

So the area of the water surface is

$$A_s = 15b = 15(0.6)z = 9z$$

Eq. (12.2): $$t = \int_8^5 \frac{9z\,dz}{0 - 0.253z^{1/2}} = -\frac{9}{0.253}\int_8^5 z^{1/2}\,dz$$

$$= -35.5\left[\tfrac{2}{3}z^{3/2}\right]_8^5 = 271 \text{ sec} \qquad \textbf{\textit{ANS}}$$

Note: If the pipe had discharged at an elevation other than zero, the integral would have been different, because the head on the pipe would then have been $z + h$, where h is the vertical distance of the discharge end of the pipe below (h positive) or above (h negative) point A of the figure.

EXERCISES

12.2.1 (a) A ship lock has vertical sides. Water enters or leaves it through a conduit area of cross-sectional area A. The flow through this conduit is given by $Q = C_d A(2gz)^{1/2}$, where z is the variable difference in level between the water surface in the lock and that outside. Prove that the time required for the water level in the lock to drop from z_1 to z_2 is

$$t = \frac{2A_s}{C_d A\sqrt{2g}}\left(z_1^{1/2} - z_2^{1/2}\right)$$

(*Note:* If the lock is being filled, the signs must be reversed.) (b) Suppose the lock is 400 ft long by 100 ft wide, and the discharge coefficient for the conduit is 0.50. If the water surface in the lock is initially 40 ft below the level of the surface of the water upstream, how large must the conduit be if the lock is to be filled in 10 min?

12.2.2 (a) Repeat Exer. 12.2.1(a). (b) Suppose the lock is 100 m long by 30 m wide, and the discharge coefficient for the conduit is 0.50. If the water surface in the lock is initially 12 m below the level of the surface of the water upstream, how large must the conduit be if the lock is to be filled in 10 min?

12.2.3 The tank in Fig. X12.2.3 has the shape of the frustum of a cone with a 2-ft^2 orifice ($C_d = 0.62$) in the bottom. Given $D_1 = 44$ ft, $D_2 = 20$ ft, $z_0 = 36$ ft, and that the water level outside the tank is constant at section 2, how long will it take the water level in the tank to drop from section 1 to section 2? (*Note:* The tank diameter = ky, and $y = z + h_2$, where z is the variable distance between surface levels.)

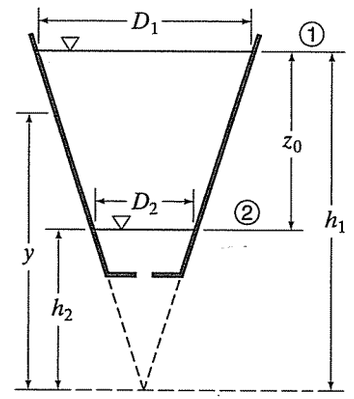


Figure X12.2.3

12.2.4 The cone-shaped tank in Fig. X12.2.3 has a 0.2-m^2 orifice ($C_d = 0.60$) in the bottom. Given $D_1 = 12$ m, $D_2 = 5.6$ m, $z_0 = 10.4$ m, and that the water level outside the tank is constant at section 2, how long will it take the water level in the tank to drop from section 1 to section 2? (*Note:* The tank diameter = ky, and $y = z + h_2$, where z is the variable distance between surface levels.)

12.2.5 Given the same tank and C_d as in Exer. 12.2.3, but now with the water surface outside constant at section 1 instead and the tank initially empty, how long will it take for the water level in the tank to rise from section 2 to section 1?

12.2.6 Given the same tank and C_d as in Exer. 12.2.4, but now with the water surface outside constant at section 1 instead and the tank initially empty, how long will it take for the water level in the tank to rise from section 2 to section 1?

12.2.7 The tank in Fig. X12.2.7 has vertical sides, $b_1 = 12$ ft, $b_2 = 24$ ft, $z_0 = 14$ ft, and its dimension normal to the plane of the paper is $a = 6$ ft. The vertical dividing plate has a submerged orifice 0.7 ft^2 in area ($C_d = 0.65$). How long will it take for the two water surfaces to equalize?

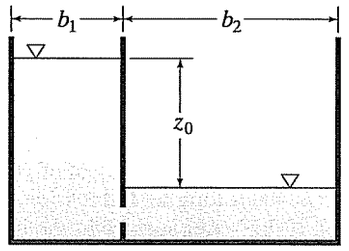


Figure X12.2.7

12.2.8 The tank in Fig. X12.2.7 has vertical sides, $b_1 = 3.6$ m, $b_2 = 7.2$ m, $z_0 = 4.2$ m, and its dimension normal to the plane of the paper is $a = 1.8$ m. The vertical dividing plate has a submerged orifice 0.063 m^2 in area ($C_d = 0.65$). How long will it take for the two water surfaces to equalize?

12.3 UNSTEADY FLOW OF INCOMPRESSIBLE FLUIDS IN PIPES

When the flow in a pipe is unsteady, we shall show that the energy equation has a term, the ***accelerative head*** $h_a = (L/g)(dV/dt)$, which accounts for the effect of the acceleration of the fluid. Let us consider an elemental length of the flow in a pipe, as in Fig. 12.2. We shall follow the same procedure we used in Sec. 8.4 by writing $\sum F = ma$; however, in this situation, with unsteady flow, at a particular point in the flow at a particular instant, we express the acceleration as $V(dV/ds) + dV/dt$. This comes from the general expression for acceleration in unsteady flow, Eq. (4.29). Applying $\sum F = ma$ to the cylindrical fluid element of Fig. 12.2, we get for unsteady flow

$$F_1 - F_2 - dW\cos\theta - \tau_0 A_s = ma$$

$$\text{i.e.,}\quad pA - (p + dp)A - \rho g A\, ds\left(\frac{dz}{ds}\right) - \tau_0(2\pi r)ds = \rho A\, ds\left(V\frac{dV}{ds} + \frac{dV}{dt}\right)$$

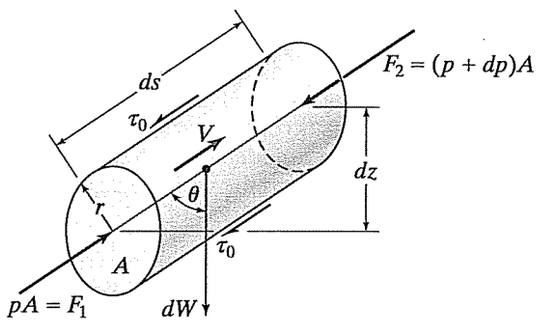


Figure 12.2
Element in pipe flow.

Dividing by $\gamma = \rho g$, dividing by $A = \pi r^2$, and simplifying, we get

$$-\frac{dp}{\gamma} - dz - \frac{2\tau_0 ds}{\gamma r} = \frac{V dV}{g} + \frac{ds}{g}\frac{dV}{dt} \tag{12.3}$$

Noting that $VdV = \frac{1}{2}d(V^2)$ this can be written

$$-\frac{dp}{\gamma} - dz - d\frac{V^2}{2g} - \frac{2\tau_0 ds}{\gamma r} - \frac{ds}{g}\frac{dV}{dt} = 0 \tag{12.4}$$

This equation applies to unsteady flow of both compressible and incompressible real fluids. However, for compressible fluids γ is a variable, and we must introduce a gas law equation relating γ to p and T before integration.

For incompressible fluids that we are considering here γ is a constant, so we can integrate directly. Integrating from some section 1 to another section 2, where the distance between them is L, we get

$$\frac{p_1 - p_2}{\gamma} + z_1 - z_2 + \frac{V_1^2 - V_2^2}{2g} - \frac{2\tau_0 L}{\gamma r} - \frac{L}{g}\frac{dV}{dt} = 0$$

or, noting from Sec. 8.3 that $R_h = D/4 = r/2$,

$$\left(\frac{p}{\gamma} + z + \frac{V^2}{2g}\right)_1 - \frac{\tau_0 L}{\gamma R_h} - \frac{L}{g}\frac{dV}{dt} = \left(\frac{p}{\gamma} + z + \frac{V^2}{2g}\right)_2 \tag{12.5}$$

But when there is no acceleration ($a = 0$), i.e., the flow is steady and V is constant, from Eq. (12.5) we see that the dV/dt term and the V^2 terms drop out, leaving

$$\left(\frac{p}{\gamma} + z\right)_1 - \frac{\tau_0 L}{\gamma R_h} = \left(\frac{p}{\gamma} + z\right)_2$$

which, by comparison with Eqs. (8.7) and (8.8), indicates that the term $\tau_0 L/\gamma R_h = h_{f_{1-2}}$, the pipe friction head loss over length L. Thus Eq. (12.5) becomes

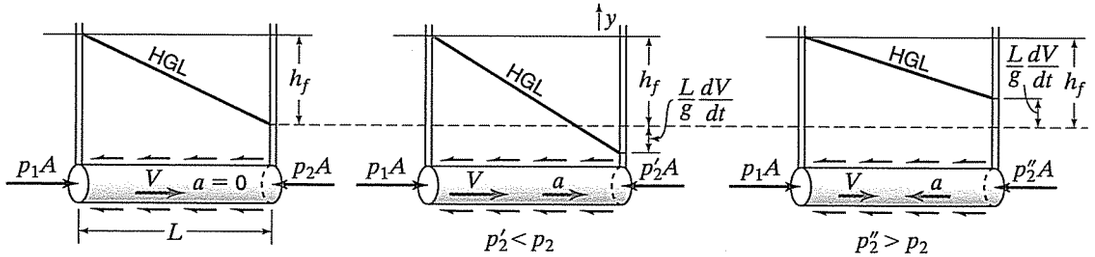
$$\left(\frac{p}{\gamma} + z + \frac{V^2}{2g}\right)_1 - h_f - \frac{L}{g}\frac{dV}{dt} = \left(\frac{p}{\gamma} + z + \frac{V^2}{2g}\right)_2 \tag{12.6a}$$

or

$$H_1 - h_f - h_a = H_2 \tag{12.6b}$$

This is the same as the steady flow Eq. (5.28), with the addition of the third term, the accelerative head $h_a = (L/g)(dV/dt)$. Equation (12.6) states that the energy loss between sections 1 and 2 is equal to (a) that required to overcome the friction plus (b) that required to produce the acceleration. We are presuming that the head loss at any instant is equal to the steady-flow head loss for the flow rate at that instant. Experimental evidence indicates that this presumption is reasonably valid.

If the pipe consists of two or more pipes in series, an $(L/g)(dV/dt)$ term for each pipe should appear in the equation just as there would be a separate term for the head loss in each pipe. To clarify the discussion further, the simple case of unsteady flow of an incompressible fluid in a horizontal pipe is shown in Fig. 12.3. The left sketch shows the steady-flow case, while unsteady flow is depicted in the



(*a*) Steady flow (*b*) Acceleration (*c*) Deceleration

(*a*)	(*b*)	(*c*)
$\Sigma F = ma = 0$	$\Sigma F = ma \neq 0$	$\Sigma F = ma \neq 0$
$p_1A - p_2A - \tau_0 PL = 0$	$p_1A - p_2'A - \tau_0 PL = \frac{\gamma AL}{g}\frac{dV}{dt}$	$p_1A - p_2''A - \tau_0 PL = \frac{\gamma AL}{g}\frac{dV}{dt}$
$\frac{p_1}{\gamma} - \frac{p_2}{\gamma} = \frac{\tau_0 PL}{\gamma A}$	$\frac{p_1}{\gamma} - \frac{p_2'}{\gamma} = \frac{\tau_0 PL}{\gamma A} + \frac{L}{g}\frac{dV}{dt}$	$\frac{p_1}{\gamma} - \frac{p_2''}{\gamma} = \frac{\tau_0 PL}{\gamma A} + \frac{L}{g}\frac{dV}{dt}$
$h_{f_{1-2}} = \frac{\tau_0 L}{\gamma R_h}$	$\frac{p_1}{\gamma} = \frac{p_2'}{\gamma} + h_f + \frac{L}{g}\frac{dV}{dt}$	$\frac{p_1}{\gamma} = \frac{p_2''}{\gamma} + h_f + \frac{L}{g}\frac{dV}{dt}$
	where dV/dt is positive	where dV/dt is negative

P = wetted perimeter; R_h = hydraulic radius.

Figure 12.3
Steady and unsteady flow of incompressible fluid in a horizontal pipe. (Flow is instantaneously equal in all three pipes.) (*a*) Steady flow ($dV/dt = 0$). (*b*) Unsteady flow (dV/dt is positive, p_2 is reduced). (*c*) Unsteady flow (dV/dt is negative, p_2 is increased).

two right sketches. The analysis below the sketches indicates that, with the same instantaneous flow rates, the pressure is depressed at section 2 if the acceleration is positive or increased if it is negative.

SAMPLE PROBLEM 12.2 Although the assumptions of instantaneous change in pump speed and head made in this example are unrealistic, it will serve to illustrate application of Eq. (12.6). When the centrifugal pump in Fig. S12.2 is rotating at 1650 rpm, the steady flow rate is 1600 gpm. Suppose the pump speed can be increased instantaneously to 2000 rpm. Determine the flow rate as a function of time. Assume that the head developed by the pump is proportional to the square of the rotative speed (see Sec. 15.4).

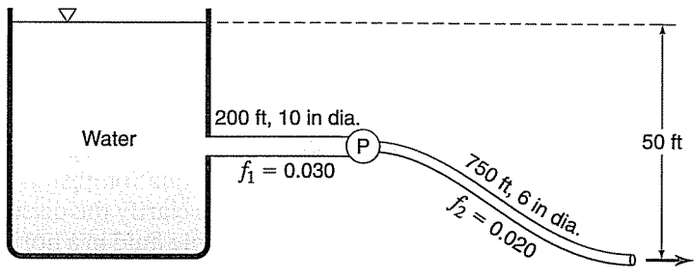


Figure S12.2

Solution
Writing the unsteady-flow energy equation from the water surface to the jet,

$$0 + 50 + 0 - 0.5\frac{V_1^2}{2g} - f_1\frac{L_1}{D_1}\frac{V_1^2}{2g} + h_p - f_2\frac{L_2}{D_2}\frac{V_2^2}{2g} - \frac{L_1}{g}\frac{dV_1}{dt} - \frac{L_2}{g}\frac{dV_2}{dt} = 0 + 0 + \frac{V_2^2}{2g}$$

where the subscripts 1 and 2 refer to the 10- and 6-in-diameter pipes, respectively. Note that the accelerative head for each pipe depends on the respective L and dV/dt values.

From continuity: $$V_1 = \frac{A_2V_2}{A_1} = \left(\frac{6}{10}\right)^2 V_2 = 0.36V_2$$

So $$\frac{dV_1}{dt} = \frac{A_2}{A_1}\frac{dV_2}{dt} = 0.36\frac{dV_2}{dt}$$

Thus $$50 - 0.5\frac{(0.36V_2)^2}{2g} - 0.030\left(\frac{200}{10/12}\right)\frac{(0.36V_2)^2}{2g} + h_p - 0.020\left(\frac{750}{6/12}\right)\frac{V_2^2}{2g} = \frac{V_2^2}{2g} + \frac{200}{g}(0.36)\frac{dV_2}{dt} + \frac{750}{g}\frac{dV_2}{dt}$$

Evaluating and combining terms: $$50 + h_p = 32.0\frac{V_2^2}{2g} + \frac{822}{g}\frac{dV_2}{dt} \qquad (1)$$

With the original steady-flow conditions ($dV/dt = 0$):

$$V_2 = \frac{Q}{A_2} = \frac{1600/449}{(\pi/4)(0.5)^2} = 18.16 \text{ fps}$$

and $$h_p = 32\frac{V_2^2}{2g} - 50 = 113.8 \text{ ft}$$

Given that $h_p \propto (\text{rpm})^2$, so after the speed is increased to 2000 rpm

$$h_p = 113.8\left(\frac{2000}{1650}\right)^2 = 167.2 \text{ ft}$$

Substituting into (1): $$50 + 167 = 32\frac{V_2^2}{2g} + \frac{822}{g}\frac{dV_2}{dt}$$

Expressing this in terms of Q, using $V_2 = Q/A_2 = 5.09Q$:

$$217 = 12.89Q^2 + 130.0\frac{dQ}{dt} \qquad (2)$$

Solving (2) for dt and integrating, noting that at $t = 0$, $Q = 3.57$ cfs (1600 gpm):

$$\int_0^t dt = 130 \int_{3.57}^{Q} \frac{dQ}{217 - 12.89Q^2}$$

$$t = 1.229 \ln \frac{4.10 + Q}{4.10 - Q} - 3.27$$

$$e^{0.814t+2.66} = \frac{4.10 + Q}{4.10 - Q}$$

Finally

$$Q = 4.10 \frac{e^{0.814t+2.66} - 1}{e^{0.814t+2.66} + 1} \qquad \textit{ANS}$$

Notes:

1. As t gets larger, Q approaches 4.10 cfs (1840 gpm), the steady-state flow rate for the condition where $h_p = 169$ ft.
2. We cannot instantaneously change the speed of a pump from one value to another, as we assumed in this example. To solve this problem correctly, we would have to know the operating characteristics of the pump and motor and the moment of inertia of the rotating system.

EXERCISES

12.3.1 Verify that the neglect of the accelerative head was justified in Sample Prob. 12.1. Find its magnitude at $z = 5$ ft from values of V and t at $z = 4.5$ and 5.5 ft.

12.3.2 Assume the speed of the pump in Sample Prob. 12.2 is reduced instantaneously from 1650 to 1150 rpm. What then is the deceleration of the flow rate immediately after the change in pump speed?

12.4 APPROACH TO STEADY FLOW

Determining the time for the flow to become steady in a pipeline when a valve is suddenly opened at the end of the pipe can be accomplished through application of Eq. (12.6). Immediately after the valve is opened (Fig. 12.4), the head h is available to accelerate the flow. Thus flow commences, but as the velocity increases the accelerating head is reduced by fluid friction and minor losses. Let us assume the total head loss h_L can be expressed as $k_L V^2/2g$, where k_L is constant, although it may vary somewhat with velocity unless the pipe is very rough. Then

$$k_L = f\frac{L}{D} + \sum k$$

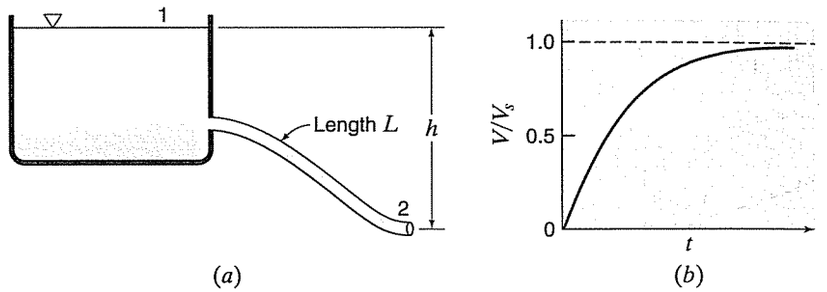


Figure 12.4
Approach to steady flow (V_s = velocity at steady flow).

where $\sum k$ is the sum of the coefficients of all minor losses in the pipeline (Secs. 8.20–8.26). Writing Eq. (12.6) between sections 1 and 2 in Fig. 12.4*a*, while noting that $p_1 = p_2 = V_1 = 0$ and $z_1 - z_2 = h$, gives

$$h - \frac{V_2^2}{2g} = h_L + \frac{L}{g}\frac{dV}{dt}$$

or

$$h = \left(\frac{fL}{D} + \sum k\right)\frac{V^2}{2g} + \frac{V^2}{2g} + \frac{L}{g}\frac{dV}{dt}$$

i.e.,

$$h = K\frac{V^2}{2g} + \frac{L}{g}\frac{dV}{dt}, \quad \text{where} \quad K = \frac{fL}{D} + 1 + \sum k$$

For the free discharge at point 2 in Fig. 12.4, note that in the expression for $K = k_L + 1$ the 1 represents the residual velocity head and $\sum k$ does *not* include a submerged discharge loss coefficient $k_d = 1$ (Sec. 8.22). If, however, there is submerged discharge into still water in another reservoir, there is no residual velocity head so that $K = k_L$, but $\sum k$ now *does* include $k_d = 1$. As a result, the value of K is the same for both situations if minor losses are included. It is not quite the same if minor losses are *neglected* (see Sec. 8.27), because the residual velocity head is not considered a minor loss.

Let us represent the steady-flow velocity by V_s. Noting that for steady flow $(dV/dt) = 0$, we get

$$V_s = \sqrt{\frac{2gh}{K}} \tag{12.7}$$

Substituting the value of h from this expression into the above energy equation gives

$$dt = \frac{2L}{K}\frac{dV}{V_s^2 - V^2}$$

Treating f and therefore K as constants, we can integrate. Noting that the constant of integration = zero, since $V = 0$ at $t = 0$ and $\ln(V_s/V_s) = 0$, we get

$$t = \frac{L}{KV_s} \ln \frac{V_s + V}{V_s - V} \tag{12.8}$$

This equation indicates that V approaches V_s asymptotically and that equilibrium will be attained only after an infinite time (Fig. 12.4*b*), but it must be remembered that this is an idealized case. In reality there will be elastic waves and damping, so that true equilibrium will be reached in a finite time. Also, as noted earlier, we can apply Eqs. (12.7) and (12.8) to a submerged discharge as well as to a free discharge.

SAMPLE PROBLEM 12.3 Two large water reservoirs are connected to one another with a 100-mm-diameter pipe ($f = 0.02$) of length 15 m. The water-surface elevation difference between the reservoirs is 2.0 m. A valve in the pipe, initially closed, is suddenly opened. (*a*) Determine the times required for the flow to reach $\frac{1}{4}$, $\frac{1}{2}$, and $\frac{3}{4}$ of the steady-state flow rate. Assume the water-surface elevations remain constant. (*b*) Repeat for pipe lengths of 150 m and 1500 m with all other data remaining the same. Assume a square-edged entrance.

Solution

(*a*) For $L = 15$ m, square-edged entrance, and submerged discharge: $L/D = 15/0.10 = 150$, so minor losses are significant (Sec. 8.27). $\sum k = 0.5 + 1.0 = 1.5$

$$K = k_L = \frac{fL}{D} + \sum k = 0.02 \frac{15}{0.10} + 1.5 = 4.5$$

For steady flow:

Eq. (12.7): $$V_s = \sqrt{\frac{2gH}{K}} = \sqrt{\frac{2(9.81)2}{4.5}} = 2.95 \text{ m/s}$$

For unsteady flow use Eq. (12.8):

$$t = \frac{L}{KV_s} \ln \frac{V_s + V}{V_s - V} = \frac{15}{4.5(2.95)} \ln \frac{2.95 + V}{2.95 - V} = 1.129 \ln \frac{2.95 + V}{2.95 - V}$$

For $Q = \frac{1}{4}Q_s$ substitute $V = \frac{1}{4}V_s$, etc.:

Q	V, m/s	$\frac{2.95 + V}{2.95 - V}$	ln	For $L = 15$ m t, s
$0.25Q_s$	0.74	1.667	0.511	0.577
$0.50Q_s$	1.48	3.00	1.099	1.240
$0.75Q_s$	2.21	7.00	1.946	2.197

ANS

(*b*) For the other two lengths the minor losses are insignificant (Sec. 8.27), and the results are as follows:

Q	For $L = 150$ m	For $L = 1500$ m
$0.25Q_s$	2.18 s	7.04 s
$0.50Q_s$	4.69 s	15.15 s
$0.75Q_s$	8.30 s	26.84 s

ANS

EXERCISES

12.4.1 A 4-in-diameter drain pipe (flush entrance, $f = 0.020$) 3000 ft long will freely discharge reservoir water at an elevation 100 ft below the reservoir water surface. Initially there is no flow, since there is a plug in the pipe outlet. What will be the steady flow velocity in this pipe, and how long after the plug is removed will the flow velocity be half the steady velocity? See Sec. 8.27 regarding the neglect of minor losses.

12.4.2 A 150-mm-diameter drain pipe (flush entrance, $f = 0.030$) 500 m long will freely discharge reservoir water at an elevation 60 m below the reservoir water surface. Initially there is no flow, because a valve at the outlet end is closed. What will be the steady flow velocity in this pipe, and how long after the valve is suddenly opened will the flow velocity be half the steady velocity? See Sec. 8.27 regarding the neglect of minor losses.

12.4.3 A 250-mm-diameter pipe (flush entrance, $f = 0.026$) 1000 m long can deliver water from a higher reservoir to a lower one (submerged discharge) (Fig. X12.4.3). The difference between the two water-surface elevations is 75 m. After a valve near the downstream end is suddenly opened, what will be the steady flow velocity in this pipe, and how long after the valve is opened will the flow velocity be one-third and two-thirds of the steady velocity?

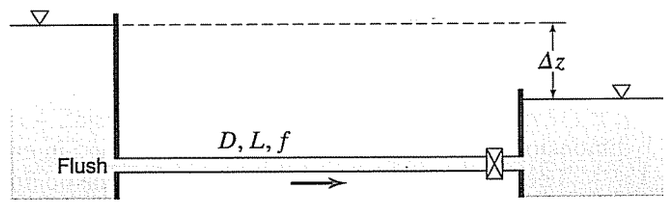


Figure X12.4.3

12.4.4 A 4-in-diameter drain pipe (flush entrance, $f = 0.022$) 300 ft long will freely discharge reservoir water at an elevation 40 ft below the reservoir water surface. Initially there is no flow, since there is a plug in the pipe outlet. What will be the steady flow velocity in this pipe, and how long after the plug is removed will the flow velocity be 50% and 75% of the steady velocity?

12.4.5 A 6-in-diameter pipe (reentrant entrance, $f = 0.028$) 475 ft long can deliver water from a higher reservoir to a lower one (submerged discharge). The difference between the two water-surface elevations is 25 ft. After a valve near

the downstream end is suddenly opened, what will be the steady flow velocity in this pipe, and how long after the valve is opened will the flow velocity be 90% of the steady velocity?

12.4.6 A 150-mm-diameter pipe (reentrant entrance, $f = 0.023$) 140 m long can deliver water from a higher reservoir to a lower one (submerged discharge). The difference between the two water-surface elevations is 12 m. After a valve near the downstream end is suddenly opened, what will be the steady flow velocity in this pipe, and how long after the valve is opened will the flow velocity be 90% of the steady velocity?

12.5 VELOCITY OF PRESSURE WAVE IN PIPES

Unsteady phenomena, with rapid changes taking place, frequently involve the transmission of pressure in waves or surges. As shown in Sec. 13.3, the velocity or ***celerity*** of a pressure (sonic) wave is

$$c = \sqrt{\frac{E_v}{\rho}} = \sqrt{\frac{g}{\gamma}E_v} \tag{12.9}$$

where E_v is the volume (or bulk) modulus of elasticity of the medium. For water, a typical value of E_v is 300,000 psi (2.07×10^6 kN/m^2), and thus the velocity (or celerity) c of a pressure wave in water is about 4720 fps (1440 m/s), depending on the temperature.

For water in an elastic pipe, this velocity is reduced due to the stretching of the pipe walls. As explained in Sec. 13.3, E_v is replaced by the joint (or combined) modulus E_j, such that

$$E_j = \frac{E_v}{1 + \dfrac{D}{t}\dfrac{E_v}{E}} = \frac{1}{\dfrac{1}{E_v} + \dfrac{D}{tE}}$$

where D and t are the diameter and wall thickness of the pipe, respectively, and E is the modulus of elasticity of the pipe material. As the ratios D/t and E_v/E are dimensionless, any consistent units may be used in each.

The velocity of a pressure wave in an elastic fluid inside an elastic pipe is then

$$c_j = \sqrt{\frac{E_j}{\rho}} = \sqrt{\frac{g}{\gamma}E_j} = \frac{c}{\sqrt{1 + \dfrac{D}{t}\dfrac{E_v}{E}}} = \sqrt{\frac{1}{\rho\left(\dfrac{1}{E_v} + \dfrac{D}{tE}\right)}} \tag{12.10}$$

The subscript j, used for water inside a pipe, corresponds to that used in E_j. Values of the modulus of elasticity E for steel, cast iron, and concrete are about

30,000,000, 15,000,000, and 3,000,000 psi, respectively.[2] Values of the volume (bulk) modulus E_v for various liquids are given in Appendix A, Table A.4, and for water in Sec. 2.5 and Table A.1.

For normal pipe dimensions the speed c_j of a pressure wave in a water pipe usually ranges between 2000 and 4000 fps (600 and 1200 m/s), but it will always be less than $c \approx 4720$ fps (1440 m/s), the velocity of a pressure wave in free water. Note, e.g., from Table A.1, that both these velocities vary with temperature, by up to 7.5%.

EXERCISES

12.5.1 (*a*) Calculate the celerity of a pressure wave in open fresh water at 70°F. (*b*) What will the celerity be if the water is contained in a cast-iron pipe with 12.00-in inside diameter and $\frac{3}{8}$-in wall thickness? (*c*) Express the celerity in the pipe as a percentage of the celerity in the open water.

12.5.2 (*a*) Calculate the celerity of a pressure wave in open fresh water at 20°C. (*b*) What will the celerity be if the water is contained in a cast-iron pipe with 489-mm inside diameter and 14.3-mm wall thickness? (*c*) Express the celerity in the pipe as a percentage of the celerity in the open water.

12.5.3 Find the celerity of a pressure wave in benzene (Appendix A, Table A.4) contained in an 8-in-diameter steel pipe having a wall thickness of 0.32 in.

12.5.4 Find the celerity of a pressure wave in benzene (Appendix A, Table A.4) contained in a 200-mm-diameter steel pipe having a wall thickness of 8.0 mm.

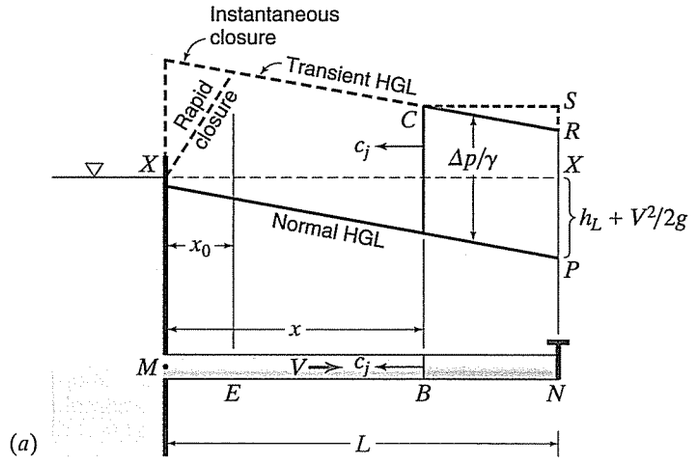
12.6 WATER HAMMER

In the preceding unsteady-flow cases in this chapter, the changes of velocity were presumed to take place slowly. But if the velocity of a liquid in a pipeline is abruptly decreased by a valve movement, the phenomenon encountered is called ***water hammer.*** This is a very important problem in the case of hydroelectric plants, where the flow of water must be rapidly varied in proportion to the load changes on the turbine. Water hammer has burst large penstocks, causing great damage to hydraulic and power generating facilities, in addition to loss of life and power generation. Water hammer occurs in liquid-flow pressure systems whenever a valve is closed or opened, fully or partially. The terminology "water hammer" is perhaps misleading, since this phenomenon can occur with any liquid.

Instantaneous Closure

Although it is physically impossible to close a valve instantaneously, such a concept is useful as an introduction to the study of real cases. For convenience let us start by considering steady flow in a horizontal pipe (Fig. 12.5*a*) with a partly

[2] Corresponding values of E for steel, cast iron, and concrete in SI units are 207×10^6, 103×10^6, and 20.7×10^6 kN/m^2, respectively.



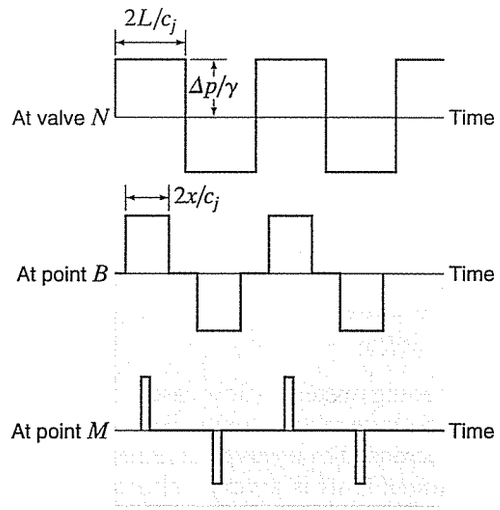


Figure 12.5
Water hammer with pipe friction and damping neglected.
(*a*) Valve *N* at the end of the pipeline is abruptly closed and the pressure wave has traveled part way up the pipe.
(*b*) Idealized water-hammer pressure heads at *N*, *B*, and *M* as a function of time for instantaneous valve closure.

open valve. Then let us assume that the valve at *N* is closed completely and instantaneously. The lamina of liquid next to the valve is compressed by the rest of the column of liquid flowing against it. At the same time the walls of the pipe surrounding this lamina are stretched by the excess pressure produced; in this way the kinetic energy of the moving water transforms into these two forms of potential (elastic) energy. The next upstream lamina will then be brought to rest, and so on. The liquid in the pipe does not behave as a rigid incompressible body, but the phenomenon is affected by the elasticity of both the liquid and the pipe.

The cessation of flow and the resulting pressure increase move upstream together along the pipe as a wave with the velocity c_j as given by Eq. (12.10).

After a short interval of time the liquid column *BN* will have been brought to rest at an increased pressure, while the liquid in the length *MB* will still be flowing with its initial velocity and initial pressure. When the pressure wave finally reaches the inlet at *M*, the entire mass in the length *L* will be at rest but will be under an excess pressure throughout. During travel of the pressure wave from *N* to *M*, there will be a transient hydraulic grade line parallel to the original steady-flow grade line *XP* but at a height $\Delta p/\gamma$ above it, where Δp represents the water hammer pressure.

It is impossible for a pressure to exist at *M* that is greater than that due to depth *MX*, and so when the pressure wave arrives at *M*, the pressure at *M* drops instantly to the value it would have for zero flow. But the entire pipe is now under an excess pressure; the liquid in it is compressed, and the pipe walls are stretched. So some liquid starts to flow back into the reservoir and a wave of pressure unloading travels along the pipe from *M* to *N*. Assuming there is no damping, at the instant this unloading wave reaches *N*, the entire mass of liquid will be under the normal pressure indicated by the line *XP*, but the liquid is still flowing back into the reservoir. This reverse velocity will provide a suction or pressure drop at *N* that ideally will be as far below the normal, steady-flow pressure as the pressure an instant before was above it. Then a wave of low pressure or suction travels back up the pipe from *N* to *M*. Ideally, there would be a series of pressure waves traveling back and forth over the length of the pipe and alternating equally between high and low pressures. Actually, because of damping due to fluid friction and imperfect elasticity of liquid and pipe, the pressure extremes at any point in the pipe will gradually tend toward the pressure for the no-flow condition indicated by *XX* in Fig. 12.5*a*.

The time for a round trip of the pressure wave from *N* to *M* and back again is

$$T_r = 2\frac{L}{c_j} \tag{12.11}$$

where *L* is the actual length of pipe, and not the length adjusted for minor losses as described in Sec. 8.20. So for an instantaneous valve closure the excess pressure at the valve remains constant for this length of time T_r, before it is affected by the return of the unloading pressure wave; and in like manner the pressure defect during the period of low pressure remains constant for the same length of time. At a distance *x* from the inlet, such as at *B*, the time for a round trip of a pressure wave is only $2x/c_j$, and so at that point the time duration of the excess or deficient pressure will be $2x/c_j$, as shown in Fig. 12.5*b*. At the inlet *M*, where $x = 0$, the excess pressure occurs for only an instant. Thus we find at points closer to the inlet *M* that the duration of the excess or deficient pressure is less, but its magnitude is the same.

We can calculate the pressure change caused by water hammer as follows. In Fig. 12.6 we see a close-up in the vicinity of the valve. If the valve is closed abruptly, but not necessarily completely, a pressure wave travels up the pipe with a celerity c_j. In a short interval of time Δt an element of liquid of length $c_j \Delta t$ is decelerated.

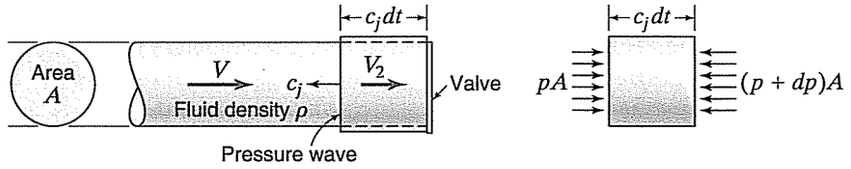


Figure 12.6
Definition sketch for analysis of water hammer in pipes.

Therefore the mass m decelerated in the same time is $m = \rho\forall = \rho A(c_j \Delta t)$. Applying Newton's second law, $F\Delta t = m\Delta V$, recalling from Sec. 6.1 that $\Delta V = V_{out} - V_{in} = V_2 - V_1$, and writing the initial V_1 as V and neglecting friction, we get

$$[pA - (p + \Delta p)A]\Delta t = (\rho A c_j \Delta t)(V_2 - V)$$

from which

Instantaneous, partial or complete closure:
$$\Delta p = \rho c_j (V - V_2) \tag{12.12}$$

which indicates the change in pressure Δp that results from an instantaneous change in flow velocity $(V - V_2)$. Partially closing a valve will affect the size of $(V - V_2)$, and Eq. (12.12) indicates that the pressure change is proportional to the velocity change.

We may also eliminate c_j between Eqs. (12.10) and (12.12) to obtain

Instantaneous, partial or complete closure:
$$\Delta p = (V - V_2)\sqrt{\rho E_j} = (V - V_2)\sqrt{\frac{\rho}{\left(\frac{1}{E_v} + \frac{D}{tE}\right)}} \tag{12.13}$$

We see that the pressure increase is independent of the length of the pipe and depends solely upon the celerity of the pressure wave in the pipe and the change in the velocity of the water. The total pressure at the valve immediately after closure is $\Delta p + p$, where p is the pressure in the pipe just upstream of the valve prior to closure.

In the case of instantaneous and *complete* closure of a valve the velocity is reduced from V to zero, that is, $V_2 = 0$. Substituting this into Eqs. (12.12) and (12.13), we get

Instantaneous, *complete* closure:
$$\Delta p = \rho c_j V = V\sqrt{\frac{\rho}{\left(\frac{1}{E_v} + \frac{D}{tE}\right)}} \tag{12.14}$$

Consider now conditions at the valve as affected by both pipe friction and damping. In Fig. 12.5*a*, when the pressure wave from N has reached B, the water in BN will be at rest and for zero flow the hydraulic grade line CR should be a

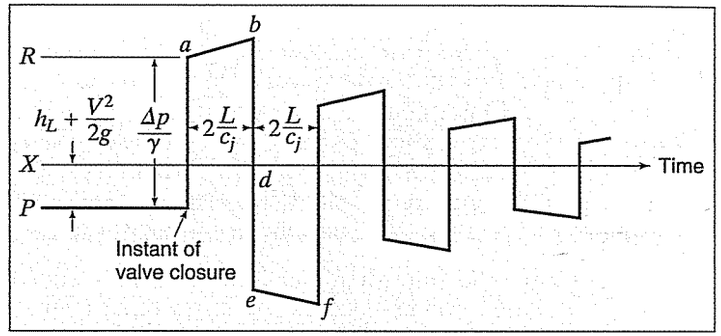


Figure 12.7
Pressure history at the valve N of Fig. 12.5 caused by instantaneous complete closure, considering pipe friction and damping.

horizontal line. There is thus a tendency for the grade line to flatten out for the portion BN. Hence, instead of the transient gradient having the slope imposed by friction, as shown in the figure, it will approach the horizontal line CS as the pressure at C cannot change. Thus the pressure head at N will be raised to a slightly higher value than NR shortly after the valve closure.

This slight increase in pressure head at the valve over the theoretical value $c_j(V - V_2)/g$ has been borne out by tests. As the wave front (B in Fig. 12.5*a*) moves toward M, C moves with it and so RS gets larger. As a result, on the diagram of the pressure history at the valve in Fig. 12.7, the line ab is shown as sloping upward, and for the same reason ef may slope slightly downward, as all conditions are now reversed. Also, because of damping, the waves will be of decreasing amplitude until the final equilibrium pressure is reached.

All of the preceding analysis assumes that the wave of low pressure (suction) will not cause the minimum pressure at any point to drop down to or below the vapor pressure. If it should do so, the water would separate and produce a discontinuity. When the pressure recovers and the water rushes back into the vapor cavity, its momentum and the resulting implosion on the valve could be very damaging.

SAMPLE PROBLEM 12.4 Water flows at 10 fps in a 400-ft-long steel pipe of 8-in diameter with 0.25-in-thick walls. Calculate the pulse interval and maximum pressure rise theoretically caused by instantaneously closing the end valve (*a*) completely and (*b*), partially, reducing the velocity to 6 fps.

Solution

Eq. (12.10):
$$c_j = \frac{c}{\sqrt{1 + \frac{D}{t}\frac{E_v}{E}}} = \frac{4720}{\sqrt{1 + \frac{8}{0.25}\left(\frac{3\times10^5}{3\times10^7}\right)}} = 4110 \text{ fps}$$

Eq. (12.11): $$T_r = \frac{2(400)}{4110} = 0.195 \text{ sec} \qquad \textit{ANS}$$

(*a*) Eq. (12.14): $$\Delta p = \rho c_j V = \frac{62.4}{32.2}(4110)10 = 79{,}600 \text{ psf} \qquad \textit{ANS}$$
$$= 553 \text{ psi}$$

for which $\Delta p/\gamma = 79{,}600/62.4 = 1276$ ft of water

For this pipe, the water-hammer impact initially equivalent to the pressure of 1276 ft of water (553 psi) occurs about five times every second!

(*b*) With partial closure, c_j and T_r are unchanged.

Eq. (12.12): $$\Delta p = \frac{62.4}{32.2}(4110)(10 - 6) = 31{,}800 \text{ psf} \qquad \textit{ANS}$$

Rapid Closure ($t_c < T_r$)

It is physically impossible for a valve to be closed instantaneously; so we shall now consider the real case where the valve is closed, completely or partially, in a finite time t_c that is more than zero but less than $T_r = 2L/c_j$. Figure 12.8 shows actual pressure recordings, made at the valve, for such a case.[3] The slope of the

[3] Figures 12.8 and 12.9 are from water-hammer studies made by the Southern California Edison Co. on an experimental pipe with the following data: $L = 3060$ ft, internal diameter = 2.06 in, $c_j = 4371$ fps, $V = 1.11$ fps, $\Delta p/\gamma = 150.7$ ft, $T_r = 1.40$ sec, static head = 306.7 ft, head before valve closure = 301.6 ft, $h_L = 5.1$ ft. In Fig. 12.8 the time of complete closure $t_c = 1$ sec and we see that the actual rise in pressure head is slightly more than 151 ft. In Fig. 12.9 the time of closure = 3 sec.

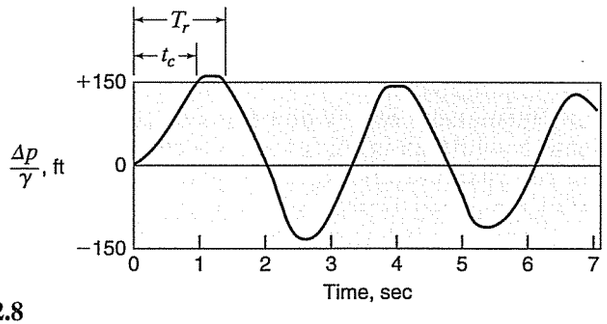


Figure 12.8
Rapid complete valve closure in time t_c (1 sec) less than T_r (1.40 sec). Actual measurements of pressure changes at the valve. (See footnote 3.)

curve during the time t_c depends entirely on the operation of the valve and its varying effect on the velocity in the pipe. But the maximum pressure rise is still the same as for instantaneous closure. The only differences are that it endures for a shorter period of time and the vertical lines of Fig. 12.7 are changed to the sloping lines of Fig. 12.8. The duration of the maximum pressure is $T_r - t_c$ (Fig. 12.8). If the time of valve closure were exactly T_r, the maximum pressure rise at the valve would still be the same, but the curves in Fig. 12.8 would all end in sharp points for both maximum and minimum values, since the time duration of maximum pressure would be reduced to zero.

Earlier we noted that at distance x from the inlet, such as B in Fig. 12.5, the time for a round trip of a pressure wave is $2x/c_j$. Therefore, no matter how rapid the valve closure, since t_c cannot be zero there must be some distance x_0 such that $t_c = 2x_0/c_j$. Clearly

$$x_0 = \frac{c_j t_c}{2} \tag{12.15}$$

Thus, at the point E in Fig. 12.5, a distance x_0 from the inlet, the time for a round trip of a pressure wave is exactly t_c. At points closer to the inlet ($x < x_0$) this travel time is shorter. But, as we see in Fig. 12.8, it takes time t_c for the pressure at a point to rise from zero to its maximum value. Therefore, at points nearer to the inlet than E, the reflected relief (unloading) pressure wave front will arrive before the maximum pressure is achieved. This effect will reduce the maximum water-hammer pressure at points where $x < x_0$. Nearer the inlet, at smaller x, the reflected wave front arrives sooner and the reduction is greater. As a result, in any real case the maximum water-hammer pressure experienced will be Δp (Eqs. 12.12–12.14) at all points where $x > x_0$, and it will decrease from Δp at x_0 to zero at the inlet. The maximum pressure rise Δp cannot extend all the way to the reservoir intake. This modification of the transient hydraulic grade line is labeled "rapid closure" on Fig. 12.5. A linear variation of maximum transient pressure within x_0 is commonly assumed, and although this appears reasonable from Fig. 12.8 (for $t < t_c$), the actual shape of this curve depends on how the valve is operated.

Slow Closure ($t_c > T_r$)

The preceding discussion assumes a closure so rapid (or a pipe so long) that there is an insufficient time for a pressure wave to make the round trip before the valve is closed. We will define slow closure as one in which the time of valve movement is greater than $T_r = 2L/c_j$. In this case the maximum pressure rise will be less than in the preceding, because the wave of pressure unloading will reach the valve before the valve is completely closed. This will prevent any further increase in pressure.

Thus in Fig. 12.9 the pressure at the valve would continue to rise if it were not for the fact that at time T_r a return unloading pressure wave reaches the valve and stops the pressure rise at a value of about 53 ft as contrasted with

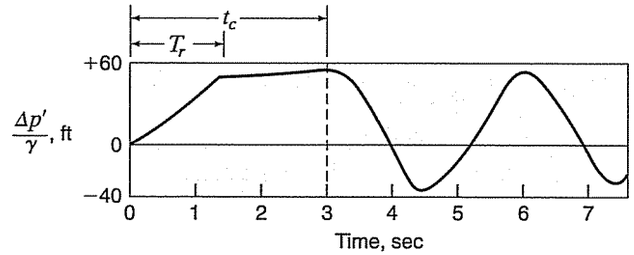


Figure 12.9
Slow complete valve closure in time t_c (3 sec) greater than T_r (1.40 sec). Actual measurements of pressure changes at the valve. (See footnote 3.)

nearly three times that value in Fig. 12.8.[4] Viewed another way, this is equivalent to saying that $x_0 > L$, or the valve is to the left of E in Fig. 12.5, so the entire pipeline only experiences maximum transient pressures which vary approximately linearly for $x < x_0$, and the maximum value given by Eqs. (12.12)–(12.14) is never achieved.

Tests have shown that for slow valve closure, i.e., in a time greater than T_r, the excess pressure produced decreases uniformly from the value at the valve to zero at the intake. The maximum water-hammer pressure $\Delta p'$ developed by gradual closure of a valve when $t_c > T_r$ is given approximately by

$$\frac{\Delta p'}{\Delta p} \approx \frac{L}{x_0}$$

from which

Slow, partial or complete closure:

$$\Delta p' \approx \frac{L}{x_0}\Delta p = \frac{2L}{c_j t_c}\Delta p = \frac{L\rho c_j(V - V_2)}{x_0} = \frac{2L\rho(V - V_2)}{t_c} \qquad (12.16)$$

where t_c is the time of closure.

After an unloading pressure wave reaches the valve and the maximum pressure $\Delta p'$ is reached, elastic waves travel back and forth and pressure changes are very complex. They require a detailed step-by-step analysis that is beyond the scope of this text.[5] In brief, the method consists of assuming the valve movement to take place in a series of steps each of which produces a pressure Δp proportional to each ΔV.

[4] See footnote 3.
[5] See, for example, E. B. Wylie and V. L. Streeter, *Fluid Transients in Systems,* Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1993.

SAMPLE PROBLEM 12.5 Assume that the elasticity and dimensions of the pipe in Fig. 12.5*a* are such that the celerity of the pressure wave is 3200 fps. Suppose that the pipe has a length of 2000 ft and a diameter of 4 ft. Find the maximum water hammer pressure at the valve, and the approximate maximum water hammer pressure at a point 300 ft from the reservoir, (*a*) if the valve is partially closed in 2.2 sec, thereby reducing the flow of water from 30 to 10 cfs; and (*b*) if water is initially flowing at 15 cfs and the valve is completely closed in 1.0 sec.

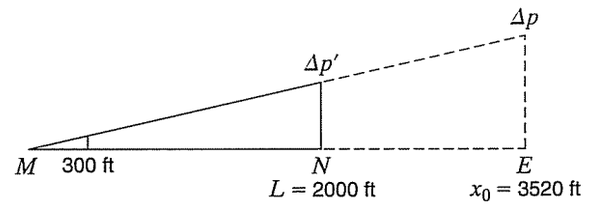
Solution

$$A = \pi(2)^2 = 12.57 \text{ ft}^2$$

Eq. (12.11):
$$T_r = \frac{2(2000)}{3200} = 1.25 \text{ sec}$$

(*a*) $t_c = 4.0$ sec $> T_r$, so closure is "slow."

Eq. (12.15):
$$x_0 = 0.5(3200)2.2 = 3520 \text{ ft}$$



$$V = Q/A = 30/12.57 = 2.39 \text{ fps}; \quad V_2 = 10/12.57 = 0.796 \text{ fps}$$

Eq. (12.12):
$$\Delta p = \frac{62.4}{32.2}(3200)(2.39 - 0.796) = 9870 \text{ psf} = 68.5 \text{ psi}$$

At the valve N ($x < x_0$),

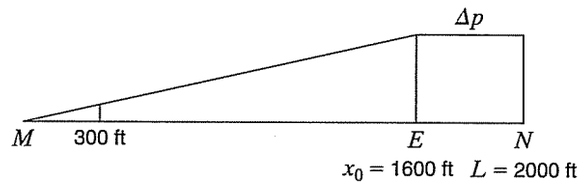
Eq. (12.16):
$$\Delta p' = \frac{2000}{3520}(68.5) = 38.9 \text{ psi} \quad \textbf{\textit{ANS}}$$

At the point $x = 300$ ft from the reservoir, assuming a linear variation,

$$(\Delta p)_x = \frac{300}{2000}(38.9) = 5.84 \text{ psi} \quad \textbf{\textit{ANS}}$$

(*b*) $t_c = 1.0$ sec $< T_r$, so closure is "rapid."

Eq. (12.15):
$$x_0 = 0.5(3200)1.0 = 1600 \text{ ft}$$



$$V = Q/A = 15/12.57 = 1.194 \text{ fps}$$

At the valve N ($x > x_0$),

Eq. (12.14): $$\Delta p = \frac{62.4}{32.2}(3200)1.194 = 7400 \text{ psf}$$

$$= 51.4 \text{ psi} \quad \textbf{\textit{ANS}}$$

At the point $x = 300$ ft from the inlet, assuming a linear variation within x_0,

$$(\Delta p)_x = \frac{300}{1600}(51.4) = 9.64 \text{ psi} \quad \textbf{\textit{ANS}}$$

Computer Techniques for Water Hammer

A stricter analysis of water hammer, based on Newton's second law of motion and continuity, results in two nonlinear partial differential equations each in two unknowns, V (or Q) and p (or p/γ), which are functions of x and t. These equations can take into account pipe friction and the slope of the pipe. Although there is no general solution to these equations, they can be solved by the ***method of characteristics*** using a finite difference solution on a computer. The two partial differential equations are combined in two particular ways to convert them into two ordinary differential equations, which we can write in finite difference form and solve simultaneously for the two unknowns. To do this, we must divide the pipe into a number of incremental lengths, assign initial values to all the nodes, and prescribe boundary conditions at each end of the pipe.

A major advantage of such a computerized solution method is that it can incorporate complex boundary conditions. We can specify these as variations in V or p, or relations between them, or connection to another pipeline or lines, or to a pump.

The details of these techniques are rather extensive, and can be found in more advanced texts.[6]

Protection from Water Hammer

A pipeline can be protected from damage by water hammer by connecting to it any of a variety of devices that keep the transient pressures within desired limits.

[6] See footnote 5.

Slow-closing valves can be used to control the flow, and by-pass valves can divert sudden changes in flow. Air valves can be connected to the line to admit air if the pressure falls too low. Similarly, automatic relief valves may be fitted to allow water to escape when pressure exceeds a certain value, but these are usually initiated by a pressure drop, and they only work if the reflection time T_r is long enough. Air chambers include so-called bladder accumulators and hydropneumatic surge arrestors. They contain compressed air or gas, which acts as a cushion to absorb pressure changes. Pump flywheels have been used to absorb energy by increasing the inertia of the rotating element, and they thus increase the time required for incremental changes in the flow rate.

Surge tanks are standpipes connected to the pipeline that store liquid when pressures are high and return it to the line when pressures fall. The simple surge tank, which allows diversion of the flow when the downstream valve is closed, is explained next in Sec. 12.7. Other surge tanks admit the liquid through an orifice, which in addition helps to dissipate energy. Yet others, known as differential surge tanks, have multiple chambers with interconnections; the different water levels that occur in the various chambers help to dampen out the oscillations.

EXERCISES

12.6.1 Given that the pipe in Fig. 12.5a is 1500 ft long, that water is initially flowing through it at 2 fps, and that the pressure waves travel along it at 3750 fps, find the maximum water hammer pressure at the valve, and at points 500 ft and 1000 ft from the valve, (*a*) if the valve is completely closed instantaneously, and (*b*) if the valve is partially closed instantaneously, reducing the flow velocity to 0.6 fps.

12.6.2 Given that the pipe in Fig. 12.5*a* is 300 m long, that water is initially flowing through it at 0.8 m/s, and that pressure waves travel along it at 1000 m/s, find the maximum water hammer pressure at the valve, and at points 100 m and 200 m from the valve, (*a*) if the valve is completely closed instantaneously, and (*b*) if the valve is partially closed instantaneously, reducing the flow velocity to 0.2 m/s.

12.6.3 Repeat Exer. 12.6.1 for valve closure times of 0.36 sec.

12.6.4 Repeat Exer. 12.6.2 for valve closure times of 0.33 s.

12.6.5 Repeat Exer. 12.6.1 for valve closure times of 1.00 sec.

12.6.6 Repeat Exer. 12.6.2 for valve closure times of 0.84 s.

12.7 SURGE TANKS

In a hydroelectric plant the flow of water to a turbine must be decreased very rapidly whenever there is a sudden drop in load. This rapid decrease in flow will result in high water-hammer pressures and may result in the need for a very strong and hence expensive supply pipe called a ***penstock*** (Sec. 16.2). There are

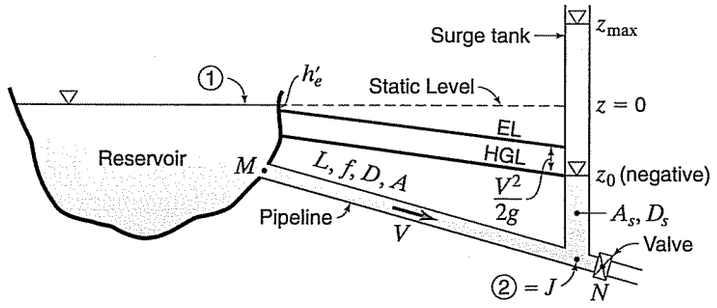


Figure 12.10
Definition sketch for surge tank analysis.

several ways to handle a situation of this sort; one is by use of a ***surge tank,*** or ***surge chamber.*** A simple surge tank is a vertical standpipe connected to the pipeline as shown in Fig. 12.10. With steady flow V_0 in the pipe, the initial water level z_0 in the surge tank is below the static (no flow) level ($z = 0$), as shown in the figure. When the valve is suddenly closed, water rises in the surge tank. The water surface in the tank will then fluctuate up and down until damped out by fluid friction. The section of pipe upstream of the surge tank is in effect afforded protection from the high water-hammer pressures that would exist on valve closure if there were no tank.

An approximate analysis for this simple surge tank may be performed as follows. Initially, for steady conditions, indicated by the subscript 0, the amount by which the water level in the surge tank is below the static level is the sum of the head loss due to friction, the velocity head, and any minor losses in the line such as the entrance loss h'_e shown (see Sec. 8.21). Therefore, as z_0 is negative, we have

$$-z_0 = h_f + \frac{V_0^2}{2g} + \sum k \frac{V_0^2}{2g}$$

or

$$-z_0 = K\frac{V_0^2}{2g} \qquad \text{where} \qquad K = \frac{fL}{D} + 1 + \sum k \tag{12.17}$$

where $\sum k$ is the sum of the coefficients for all minor losses in the line. For the condition soon after the valve is instantaneously closed, when the water is at some level z as it rises up the surge tank, we can write the energy equation (12.6) for unsteady flow between the surface of the reservoir (point 1 on Fig. 12.10) and the end of the pipeline at the surge tank (point 2 = J, on the centerlines). Neglecting fluid friction in the surge tank and inertial effects in the surge tank, and letting h be the vertical distance from point 2 to elevation $z = 0$, at point 2 we have $p_2/\gamma = h + z$ and $z_2 = -h$, so that $(p/\gamma + z)_2 = h + z - h = z$. Substituting

this and conditions at point 1 into Eq. (12.6), we therefore obtain

$$0 - \left(z + \frac{V^2}{2g}\right) = \left(\sum k + \frac{fL}{D}\right)\frac{V^2}{2g} + \frac{L}{g}\frac{dV}{dt}$$

In this equation z represents the level of the water surface in the surge tank measured positively upward from the static water level where $z = 0$ and $(L/g)(dV/dt)$ represents the accelerative head h_a in the pipe between the reservoir and the surge tank (Sec. 12.3).

Using our definition of K from Eq. (12.17), this equation can be written as

$$-z = K\frac{V^2}{2g} + \frac{L}{g}\frac{dV}{dz}\frac{dz}{dt} \tag{12.18}$$

With the valve completely closed, since the flow velocity up the surge tank is $V_s = dz/dt$, the continuity equation is

$$AV = A_s\frac{dz}{dt} \tag{12.19}$$

where A and A_s are the cross-sectional areas of the pipe and surge tank, respectively. Combining Eqs. (12.18) and (12.19) to eliminate dt, and then assuming f and K are constant while integrating and solving for V, yields

$$V^2 = \frac{2g}{K}\left(\frac{LA}{KA_s} - z\right) - C\exp\left(-\frac{KA_s}{LA}z\right) \tag{12.20}$$

This is the general equation relating the velocity V in the pipe to the water-surface level z in the surge tank over the time interval from valve closure to the top of the first surge. After that, the water changes its flow direction and so this equation is no longer valid. However, we know that subsequent oscillations will be smaller due to damping by friction losses.

Before using this equation to find the maximum surge height z_{max}, we must first take care of the unknown constant of integration C. We can do this as follows. The initial conditions for the time interval considered are $V = V_0$ and $z = z_0$ (Eq. 12.17). Substituting these into Eq. (12.20) gives

$$V_0^2 = \frac{2g}{K}\left(\frac{LA}{KA_s} + K\frac{V_0^2}{2g}\right) - C\exp\left[\left(\frac{KA_s}{LA}\right)K\frac{V_0^2}{2g}\right]$$

which simplifies to

$$\frac{2gLA}{K^2A_s} = C\exp\left[\left(\frac{KA_s}{LA}\right)K\frac{V_0^2}{2g}\right] \tag{12.21}$$

The conditions at the end of the time interval, at the top of the first surge, are $V = 0$ and $z = z_{max}$. Substituting these into Eq. (12.20) and rearranging gives

$$\frac{2g}{K}\left(\frac{LA}{KA_s} - z_{max}\right) = C\exp\left(-\frac{KA_s}{LA}z_{max}\right) \tag{12.22}$$

Finally, dividing Eq. (12.22) by Eq. (12.21) to eliminate C results in

$$1 - \frac{KA_s}{LA} z_{\max} = \exp\left[-\frac{KA_s}{LA}\left(K\frac{V_0^2}{2g} + z_{\max}\right)\right] \tag{12.23}$$

Reviewing our result, we see that this equation is explicit for V_0, but is implicit for $z_{\max}$, A_s, A, L, and K, and so therefore f (Appendix B). Also, we note that the right-hand side of this equation must always be positive because it is an exponential, so it follows that the left-hand side must always be positive also, leading to

$$\frac{KA_s}{LA} z_{\max} < 1 \tag{12.24}$$

Note that in all cases we can conveniently replace A_s/A by $(D_s/D)^2$, where D_s is the diameter of the surge tank.

If we wish to find V_0, we can rearrange Eq. (12.23) into

$$V_0^2 = -\frac{2g}{K}\left[z_{\max} + \frac{LA}{KA_s}\ln\left(1 - \frac{KA_s}{LA} z_{\max}\right)\right] \tag{12.25}$$

where the right side will be positive because the ln term will be negative. However, if we wish to solve Eq. (12.23) for $z_{\max}$, A_s, or K (in order to find f), we must either use an equation solver (Appendixes C.1–C.3 and D), or manually use trial and error. For the manual method, because the right side of Eq. (12.23) is usually quite small, it turns out that the limiting value given by Eq. (12.24) is a good indicator of the true answer. For example, when solving for $z_{\max}$, Eq. (12.24) gives $z_{\max} < LA/KA_s$, which serves as a good first estimate for trial and error.

Since this derivation neglected fluid friction and inertial effects in the surge tank, minor losses at the surge tank junction, and assumed constant f and instantaneous valve closure, the value of $z_{\max}$ as computed by Eq. (12.23) will be larger than the true value, and so the results provide a conservative estimate for preliminary design of simple surge tanks. For long pipelines, minor losses and the velocity head in the pipeline will be small compared with the friction head loss. If these are also neglected, according to Eq. (12.17) K must be replaced by fL/D. The results of doing this will provide a slightly more conservative estimate.

More accurate analysis of surge tank phenomena commonly uses computer programs. These use numerical methods to solve the differential and algebraic equations that represent the unsteady liquid motion, taking into account all the factors just mentioned, and including the actual rates of valve closure.

Surge tanks are usually open at the top and of sufficient height that they will not overflow. In some instances they are permitted to overflow if no damage will result. There are many types of surge tanks; other types were discussed at the end of Sec. 12.6.

The surge tank, in addition to providing protection against water-hammer pressures, fulfills another desirable function. That is, in the event of a sudden demand for increased flow, it can quickly provide some excess water, while the entire mass of water in a long pipeline is accelerating. The acceleration of masses of liquids in pipelines was discussed in Sec. 12.4.

SAMPLE PROBLEM 12.6 You are asked to design a surge tank for a 1.5-m-diameter steel pipeline ($f = 0.018$) 800 m long that supplies water to a small power plant. The surge tank will be connected to the pipeline at a point where the centerline is 35 m below the water surface in the reservoir. For a discharge of 12 m^3/s, use only a basic scientific calculator to find the smallest surge tank diameter that will prevent surges from exceeding 18.5 m above the reservoir surface. Neglect inertial effects and fluid friction in the surge tank only. The pipeline inlet is square-edged.

Solution

Sec. 8.21 for square-edged entrance: $k_e = 0.5$

Eq. (12.17): $$K = \frac{0.018(800)}{1.5} + 1 + 0.5 = 11.10$$

$$V_0 = Q_0/A = 12/\pi(1.5/2)^2 = 6.79 \text{ m/s}$$

Eq. (12.17): $z_0 = -11.1(6.79)^2/2(9.81) = -26.09$ m, possible cf 35 m.

From Eq. (12.24): $$A_s < \frac{LA}{Kz_{\max}} = \frac{800\pi(1.5/2)^2}{11.10(18.5)} = 6.88 \text{ m}^2$$

Solving Eq. (12.23) by trial and error (per Sample Prob. 3.5):

First trial: Try $A_s = 6.88$

Last trial: $A_s = 6.06 \text{ m}^2 = (\pi/4)D_s^2$; $D_s = 2.78$ m ***ANS***

Note: We can solve Eq. (12.23) without manual trial and error by using a programmable scientific calculator with an equation (root) solving capability (Appendixes C.1 and D.1), or by using equation solving software (Appendixes C.2–C.3 and D.2–D.3).

EXERCISES

12.7.1 A 48-in-diameter steel pipe ($f = 0.20$, flush inlet) 4000 ft long delivers 275 cfs of water to a power plant. It is protected from water hammer by a 7-ft-diameter surge tank located at the downstream end just before the control valve (Fig. X12.7.1). Without using trial and error, obtain a safe, preliminary estimate of the largest height (above the reservoir surface elevation) to which a surge would rise. Neglect all velocity heads and minor losses; in the surge tank (only) neglect fluid friction and inertial effects.

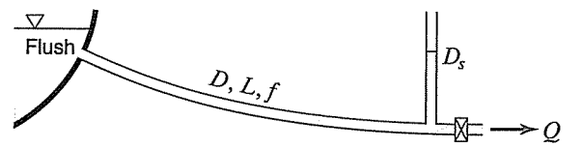


Figure X12.7.1

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

12.7.2 A 900-mm-diameter steel pipe ($f = 0.018$, flush inlet) 850 m long delivers 4 m^3/s of water to a power plant. It is protected from water hammer by a surge tank located at the downstream end just before the control valve (Fig. X12.7.1). Without using trial and error, obtain a safe, preliminary estimate of the smallest surge tank diameter that would be required to prevent surges from exceeding 12 m above the reservoir water-surface elevation. Neglect all velocity heads and minor losses; in the surge tank (only) neglect fluid friction and inertial effects.

PROBLEMS

12.1 (*a*) Outflow from a reservoir with vertical sides is over a spillway ($Q = C_W L H^{3/2}$, Eq. 11.26). Initially there is a steady inflow so that the height of the water surface above the spillway level is z_1. If the inflow is suddenly cut off, prove that the time required for the water level to fall from z_1 to z_2 is $t = (2A_s/C_W L)(z_2^{-0.5} - z_1^{-0.5})$. (*Note:* $z = H$.) (*b*) How long will it take theoretically for the outflow to cease entirely? What factors make this theoretical answer unrealistic? (*c*) Given the crest of the overflow spillway is 100 ft long, the value of C_W in Eq. (11.26) is 3.45, and the constant water-surface area is 700,000 ft^2. With no inflow, find the time required for the water surface to fall from 3 ft to 1 ft above the spillway level.

12.2 Repeat parts (*a*) and (*b*) of Prob. 12.1. (*c*) Given the crest of the overflow spillway is 30 m long, the value of C_W in Eq. (11.26) is 1.91, and the constant water-surface area is 65 000 m^2. With no inflow find the time required for the water surface to fall from 900 mm to 300 mm above the spillway level.

12.3 Figure P12.3 shows a tank with vertical sides containing liquid with a surface area of A_s. The liquid discharges through an orifice under a head z, which varies from the initial height h to zero as the tank empties down to the orifice level. (*a*) Neglecting friction losses, what is the cumulative kinetic energy of the jet during the time required for the liquid surface to drop from h to zero? (*b*) How does this kinetic-energy summation compare with the total energy of the mass of fluid initially in the tank above the orifice level?

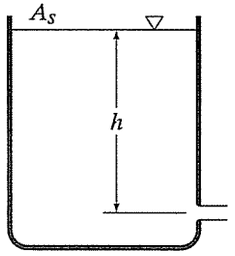


Figure P12.3

12.4 How long will it take for the tank in Prob. 12.3 to empty down to the orifice level if A_s is 8 ft^2, h is 16 ft, and the jet diameter is 5 in? Plot a graph of h versus time, using increments of 4 ft.

12.5 How long will it take for the tank in Prob. 12.3 to empty down to the orifice level if A_s is 0.72 m^2, h is 5 m, and the jet diameter is 125 mm? Plot a graph of h versus time, using increments of 1 m.

12.6 Water enters a reservoir at such a rate that the height z of water above the level of the spillway crest is 3 ft. The spillway ($Q = C_W L H^{3/2}$, $H = z$) is 100 ft long, and the value of C_W is 3.50. The area of the water surface for various water levels is as follows:

z, ft	A_s, ft^2
3.00	850,000
2.50	800,000
2.00	700,000
1.50	550,000
1.25	510,000
1.00	470,000

If the inflow is suddenly reduced to 150 cfs, what will be the water level for equilibrium? How long will it take, theoretically, for equilibrium to be attained? How long will it take for the level to drop from 3 to 1 ft above the spillway level?

12.7 Work Prob. 12.6 using the same numbers but changing ft to m, ft^2 to m^2, and cfs to m^3/s.

12.8 A reservoir has an overflow spillway with a 40-ft-long crest, and a value of $C_W = 3.50$ (Eq. 11.26). For the range of water levels considered here, the area of the water surface is essentially constant at 600,000 ft^2. Initially, the water surface is 3 ft below the level of the spillway crest. If a flow of 500 cfs is suddenly discharged into this reservoir, what will be the height z of water in the reservoir (above spillway crest) for equilibrium? How much time will be required for this equilibrium height to be reached? How much time will be required for the water surface to reach 2 ft above the spillway level? (*Note:* This last part can be solved by integration after substituting x^3 for $z^{3/2}$ and consulting integral tables. However, it will be easier to solve it graphically, either by plotting and actually measuring the area under the curve, or by computing the latter by some method such as Simpson's rule.)

12.9 Attached to the tank in Fig. P12.9 is a flexible 1.5-in-diameter hose ($f = 0.020$) 250 ft long. The tank is hoisted in such a manner that $h = 25 + 2t$, where h is the head in feet and t is the time in seconds. (*a*) Find as accurately as you can the flow rate at $t = 10$ sec. (*b*) Suppose h were decreasing at the same rate. What would be the flow rate when $h = 45$ ft?

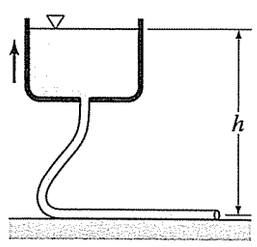


Figure P12.9

12.10 Attached to the tank in Fig. P12.9 is a flexible 40-mm-diameter hose ($f = 0.024$) 65 m long. The tank is hoisted in such a manner that $h = 10 + 0.5t$, where h is the head in meters and t is the time in seconds. (*a*) Find as accurately as you can the flow rate at $t = 10$ s. (*b*) Suppose h were decreasing at the same rate. What would be the flow rate when $h = 15$ m?

12.11 Repeat Prob. 12.9 for the case of a 6-in-diameter hose with all other data remaining the same.

12.12 Repeat Prob. 12.10 for the case of a 150-mm-diameter hose with all other data remaining the same.

12.13 Work Sample Prob. 12.2 for the case where the pipe lengths are 400 and 1500 ft rather than 200 and 750 ft. All other data are the same.

12.14 A 1-in-diameter smooth brass pipe 1000 ft long drains an open 4-ft-diameter cylindrical tank that contains oil having $\rho = 1.8$ slugs/ft^3 and $\mu = 0.0006$ lb·s/ft^2 (Fig. P12.14). The pipe discharges at elevation 100 ft. Find the time required for the oil level to drop from elevation 120 to elevation 108 ft.

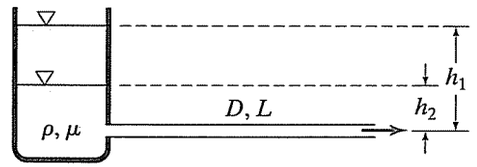


Figure P12.14

12.15 A 25-mm-diameter smooth brass pipe 300 m long drains an open 1.2-m-diameter cylindrical tank which contains oil having $\rho = 950$ kg/m^3 and $\mu = 0.03$ N·s/m^2 (Fig. P12.14). The pipe discharges at elevation 30 m. Find the time required for the oil level to drop from elevation 36 to elevation 32.5 m.

12.16 Figure P12.16 depicts a large reservoir being drained with a pipe system ($f = 0.025$, $L = 2000$ ft, $D = 9$ in, $z_0 = 45$ ft). The flow rate is 3.5 cfs when the pump is initially rotating at 180 rpm. If the pump speed is increased instantaneously to 220 rpm, determine the flow rate as a function of time. Assume that the head h_p developed by the pump is proportional to the square of the rotative speed; that is $h_p \propto n^2$.

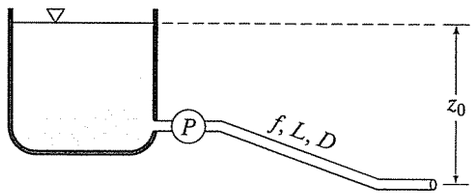


Figure P12.16

12.17 Figure P12.16 depicts a large reservoir being drained with a pipe system ($f = 0.022$, $L = 825$ m, $D = 250$ mm, $z_0 = 14$ m). The flow rate is 120 L/s when the pump is initially rotating at 200 rpm. If the pump speed is increased instantaneously to 250 rpm, determine the flow rate as a function of time. Assume that the head h_p developed by the pump is proportional to the square of the rotative speed; that is $h_p \propto n^2$.

12.18 (*a*) Repeat Prob. 12.16 with all the data the same except use an 18-in-diameter pipe rather than a 9-in pipe. (*b*) Repeat also for the case of an 8-in-diameter pipe.

12.19 (*a*) Repeat Prob. 12.17 with all the data the same except use a 500-mm-diameter pipe rather than a 250-mm pipe. (*b*) Repeat also for the case of a 200-mm-diameter pipe.

12.20 A reservoir is to be drained by a 6-in-diameter pipe (flush entrance, $f = 0.020$) 3500 ft long. The pipe outlet is at an elevation 110 ft below the reservoir water surface. Initially there is no flow since there is a plug at the pipe outlet. Plot Q versus t for time after the plug is removed.

12.21 Repeat Prob. 12.20 for the case where the pipe length is 400 ft rather than 3500 ft.

12.22 A reservoir is to be drained by a 200-mm-diameter pipe ($f = 0.028$) 700 m long. The elevation of the pipe outlet is 50 m below the reservoir water surface. Initially there is no flow because a valve at the pipe outlet is closed. Plot Q (L/s) versus t for time after the valve is suddenly opened.

12.23 A reservoir with its water surface at elevation 850 ft may be drained with a long pipe that discharges at elevation 50 ft. The pipe consists of 4000 ft of 4-in-diameter pipe ($f = 0.020$) followed by 500 ft of 2-in-diameter pipe ($f = 0.020$). Initially a valve at the discharge end is closed and the water in the pipe is at rest. The valve is then suddenly opened and as time ensues the flow rate gradually increases. Determine the time rate of change of the flow rate (cfs/sec) at the instant when the flow rate is 0.40 cfs.

12.24 The pipeline for draining a large reservoir consists of 200 ft of 6-in-diameter pipe ($f = 0.030$) followed by 500 ft of 10-in-diameter pipe ($f = 0.020$). The elevation of the outlet is 100 ft below the reservoir water surface (Fig. P12.24). A valve at the outlet, initially closed, is quickly opened. Derive an equation similar to Eq. (12.8) that is applicable to this situation, and plot flow rate versus time. Neglect minor losses.

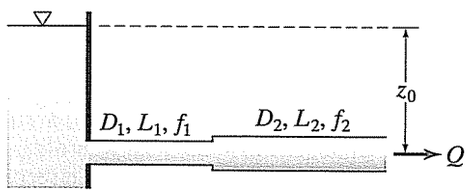


Figure P12.24

12.25 The pipeline for draining a large reservoir consists of 60 m of 150-mm-diameter pipe ($f = 0.030$) followed by 150 m of 250-mm-diameter pipe ($f = 0.020$). The elevation of the outlet is 30 m below the reservoir water surface (Fig. P12.24). A valve at the outlet, initially closed, is quickly opened. Derive an equation similar to Eq. (12.8) that is applicable to this situation, and plot flow rate (L/s) versus time. Neglect minor losses.

12.26 Water in a large reservoir may be released through a 12-in-diameter pipe ($f = 0.020$) 350 ft long. Losses at the pipe entrance are negligible; a nozzle ($C_v = 0.95$) at the outlet end produces a 6-in-diameter jet. The jet discharges at an elevation 50 ft below the reservoir water surface. Initially, there is a tight-fitting plug in the nozzle, which is then removed. For this situation derive an equation similar to Eq. (12.8) and plot flow rate versus time.

12.27 A large open tank containing oil ($s = 0.85$, $\mu = 0.0005$ lb·s/ft^2) is connected to a 3-in-diameter smooth pipe 4000 ft long. The elevation of the discharge point is 10 ft below the liquid surface in the tank (Fig. P12.27). A closed valve at the discharge end of the pipe is opened suddenly. Plot the ensuing flow rate versus time.

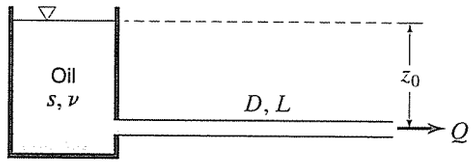


Figure P12.27

12.28 A large open tank containing oil ($s = 0.82$, $\nu = 6.5 \times 10^{-5}$ m^2/s) is connected to a 150-mm-diameter pipe 450 m long. The elevation of the discharge point is 2.2 m below the liquid surface in the tank (Fig. P12.27). A closed valve at the discharge end of the pipe is opened suddenly. Plot the ensuing flow rate (L/s) as a function of time.

12.29 A vertical pipe full of oil is allowed to drain by removing a plug from its lower end. (*a*) Obtain an equation for the varying flow velocity as a function of time t, the pipe diameter D, and the specific weight γ and viscosity μ of the oil. Assume that the head loss is given by the equation of established laminar flow and that surface-tension effects are negligible. (*b*) Obtain an equation for the distance x of the oil surface from the top of the pipe, in terms of the same variables. (*c*) Given that the pipe is 10 ft long, its diameter is 1 in, and the oil has specific gravity 0.88 and viscosity 0.004 lb·s/ft^2, find the time required to drain the pipe after the plug is removed. Check whether the laminar flow assumption is reasonable.

12.30 Repeat parts (*a*) and (*b*) of Prob. 12.29. (*c*) Given that the pipe is 2.8 m long, its diameter is 20 mm, and the oil has specific gravity of 0.88 and viscosity 0.25 N·s/m^2, find the time required to drain the pipe after the plug is removed. Check whether the laminar flow assumption is reasonable.

12.31 (*a*) What is the celerity of a pressure wave in water in a 6-ft-diameter pipe with 0.65-in steel walls? (*b*) If the pipe is 4200 ft long, what is the time required for a pressure wave to make the round trip from the valve? (*c*) If the initial water velocity is 7 fps, what will be the rise in pressure at the valve if the time of closure is less than the time of a round trip? (*d*) If the valve is closed at such a rate that the velocity in the pipe decreases uniformly with respect to time and closure is completed in a time $t_c = 5L/c_j$, approximately what will be the increase in pressure head at the valve when the first pressure unloading wave reaches the valve?

12.32 (*a*) What is the celerity of a pressure wave in water in a 1.8-m-diameter pipe with 125-mm concrete walls? (*b*) If the pipe is 1250 m long, what is the time required for a pressure wave to make the round trip from the valve? (*c*) If the initial water velocity is 2.75 m/s, what will be the rise in pressure at the valve if the time of closure is less than the time of a round trip? (*d*) If the valve is closed at such a rate that the velocity in the pipe decreases uniformly with respect to time and closure is completed in a time $t_c = 5L/c_j$, approximately what will be the increase in pressure head at the valve when the first pressure unloading wave reaches the valve?

12.33 Using Eqs. (12.12) and (12.16) and the data for Figs. 12.8 and 12.9 as given in footnote 3, compute the water-hammer pressure for each case and compare the answers with the actual measurements. Also, for the given data, compute f.

12.34 Water at 15°C is flowing through a 300-mm-diameter welded-steel pipe 2400 m long that drains a reservoir under a head of 50 m (Fig. P12.34). The pipe wall has a thickness of 8.5 mm. (*a*) If a valve at the end of the pipe is completely closed in 10 s, approximately what water-hammer pressure will be developed? (*b*) If the steady-state flow is instantaneously reduced to one-half its original value, what water-hammer pressure would you expect?

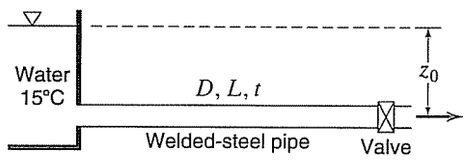


Figure P12.34

12.35 For the situation described in Sample Prob. 12.5, find the water-hammer pressure at the valve if a flow of 90 cfs is reduced to 30 cfs in 3.2 sec. Under these conditions, what would be the maximum water-hammer pressures at points 500 and 1500 ft from the reservoir?

12.36 In Fig. P12.36 the total length of pipe is 10,000 ft, its diameter is 42 in, and its wall thickness is 0.89 in. Assume E = 32,000,000 psi and E_v = 300,000 psi. If the initial velocity for steady flow is 8 fps and the valve at N is partially closed so as to reduce the flow to half of the initial velocity in 3.6 sec, find (*a*) the maximum pressure rise due to the water hammer; (*b*) the location of the point of maximum *total* pressure.

12.37 Refer to Fig. P12.36, but take all the dimensions given in feet to be in meters instead. This 10-km-long pipe has a diameter of 1.35 m and a wall thickness of 21 mm. Assume E = 205 GPa and E_v = 2 GPa. The initial steady flow velocity is 5 m/s. The valve at N is then partially closed so as to reduce the velocity to 1 m/s in 14 s. Find (*a*) the maximum pressure rise due to water hammer and (*b*) the location of the point of maximum *total* pressure.

12.38 Derive Eq. (12.20).

12.39 A 1.5-m-diameter steel pipeline 1000 m long (flush inlet, f = 0.020) carries water at Q_0 (m³/s) from a reservoir to a power plant (Fig. P12.39). When the valve at the outlet end is closed instantaneously, water rises in the 2.5-m-diameter simple surge tank immediately adjacent to the valve. Determine the maximum allowable initial discharge Q_0 so that the resulting surge will not rise more than 12 m above the reservoir surface. In the surge tank only, neglect the velocity head, minor losses, fluid friction, and inertial effects.

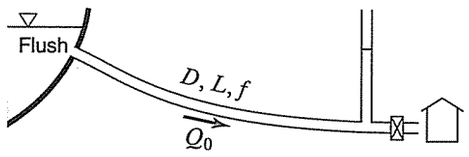


Figure P12.39

12.40 Repeat Prob. 12.39 while also neglecting the velocity head and minor losses in the pipeline.

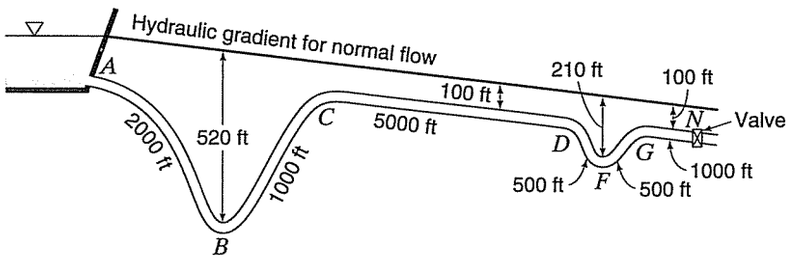


Figure P12.36

12.41 Refer to Fig. 12.10. A 42-in-diameter steel pipe MN 3600 ft long (flush inlet, f = 0.017) supplies water to a small power plant. The discharge is 200 cfs, JN = 100 ft, and the elevations of J and the valve N are respectively 130 ft and 145 ft below the reservoir water surface. To protect against instantaneous closure of the valve, what height would be required for the simple 6.5-ft-diameter surge tank if it is not to overflow? In the surge tank only, neglect the velocity head, minor losses, fluid friction, and inertial effects.

12.42 Repeat Prob. 12.41 while also neglecting the velocity head and minor losses in the pipeline.

12.43 Repeat Prob. 12.41 for a surge tank diameter of 10 ft.

12.44 Using the data of Prob. 12.41, find the diameter of surge tank that will produce a surge requiring a tank height of 165 ft.

12.45 Using the data of Prob. 12.42, find the diameter of surge tank that will produce a surge requiring a tank height of 165 ft.

12.46 Refer to Fig. 12.10. A 1-m-diameter steel pipe MN 1070 m long (flush inlet, f = 0.016) supplies water to a small power plant. The discharge is 2.45 m³/s, JN = 20 m, and the elevations of J and the valve N are respectively 25 m and 30 m below the reservoir water surface. To protect against instantaneous closure of the valve, what height would be required for the simple 3.5-m-diameter surge tank if it is not to overflow? In the surge tank only, neglect the velocity head, minor losses, fluid friction, and inertial effects.

12.47 Repeat Prob. 12.46 while also neglecting the velocity head and minor losses in the pipeline.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

CHAPTER 13
Steady Flow of Compressible Fluids

We call fluids that show appreciable variation in density ***compressible fluids.*** The variation in density is mainly caused by variations in pressure and temperature. We sometimes call the study of such fluids in motion ***gas dynamics.***

When dealing with a flowing compressible fluid, if the density change is gradual and not more than a few percent, we can treat the flow as incompressible by using an average density for best results. However, if $\Delta\rho/\rho > 0.05$, the effects of compressibility must be considered. The purpose of this chapter is to investigate compressible-fluid problems that require such considerations. The discussion will be limited to steady one-dimensional flow of compressible fluids. Before proceeding further, the reader is advised to review Secs. 2.7, 2.8, and possibly 2.9.

13.1 THERMODYNAMIC CONSIDERATIONS

As a first step in understanding the flow of compressible fluids, it is advantageous in our discussion to review briefly some thermodynamic principles. The thermodynamic properties of a gas (Appendix A, Table A.5) include the gas constant R, specific heat c_p at constant pressure, specific heat c_v at constant volume, and the specific heat ratio $k = c_p/c_v$. The density ρ (or specific volume v) of a gas depends on the absolute pressure p and absolute temperature T of the gas; for real gases at low pressures and moderate or high temperatures the relationship between these properties is closely defined by the perfect (ideal) gas law

$$\frac{p}{\rho} = pv = RT \tag{2.4}$$

where $v = 1/\rho$ is the specific volume (Sec. 2.3). Equation (2.4) is an equation of state (or property relation) that we discussed in Sec. 2.7. In this book we shall assume that all gases are perfect (also called "ideal," not to be confused with ideal fluids, discussed in Sec. 2.10). Another fundamental equation introduced in Sec. 2.7 is pv^n = constant, which describes the changes of perfect gas properties from one state to another for a particular process.

We discussed the first law of thermodynamics in Sec. 5.5. The second law of thermodynamics, discussed in Sec. 5.6, deals with reversibility of processes. In general, a ***reversible process*** is one in which both the system *and its surroundings* can be returned precisely to their initial states. Processes involving friction, heat transfer, and mixing of gases are not reversible. Clearly, reversible processes imply ideal fluids (Sec. 2.10). All real processes are irreversible, but we can approximate some well by reversible processes when the irreversibility is minor. For example, we can approximate the flow through a converging nozzle where there is little friction and little or no heat transfer as a reversible process. But flow in a pipeline is an irreversible process because of the pipe friction.

Related to irreversibility is ***entropy*** s, a property that measures the disorder, or the amount of energy unavailable for useful work, during a natural flow process. In real processes it always increases so that available energy decreases. Processes with constant entropy occur only in theory, but may be closely approximated in fact.

Process equations for perfect gases include the following:

For isothermal conditions ($n = 1$): $pv = \text{constant}$

For isentropic conditions ($n = k$): $pv^k = \text{constant}$

An ***isothermal*** process is one in which there is no change of temperature, while an ***adiabatic*** process is one in which no heat is added or removed. An adiabatic process that is reversible (entropy is constant) and for which k is a constant is known as an ***isentropic*** process. For adiabatic processes with friction, $n < k$ for expansion and $n > k$ for compression. We solve compressible-flow problems in a manner similar to that used for incompressible flows, except that the equation describing the compressible process must be included.

The ***enthalpy*** h per unit mass of a gas is defined by

$$h = i + \frac{p}{\rho} \tag{5.32}$$

and for a perfect gas, $h = i + RT$, where i, the internal energy per unit mass due to the kinetic energy of molecular motion and the forces between molecules, is a function of temperature. Hence enthalpy is a composite energy property of a gas, and is a function of only temperature for a perfect gas.

The ***specific heat at constant pressure,*** c_p, is defined as the increase in enthalpy per unit of mass when the temperature of a gas is increased one degree with its pressure held constant. Thus,

$$c_p = \left(\frac{\partial h}{\partial T}\right)_{p=\text{constant}} \tag{13.1}$$

where h is the enthalpy per unit of mass.

The ***specific heat at constant volume,*** c_v, is defined as the increase in internal energy per unit of mass when the temperature is increased one degree with

its volume held constant. Thus,

$$c_v = \left(\frac{\partial i}{\partial T}\right)_{v=\text{constant}} \tag{13.2}$$

where i is the internal energy per unit mass.

For perfect gases these equations can be written as $dh = c_p dT$ and $di = c_v dT$. Now since $h = i + p/\rho = i + RT$, $dh = di + R dT$. Combining these relationships leads to

$$c_p - c_v = R \tag{13.3}$$

Combining the specific heat ratio $k = c_p/c_v$ with Eq. (13.3) gives

$$c_p = \frac{k}{k-1}R, \quad c_v = \frac{R}{k-1} \tag{13.4}$$

SAMPLE PROBLEM 13.1 Compute the change in internal energy and the change in enthalpy of 15 kg of air if its temperature is raised from 20 to 30°C. The initial pressure is 95 kPa abs.

Solution

Properties of gases are given in Appendix A, Table A.5.

$$\Delta i = c_v(T_2 - T_1) = 716\frac{\text{N·m}}{\text{kg·K}}(30 - 20)\,\text{K} = 7160\ \text{J/kg (or N·m/kg)}$$

Change in internal energy $= \Delta i \times (15\ \text{kg}) = 107400\ \text{J}$

$$\Delta h = c_p(T_2 - T_1) = 1003(10) = 10030\ \text{J/kg}$$

Change in enthalpy $= \Delta h \times (15\ \text{kg}) = 150000\ \text{J}$ ***ANS***

SAMPLE PROBLEM 13.2 Suppose that 15 kg of air ($T_1 = 20$°C) of Sample Prob. 13.1 were compressed isentropically to 40% of its original volume. Find the final temperature and pressure, the work required, and the changes in internal energy and enthalpy.

Solution

The following relations apply: $pv = RT$ and $pv^k = \text{constant}$, where $k = 1.40$ for air.

$$pv^k = pv\frac{v^k}{v} = \frac{RT}{v}v^k = RTv^{k-1} = \text{constant}$$

Since R = constant, Tv^{k-1} = constant

$$T_2 = T_1\left(\frac{v_1}{v_2}\right)^{k-1} = (273 + 20)\left(\frac{1.0}{0.4}\right)^{0.40} = 423\ \text{K} = 159°\text{C} \quad \textit{ANS}$$

$\frac{pv}{T} = R = \text{constant}$, $p_1 = 95$ kPa abs (given in Sample Prob. 13.1)

$$\frac{p_1 v_1}{T_1} = \frac{p_2 v_2}{T_2} \quad \text{and} \quad v_2 = 0.4v_1$$

$$\frac{95v_1}{293} = \frac{p_2(0.4v_1)}{423} \quad \text{so} \quad p_2 = 343\ \text{kPa abs (or kN/m}^2\text{ abs)} \quad \textit{ANS}$$

Since this is an adiabatic process, the work required is equal to the change in internal energy. This can be confirmed by computing the values of the pressure and corresponding volumes occupied by the gas during the isentropic process, plotting a pressure-vs-volume curve, and finding the area under the curve and thereby determining the work done on the fluid. Thus the work required is

$$\int_{s_1}^{s_2} F\,ds = \int_{s_1}^{s_2} (F/A)A\,ds = \int_{\text{vol 1}}^{\text{vol 2}} p\,d(\text{vol})$$

or

$$\Delta i = c_v(T_2 - T_1) = 716(423 - 293) = 92900\ \text{J/kg}$$

Work required = change in internal energy $= \Delta i \times 15\ \text{kg} = 1393000\ \text{J}$ ***ANS***

$$\Delta h = c_p(T_2 - T_1) = 1003(130) = 130100\ \text{J/kg}$$

Change in enthalpy $= \Delta h \times 15\ \text{kg} = 1952000\ \text{J}$ ***ANS***

EXERCISES

13.1.1 Compute the change in enthalpy of 15 slugs of oxygen if its temperature is increased from 120°F to 155°F.

13.1.2 Compute the change in enthalpy of 250 kg of oxygen if its temperature is increased from 50°C to 70°C.

13.1.3 Suppose 15 slugs of oxygen are compressed isentropically to 80% of their original volume. Find the final temperature and pressure, the work required, and the change in enthalpy. Assume $T_1 = 120$°F and $p_1 = 200$ psia.

13.1.4 Suppose 250 kg of oxygen are compressed isentropically to 80% of its original volume. Find the final temperature and pressure, the work required, and the change in enthalpy. Assume $T_1 = 50$°C and $p_1 = 1400$ kPa abs.

13.1.5 Using the data of Sample Prob. 13.2 compute $\Delta(p/\rho)$ and thus show that $\Delta h = \Delta i + \Delta(p/\rho)$.

13.2 FUNDAMENTAL EQUATIONS APPLICABLE TO THE FLOW OF COMPRESSIBLE FLUIDS

We have already stated the fundamental equations for the flow of compressible fluids in Chaps. 4, 5, 6, and 7. For convenience, we restate them here.

Continuity

The expression for continuity for steady one-dimensional flow of a compressible fluid (from Sec. 4.7) is

$$\dot{m} = \rho AV = \rho Q = \text{constant} \tag{13.5}$$

where $\dot{m}$ is the mass flow rate and $Q = AV$.

Energy Equation

For one-dimensional steady flow of a compressible fluid if there is no work interaction (no machine) between sections 1 and 2 the energy equation[1] is expressible as:

$$h_1 + \frac{V_1^2}{2} + q_H = h_2 + \frac{V_2^2}{2} \tag{13.6}$$

where q_H is the heat interaction per unit mass. It is convenient to combine the static enthalpy per unit mass h and the kinetic energy of the fluid into a single term we call ***stagnation enthalpy*** or ***total enthalpy*** h_O, defined as

$$h_O = h + \frac{V^2}{2} \tag{13.7}$$

Momentum Equation

The momentum equation (Sec. 6.1) for steady one-dimensional flow of a compressible fluid is

$$\sum F = \dot{m}(V_2 - V_1) = \rho_2 Q_2 V_2 - \rho_1 Q_1 V_1 \tag{13.8}$$

Euler Equation

For one-dimensional flow of an ideal fluid in a frictionless constant-area duct the Euler equation (Sec. 5.2) may be expressed as

$$\frac{dp}{\rho} + VdV = 0 \tag{13.9}$$

[1] The reader should review Secs. 5.5 and 5.7 for a discussion of the energy equation as it applies to compressible fluids.

In Eqs. (13.6) and (13.9) the z terms were dropped, for in the flow of compressible fluids the z terms are almost always negligible compared with the other terms in the energy equation.

Mach Number

In Chap. 7 we mentioned an important dimensionless parameter, the ***Mach number*** **M**,

$$\mathbf{M} = \frac{V}{c} \tag{13.10}$$

where V is the velocity of flow and c is the ***speed of sound*** (***sonic*** or ***acoustic velocity***), i.e., the ***celerity*** at which a pressure wave will travel through a compressible fluid. If $\mathbf{M} < 1$, the flow is ***subsonic;*** if $\mathbf{M} = 1$, the flow is ***sonic;*** if $\mathbf{M} > 1$, the flow is ***supersonic.***

13.3 SPEED OF SOUND

Consider an elastic fluid at a pressure p and a density ρ at rest in a rigid pipe of cross-sectional area A as shown in Fig. 13.1*a*. Suppose a piston at one end is suddenly moved to the right with a small velocity dV. This will produce an infinitesimal pressure wave, which will travel through the fluid with a velocity (celerity) c,

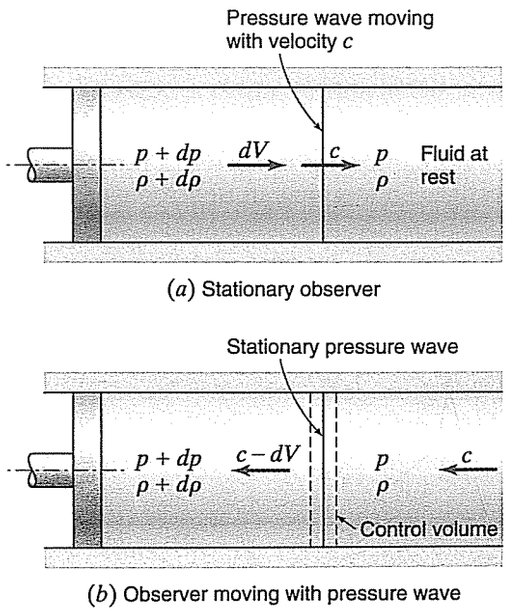


Figure 13.1
Propagation of a weak pressure pulse.

and the pressure and density of the fluid will change to $p + dp$ and $\rho + d\rho$. The fluid behind the wave is set in motion with a velocity dV. In Fig. 13.1*b* this process is shown relative to an observer moving with the wave. The continuity equation, applied to the control volume shown, gives

$$\rho A c = (\rho + d\rho)A(c - dV)$$

Since the amplitude of the compression wave is assumed infinitesimal, the term $d\rho\, dV$ is of second order and can be neglected. The above equation then becomes

$$\rho\, dV = c\, d\rho \tag{13.11}$$

The momentum equation, applied to a control volume, is

$$pA - (p + dp)A = \rho A c[(c - dV) - c]$$

which reduces to

$$dp = c\rho\, dV \tag{13.12}$$

Combining Eqs. (13.11) and (13.12) to eliminate dV gives

$$c^2 = \frac{dp}{d\rho} \tag{13.13}$$

A sonic, or pressure, wave travels through a fluid with such a high velocity that there is no time for any appreciable heat transfer from any heat of compression; moreover, the fluid friction is negligible and thus the process is isentropic. The speed of sound c can thus be written as

$$c = \sqrt{\left(\frac{\partial p}{\partial \rho}\right)_s} \tag{13.14}$$

where the subscript s indicates isentropic (constant entropy) conditions. For a perfect gas undergoing an isentropic process, pressure is related to density by

$$p = C\rho^k$$

Differentiating with respect to ρ gives

$$\left(\frac{\partial p}{\partial \rho}\right)_s = Ck\rho^{k-1} = kC\left(\frac{\rho^k}{\rho}\right) = k\left(\frac{p}{\rho}\right) = kRT$$

Substituting this expression into Eq. (13.14) gives the speed of sound in a perfect gas:

$$c = \sqrt{kRT} = \sqrt{\frac{kp}{\rho}} \tag{13.15}$$

This shows that for perfect gas the sonic speed is a function of its absolute temperature.

The ***isentropic compressibility*** of a fluid (see Sec. 13.1) is defined as

$$K_s \equiv \frac{1}{\rho}\left(\frac{\partial \rho}{\partial p}\right)_s$$

But in liquids and solids, changes in pressure generally produce small changes in temperature, so that $(\partial p/\partial \rho)_s \approx (\partial p/\partial \rho)_T$. Also, the bulk modulus of elasticity (Secs. 2.5, 2.8)

$$E_v = -v\left(\frac{\partial p}{\partial v}\right)_T = \rho\left(\frac{\partial p}{\partial \rho}\right)_T \approx \rho\left(\frac{\partial p}{\partial \rho}\right)_s$$

so that

$$\left(\frac{\partial p}{\partial \rho}\right)_s \approx \frac{E_v}{\rho} \tag{13.16}$$

Combining Eqs. (13.14) and (13.16) gives the speed of sound in terms of the modulus of elasticity as

$$c = \sqrt{\frac{E_v}{\rho}} \tag{13.17}$$

The foregoing analysis has considered the pipe to be rigid. In reality the pipe is elastic, and the stretching of the pipe walls due to the pressure wave makes the modulus of the combination less than that of the fluid alone.

We will represent this new joint (or combined) modulus by E_j, and we shall let $dv = dv' + dv''$, where dv' is due to compression of the fluid and dv'' is due to stretching of the pipe wall. Thus $E_j = -v\, dp/(dv' + dv'')$, from which

$$\frac{1}{E_j} = -\frac{dv'}{v\, dp} - \frac{dv''}{v\, dp} \tag{13.18}$$

The first term on the right side is seen to be $1/E_v$ (Sec. 2.5). From the concept of hoop tension, the increment of stress in the wall of the pipe is $r\, dp/t$, where r is the radius of the pipe and t is its thickness. If the circumference is stretched an amount dl, the unit deformation is $dl/2\pi r$. Since $dl = 2\pi\, dr$, the increment of unit deformation becomes dr/r. From these relations, recalling that the modulus of elasticity E of a solid = (increment of stress) ÷ (increment of unit deformation), we obtain $dp = Et\, dr/r^2$. For unit length of pipe and unit mass of fluid, $v = \pi r^2$, and the increase in volume per unit length of pipe for a unit mass of fluid is equal to the increase in area, so that $dv'' = 2\pi r\, dr$. Substituting these quantities for the three items in the last term of Eq. (13.18) gives $dv''/(v\, dp) = 2r/Et = D/Et$, where D is the pipe diameter. Therefore

$$\frac{1}{E_j} = \frac{1}{E_v} + \frac{D}{Et} \tag{13.19}$$

Solving for E_j gives

$$E_j = \frac{E_v}{1 + (D/t)(E_v/E)} \tag{13.20}$$

The celerity c_j of a pressure wave in an elastic fluid inside an elastic pipe is then

$$c_j = \sqrt{\frac{E_j}{\rho}} = \sqrt{\frac{E_j}{\rho}\frac{E_v}{E_v}} = c\sqrt{\frac{E_j}{E_v}} = c\sqrt{\frac{1}{1 + (D/t)(E_v/E)}} \tag{13.21}$$

From Eqs. (13.20) and (13.21) we see that for a rigid pipe (i.e., $E = \infty$), E_j reduces to E_v and c_j reduces to c.

EXERCISES

13.3.1 Use Sec. 13.3 to calculate the sonic velocity in air at sea level and at elevations 5000, 10,000, 20,000, and 30,000 ft. Assume standard atmosphere (Appendix A, Table A.3).

13.3.2 Use Sec. 13.3 to calculate the sonic velocity in air at sea level and at elevations 2000 and 10000 m, expressing the answers in SI units. Assume standard atmosphere (Appendix A, Table A.3).

13.4 ADIABATIC FLOW (WITH OR WITHOUT FRICTION)

If heat transfer q_H is zero, the flow is ***adiabatic.*** Hence the energy Eq. (13.6) may be written as

$$h_1 + \frac{V_1^2}{2} = h_2 + \frac{V_2^2}{2} \tag{13.22}$$

Since $\Delta h = c_p\,\Delta T$, we get for adiabatic flow

$$V_2^2 - V_1^2 = 2(h_1 - h_2) = 2c_p(T_1 - T_2) \tag{13.23}$$

From Eq. (13.4), $c_p = kR/(k-1)$, and for a perfect gas from Eq. (2.4), $pv = RT$. Substituting these into Eq. (13.23) gives for adiabatic flow

$$V_2^2 - V_1^2 = \frac{2k}{k-1}(p_1v_1 - p_2v_2) = \frac{2k}{k-1}RT_1\left(1 - \frac{T_2}{T_1}\right) \tag{13.24}$$

The preceding equations are valid for flow either with or without friction. Note that in these equations the temperatures and pressures are absolute.

Equation (13.22) can be written as

$$c_pT_1 + \frac{V_1^2}{2} = c_pT_2 + \frac{V_2^2}{2} = c_pT_O \tag{13.25}$$

where T_O is the ***stagnation temperature*** (where V is zero, see Sec. 5.4). Thus, in adiabatic flow, the stagnation temperature is constant along a streamline regardless of whether or not the flow is frictionless.

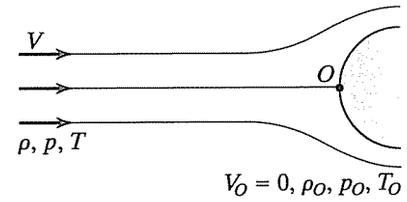


Figure 13.2
Stagnation point.

13.5 STAGNATION PROPERTIES

In this section we develop relations between static and stagnation properties (Sec. 5.4). Figure 13.2 shows a stagnation point in compressible flow. At the stagnation point O the velocity V_O is zero and the increased pressure causes a rise in density and temperature.

Equation (13.25) can be used to define the ***stagnation temperature*** T_O:

$$T_O = T + \frac{V^2}{2c_p} \quad \text{or} \quad \frac{T_O}{T} = 1 + \frac{V^2}{2c_pT}$$

But from Eqs. (13.10) and (13.15), $V^2/(kRT) = \mathbf{M}^2$, and from Eq. (13.4), $c_p = kR/(k-1)$; therefore the ratio of the stagnation temperature to the static temperature becomes

$$\frac{T_O}{T} = 1 + \frac{k-1}{2}\mathbf{M}^2 \tag{13.26}$$

This relationship is valid for adiabatic flow and for isentropic flow. Note also that T_O is the same for all points in the flow, provided that the flow is adiabatic.

When a perfect gas flows isentropically, its pressure and density are related to temperature in the following ways:

$$\frac{p_O}{p} = \left(\frac{T_O}{T}\right)^{k/(k-1)}, \quad \frac{\rho_O}{\rho} = \left(\frac{T_O}{T}\right)^{1/(k-1)}$$

where p_O and ρ_O are the ***stagnation pressure*** and ***stagnation density,*** respectively. By combining these with Eq. (13.26), pressure and density can be expressed in terms of Mach number:

$$\frac{p_O}{p} = \left(1 + \frac{k-1}{2}\mathbf{M}^2\right)^{k/(k-1)} \tag{13.27}$$

and

$$\frac{\rho_O}{\rho} = \left(1 + \frac{k-1}{2}\mathbf{M}^2\right)^{1/(k-1)} \tag{13.28}$$

p_O and ρ_O are the same at all points in the flow, provided that the flow is isentropic.

TABLE 13.1
One-dimensional isentropic flow functions with $k = 1.4$

M	T/T_o	p/p_o	ρ/ρ_o
0	1.00000	1.00000	1.00000
0.10	0.99800	0.99303	0.99502
0.20	0.99206	0.97250	0.98028
0.30	0.98232	0.93947	0.95638
0.40	0.96899	0.89561	0.92427
0.50	0.95238	0.84302	0.88517
0.60	0.93284	0.78400	0.84045
0.70	0.91075	0.72093	0.79158
0.80	0.88653	0.65602	0.73999
0.90	0.86059	0.59126	0.68704
1.00	0.83333	0.52828	0.63394
1.10	0.80515	0.46835	0.58170
1.20	0.77640	0.41238	0.53114
1.30	0.74738	0.36091	0.48290
1.40	0.71839	0.31424	0.43742
1.50	0.68966	0.27240	0.39498
1.60	0.66138	0.23527	0.35573
1.70	0.63371	0.20259	0.31969
1.80	0.60680	0.17404	0.28682
1.90	0.58072	0.14924	0.25699
2.00	0.55556	0.12781	0.23005

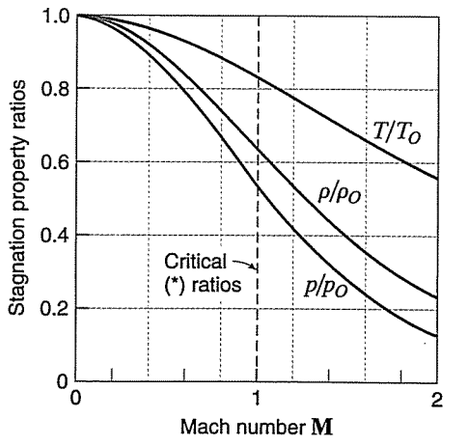


Figure 13.3
Stagnation property ratios for one-dimensional flow of a perfect gas with $k = 1.4$ (see also Table 13.1).

Stagnation property ratios are given numerically in Table 13.1 and presented qualitatively in Fig. 13.3, as functions of **M**.

The speed of sound also is a function of temperature:

$$\frac{c_o}{c} = \sqrt{\frac{T_o}{T}}$$

Therefore sonic speed can be expressed as a function of the Mach number:

$$\frac{c_o}{c} = \left(1 + \frac{k-1}{2}\mathbf{M}^2\right)^{1/2} \tag{13.29}$$

where c is the speed of sound in the gas, while c_o is the speed of sound when the gas is at the stagnation temperature. When a flow is decelerated adiabatically to zero velocity, the temperature of the fluid becomes T_o. However, the fluid will not necessarily exist at the isentropic stagnation state. Unless deceleration is accomplished isentropically, the actual stagnation pressure and stagnation density fall below the isentropic stagnation values.

If we expand Eq. (13.27) by the binomial theorem, we obtain

$$p_o = p\left[1 + \frac{k}{2}\mathbf{M}^2 + \frac{k}{8}\mathbf{M}^4 + \frac{k(2-k)}{48}\mathbf{M}^6 + \cdots\right]$$

But since

$$\frac{1}{2}kp\mathbf{M}^2 = \frac{1}{2}kp\frac{V^2}{kRT} = \frac{1}{2}\rho V^2$$

this expression becomes

$$p_o = p + \frac{1}{2}\rho V^2\left[1 + \frac{\mathbf{M}^2}{4} + \frac{(2-k)\mathbf{M}^4}{24} + \cdots\right] \tag{13.30}$$

or

$$\frac{p_o - p}{\frac{1}{2}\rho V^2} = 1 + \frac{\mathbf{M}^2}{4} + \frac{(2-k)\mathbf{M}^4}{24} + \cdots \tag{13.31}$$

where $\frac{1}{2}\rho V^2$ is known as the ***dynamic pressure.***

Equation (13.30) is identical with Eq. (5.18); it is applicable only if $\frac{1}{2}(k-1)\mathbf{M}^2 < 1$. In determining the stagnation pressure, it indicates that the error caused by neglecting compressibility depends on the Mach number of the flow. At low Mach numbers the error is insignificant, but as **M** increases, the error becomes sizeable. Note also that for incompressible flow ($\mathbf{M} = 0$), $p_o = p + \frac{1}{2}\rho V^2$.

When $\mathbf{M} = 1$, the static-to-stagnation property ratios are called ***critical ratios.*** They are

$$\frac{T^*}{T_o} = \frac{2}{k+1} = 0.833 \quad (\text{for } k = 1.4) \tag{13.32}$$

$$\frac{p^*}{p_o} = \left(\frac{2}{k+1}\right)^{k/(k-1)} = 0.528 \quad (\text{for } k = 1.4) \tag{13.33}$$

$$\frac{\rho^*}{\rho_o} = \left(\frac{2}{k+1}\right)^{1/(k-1)} = 0.634 \quad (\text{for } k = 1.4) \tag{13.34}$$

where the asterisks identify critical properties.

SAMPLE PROBLEM 13.3 Find the Mach number, stagnation pressure, and stagnation temperature of nitrogen flowing at 600 ft/sec if the pressure and temperature in the undisturbed flow field are 100 psia and 200°F, respectively. (See Appendix A, Table A.5, for properties of gases.)

Solution

Eq. (13.15): $$c = \sqrt{kRT} = \sqrt{1.4(1773)660} = 1280 \text{ ft/sec}$$

Eq. (13.10): $$\mathbf{M} = \frac{V}{c} = \frac{600}{1280} = 0.469 \qquad \textit{ANS}$$

From Eq. (13.15): $$\rho = \frac{kp}{c^2} = \frac{1.4(100 \times 144)}{(1280)^2} = 0.0123 \text{ slug/ft}^3$$

From Eq. (13.30):

$$p_O = 100(144) + \tfrac{1}{2}(0.0123)(600)^2\left[1 + \tfrac{1}{4}(0.469)^2 + \frac{(2 - 1.4)}{24}(0.469)^4 + \cdots\right]$$

$$= 14{,}400 + 2214(1 + 0.0550 + 0.00121 + \cdots)$$

$$= 16{,}740 \text{ lb/ft}^2 = 116.2 \text{ psia} \qquad \textit{ANS}$$

Applying Eq. (13.25): $$c_pT_1 + \frac{V_1^2}{2} = c_pT_O$$

$$6210(660) + \frac{(600)^2}{2} = 6210T_O$$

$$T_O = 689°\text{R} = 229°\text{F} \qquad \textit{ANS}$$

Alternatively, from Eq. (13.27),

$$p_O = p\left(1 + \frac{k-1}{2}\mathbf{M}^2\right)^{k/(k-1)} = 100[1 + 0.2(0.469)^2]^{1.4/0.4}$$

$$= 116.3 \text{ psia} \qquad \textit{ANS}$$

and from Eq. (13.26)

$$T_O = T\left(1 + \frac{k-1}{2}\mathbf{M}^2\right) = 660[1 + 0.2(0.469)^2] = 689°\text{R} \qquad \textit{ANS}$$

A second alternative is to use Table 13.1. At $\mathbf{M} = 0.469$, we find by interpolation that $p/p_O = 0.859$ and $T/T_O = 0.958$. Therefore $p_O = 100/0.859 = 116.4$ psia and $T_O = 660/0.958 = 689°\text{R}$.

EXERCISES

13.5.1 Air flows past an object at 600 ft/sec. Determine the stagnation pressures and temperatures in the standard atmosphere at elevations of sea level, 5000 and 30,000 ft.

13.5.2 Air flows past an object at 200 m/s. Determine the stagnation pressures and temperatures in the standard atmosphere at elevations of sea level, 2000 and 10000 m.

13.5.3 Air at 250 psia is moving at 500 ft/sec in a high-pressure wind tunnel at a temperature of 100°F. Find the stagnation pressure and temperature. Note the magnitude of the sonic velocity for the 250-psia 100°F air.

13.5.4 Air at 1750 kPA abs is moving at 150 m/s in a high-pressure wind tunnel at a temperature of 40°C. Find the stagnation pressure and temperature. Note the magnitude of the sonic velocity for the 1750-kPa 40°C air.

13.6 ISENTROPIC FLOW

Adiabatic flow (zero heat transfer) which is *frictionless* (or ideal) is referred to as ***isentropic*** (constant entropy) flow. Such flow does not occur in nature. However, we can consider flow through a nozzle or flow in a free stream of fluid over a reasonably short distance to be isentropic because there is very little heat transfer and fluid-friction effects are small. We can derive equations for isentropic flow by substituting $pv^k = \text{constant}$ into Eq. (13.24). The resulting equations for isentropic flow are

$$\frac{V_2^2 - V_1^2}{2} = \frac{p_1}{\rho_1}\frac{k}{k-1}\left[1 - \left(\frac{p_2}{p_1}\right)^{(k-1)/k}\right] = \frac{p_2}{\rho_2}\frac{k}{k-1}\left[\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right] \tag{13.35}$$

Often, using Eq. (2.4), p/ρ is replaced by RT. Equation (13.35) may also be derived by integrating the Euler equation $dp/\rho + VdV = 0$ along a stream tube, noting that $pv^k = \text{constant}$.

The relation between the temperatures at two points along a streamline in isentropic flow can be derived by equating the expressions for $V_2^2 - V_1^2$ from Eqs. (13.24) and (13.35). The result is

$$T_2 = T_1 - \frac{p_1}{R\rho_1}\left[1 - \left(\frac{p_2}{p_1}\right)^{(k-1)/k}\right] = T_1 - T_1\left[1 - \left(\frac{p_2}{p_1}\right)^{(k-1)/k}\right]$$

from which $$\frac{T_2}{T_1} = \left(\frac{p_2}{p_1}\right)^{(k-1)/k} \tag{13.36}$$

This is the isentropic relation between temperature and pressure; once again, temperatures and pressures are absolute. We first met this equation in Sec. 2.7.

EXERCISES

13.6.1 Air at a pressure of 150 psia and a temperature of 100°F expands in a suitable nozzle to 15 psia. (*a*) If the flow is frictionless and adiabatic and the initial velocity is negligible, find the final velocity by Eq. (13.35). (*b*) Find the final temperature at the end of the expansion through use of Eq. (13.23).

13.6.2 Air at a pressure of 1000 kPa abs and a temperature of 40°C expands in a suitable nozzle to 100 kPa abs. (*a*) If the flow is frictionless and adiabatic and the initial velocity is negligible, find the final velocity by Eq. (13.35). (*b*) Find the final temperature at the end of the expansion through use of Eq. (13.23).

13.6.3 Carbon dioxide flows isentropically. At a point in the flow the velocity is 50 ft/sec and the temperature is 125°F. At a second point on the same streamline the temperature is 80°F. What is the velocity at the second point? Check if your answer is valid.

13.6.4 Carbon dioxide flows isentropically. At a point in the flow the velocity is 15 m/s and the temperature is 50°C. At a second point on the same streamline the temperature is 25°C. What is the velocity at the second point? Check if your answer is valid.

13.7 EFFECT OF AREA VARIATION ON ONE-DIMENSIONAL COMPRESSIBLE FLOW

In steady flow the velocity of an *incompressible* fluid varies inversely with the cross-sectional area of the flow stream. This is not the case with a compressible fluid because variations in density will also influence the velocity. Moreover, the behavior of a compressible fluid, when there is a change in cross-sectional area, depends on whether the flow is ***subsonic*** ($\mathbf{M} < 1$) or ***supersonic*** ($\mathbf{M} > 1$). We shall now examine this phenomenon. In this discussion we shall confine our remarks to ideal flow.

Taking the logarithm of the continuity equation (13.5) and differentiating gives

$$\frac{dA}{A} + \frac{d\rho}{\rho} + \frac{dV}{V} = 0 \tag{13.37}$$

Noting that $c^2 = dp/d\rho$ for constant entropy (Eq. 13.13), the Euler equation (13.9) for an isentropic flow of an *ideal* fluid may be expressed as

$$\frac{dp}{d\rho}\frac{d\rho}{\rho} + VdV = c^2\frac{d\rho}{\rho} + VdV = 0 \tag{13.38}$$

Combining this with Eq. (13.37) to eliminate $d\rho/\rho$, using Eq. (13.10) to replace V/c with $\mathbf{M}$, and rearranging, we get

$$\frac{dV}{V} = \frac{1}{\mathbf{M}^2 - 1}\frac{dA}{A} \tag{13.39}$$

From this equation, we can arrive at some significant conclusions as follows:

1. For subsonic flow ($\mathbf{M} < 1$):

   If $dA/A < 0$, $dV/V > 0$ (decreasing area causes velocity to increase)
   If $dA/A > 0$, $dV/V < 0$ (increasing area causes velocity to decrease)

2. For supersonic flow ($\mathbf{M} > 1$):

   If $dA/A < 0$, $dV/V < 0$ (decreasing area causes velocity to decrease)
   If $dA/A > 0$, $dV/V > 0$ (increasing area causes velocity to increase)

3. For sonic flow ($\mathbf{M} = 1$): $dA/A = 0$

So we see that subsonic and supersonic flows behave *oppositely* if there is an area variation. To accelerate a flow at subsonic velocity, a converging passage is required, just as in the case of an incompressible flow. To accelerate a flow at supersonic velocity, however, we require a diverging passage.

Note that for sonic velocity $dA/A = 0$. This condition occurs at the throat of a converging or a converging-diverging passage. However, the flow will be sonic at the throat only if the pressure differential between the upstream region and the throat is large enough to accelerate the flow sufficiently. At modest pressure differentials the velocity at the throat will be subsonic ($\mathbf{M} < 1$). As the pressure differential between the upstream region and the throat is increased, the velocity at the throat will increase until sonic velocity ($\mathbf{M} = 1$) occurs there. With further increase in pressure differential, the flow rate will increase (due to density increase) but the velocity at the throat will remain sonic. We examine this further in Secs. 13.8 and 13.9. Supersonic flow ($\mathbf{M} > 1$) will occur in the diverging section of a converging-diverging passage only if the flow at the throat is sonic. If the flow at the throat is subsonic, the flow in the diverging section will also be subsonic and the velocity will decrease in the diverging section. In Fig. 13.4 we see the behavior of subsonic and supersonic flow through converging and diverging passages.

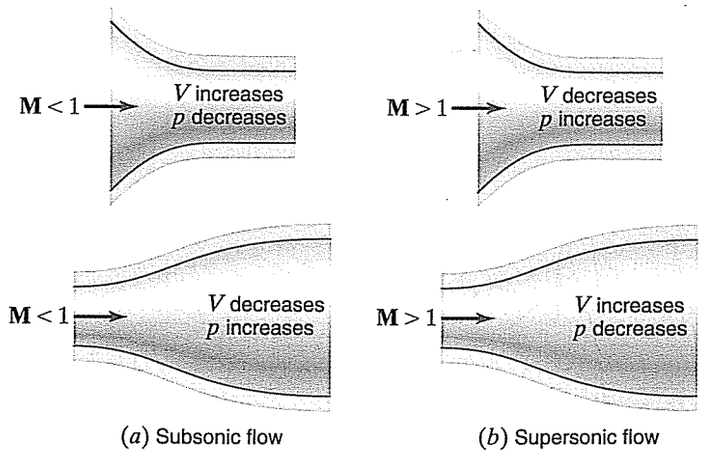


Figure 13.4
Effect of area variation on compressible flow.

EXERCISE

13.7.1 Show in detail the development of Eq. (13.39) from Eqs. (13.38) and (13.37).

13.8 COMPRESSIBLE FLOW THROUGH A CONVERGING NOZZLE

Let us now consider one-dimensional flow of a compressible fluid through the converging nozzle of Fig. 13.5. Since there is little opportunity for heat transfer between sections 1 and 2 and since frictional effects are small, we shall assume isentropic conditions. If the velocity of approach is negligible, p_1 becomes the stagnation pressure p_o in Eq. (13.35), which can then be expressed as

$$\frac{V_2^2}{2} = \frac{p_2}{\rho_2}\frac{k}{k-1}\left[\left(\frac{p_o}{p_2}\right)^{(k-1)/k} - 1\right] \tag{13.40}$$

Noting from Sec. 13.3 that $c_2 = \sqrt{kp_2/\rho_2}$, Eq. (13.40) can be rearranged to give

$$\left(\frac{V_2}{c_2}\right)^2 = \mathbf{M}_2^2 = \frac{2}{k-1}\left[\left(\frac{p_o}{p_2}\right)^{(k-1)/k} - 1\right] \tag{13.41}$$

Alternatively, we can obtain this by rearranging Eq. (13.27). From Eq. (13.41), we note that the velocity of flow at the throat (section 2 of the figure) depends on the p_o/p_2 ratio. If there is a large enough pressure differential between p_o and the ***back pressure*** p_b, sonic velocity will occur at section 2. From the discussion

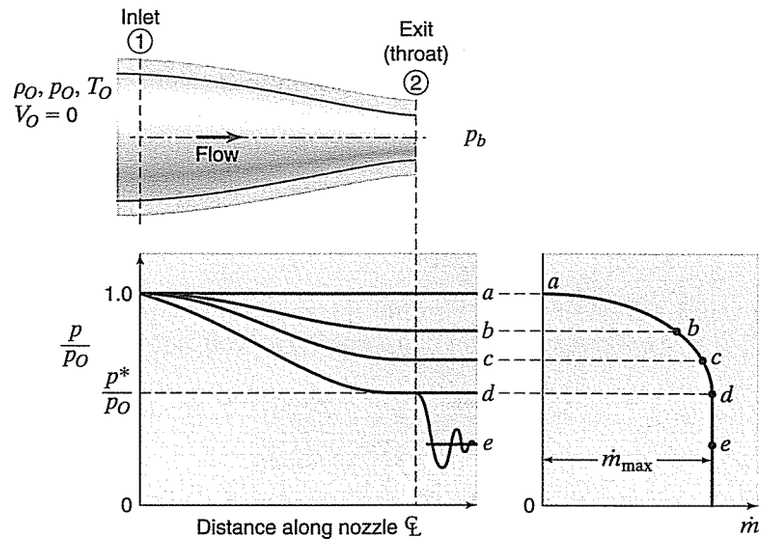


Figure 13.5
Characteristics of compressible flow through a converging nozzle.

of Sec. 13.7, it is recognized that supersonic flow is impossible in this situation. If the flow through the throat is subsonic, the pressure at the throat is identical with that downstream of the nozzle ($p_2 = p_b$, the back pressure). If the flow through the throat is sonic, the pressure at the throat may be equal to, but is generally greater than, that downstream ($p_2 \geqslant p_b$).

Let us now assume the condition of sonic flow at the throat (that is, $\mathbf{M}_2 = 1.0$). Substituting $\mathbf{M}_2 = 1.0$ into Eq. (13.41) and rearranging, we get the ***critical pressure ratio***[2]

$$\frac{p_2^*}{p_o} = \left(\frac{2}{k+1}\right)^{k/(k-1)} \tag{13.42}$$

which is the same as (13.33). This critical pressure ratio exists whenever the pressure differential between p_1 and p_2 is great enough to create sonic velocity at the throat. If the flow through the throat is subsonic, then p_2/p_o will be larger than the ratio given by Eq. (13.42). We note that p_2/p_o can never be smaller than p_2^*/p_o.

The rate of flow through the nozzle of Fig. 13.5 may be found by substituting V_2 from Eq. (13.40) into $\dot{m} = \rho_2 A_2 V_2$. Making use of the isentropic relation between the p's and ρ's and rearranging, we get

Subsonic flow at throat:
$$\dot{m} = A_2\sqrt{\frac{2k}{k-1}p_o\rho_o\left[\left(\frac{p_2}{p_o}\right)^{2/k} - \left(\frac{p_2}{p_o}\right)^{(k+1)/k}\right]} \tag{13.43}$$

This expression is applicable so long as $p_2/p_o > p_2^*/p_o$ as given by Eq. (13.42). Thus it applies to the condition where there is subsonic flow at the throat.

The maximum rate of flow through the nozzle occurs when there is sonic flow at the throat. Substituting $p_2/p_o = p_2^*/p_o = [2/(k+1)]^{k/(k-1)}$ into Eq. (13.43) gives

Sonic flow at throat:
$$\dot{m}_{max} = A_2\sqrt{p_o\rho_o k\left(\frac{2}{k+1}\right)^{(k+1)/(k-1)}} \tag{13.44}$$

Introducing Eq. (2.4), we can also express this equation as

Sonic flow at throat:
$$\dot{m}_{max} = \frac{A_2 p_o}{\sqrt{T_o}}\sqrt{\frac{k}{R}\left(\frac{2}{k+1}\right)^{(k+1)/(k-1)}} \tag{13.45}$$

In this equation the expression under the second radical depends only on the properties of the gas. Thus a simple device for metering compressible flows is a converging nozzle at whose throat the sonic velocity is produced. It should be noted that with subsonic flow, as p_o is increased, the value of p_2/p_o becomes smaller and approaches p_2^*/p_o. At the point where p_2/p_o first reaches the value of p_2^*/p_o the flow at the throat changes from subsonic to sonic. At this point, a threshold point, Eq. (13.43) for subsonic flow and Eqs. (13.44) and (13.45) for sonic flow give the same result. As p_o is increased beyond the threshold point,

[2] In all equations here the pressure p is absolute pressure.

p_2/p_O maintains the value of p_2^*/p_O and the velocity at the throat remains sonic. However, the flow rate increases directly with p_O, as we can see from Eq. (13.45).

For air ($k = 1.4$), according to Eqs. (13.32) and (13.33), $T_2^*/T_O = 0.833$ and $p_2^*/p_O = 0.528$. We have assumed isentropic conditions in the preceding equations; hence the flows represent those for an ideal fluid. The flows for real fluids through converging nozzles are only slightly less than those given by these equations.

From all of this, we can summarize for compressible flow through a converging nozzle:

$$\text{For } p_b \leq p_O\left(\frac{p_2^*}{p_O}\right) = p_2^*, \text{ sonic flow at throat, } p_2 = p_2^* \geqslant p_b$$

$$\text{For } p_b > p_O\left(\frac{p_2^*}{p_O}\right) = p_2^*, \text{ subsonic flow at throat, } p_2 = p_b > p_2^*$$

SAMPLE PROBLEM 13.4 Air at 80°F flows from a large tank through a converging nozzle of 2.0-in exit diameter. The discharge is to an atmospheric pressure of 13.5 psia. Determine the mass flow rate through the nozzle for pressures within the tank of 5, 10, 15, and 20 psig. Assume isentropic conditions. Plot $\dot{m}$ as a function of p_O. Assume that the temperature within the tank is 80°F in all cases.

Solution

From Eq. (13.42), the critical pressure ratio for air is $p_2^*/p_O = 0.528$. If the flow at the throat is subsonic, $p_2/p_O > p_2^*/p_O$. Thus for subsonic flow at the throat, $p_2/p_O > 0.528$ and $p_b = p_2$. So $p_b/p_O > 0.528$ and $p_O < p_b/0.528$.

Since $p_b = 13.50$ psia, the flow at the throat will be subsonic if $p_O < 25.6$ psia (12.1 psig) and sonic if $p_O > 25.6$ psia (12.1 psig).

To find the flow rate for conditions where $p_O < 12.1$ psig (subsonic flow at throat), we use Eq. (13.43). Substituting the appropriate value of p_O into the equation and noting that for this condition $p_2 = p_b = 13.50$ psia, we get

For $p_O = 5$ psig (18.5 psia): $\dot{m} = 0.0374$ slug/sec ***ANS***

For $p_O = 10$ psig (23.5 psia): $\dot{m} = 0.0525$ slug/sec ***ANS***

To find the flow rate for conditions where $p_O > 12.1$ psig (sonic flow at the throat) we use Eq. (13.45). We get

For $p_O = 15$ psig (28.5 psia): $\dot{m}_{\max} = 0.0634$ slug/sec ***ANS***

For $p_O = 20$ psig (33.5 psia): $\dot{m}_{\max} = 0.0745$ slug/sec ***ANS***

Substituting $p_O = 25.6$ psia in Eq. (13.43) for subsonic flow gives $\dot{m} = 0.0571$ slug/sec as does Eq. (13.44) or Eq. (13.45). This is the threshold point at which the flow in the throat changes from subsonic to sonic. When $p_O = 25.6$ psia the flow rate as found from Eq. (13.45) is

$$(\dot{m}_{\max})_{p_b = 25.6 \text{ psia}} = \frac{(0.0218)144(25.6)}{\sqrt{540}}\sqrt{\frac{1.4}{1715}\left(\frac{2}{24}\right)^{2.4/0.4}} = 0.0571 \text{ slug/sec}$$

POLITECNICA SALESIANA
BIBLIOTECA

As p_O is increased beyond 12.1 psig, sonic flow prevails at the throat and the flow rate increases linearly with $(p_O)_{\text{abs}}$ as indicated by Eq. (13.45). The variation of the flow rate with p_O is shown in Fig. S13.4. We can find other information

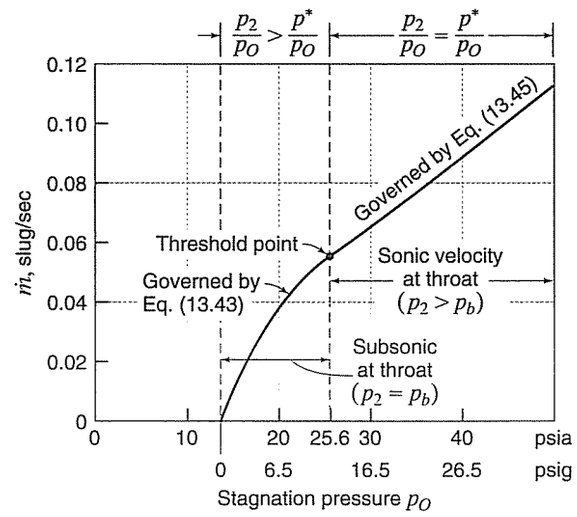


Figure S13.4

concerning various aspects of this problem by applying the equations that have been presented, namely, the gas law ($pv = RT$ or $\rho = p/RT$), the process equation (pv^k = constant or p/ρ^k = constant), continuity ($\dot{m} = \rho AV$), and the energy equation, Eq. (13.40). Applying these, for example, for the case where $p_O = 5$ psig (18.5 psia) yields $\rho_O = 0.00288$ slug/ft^3, $\rho_2 = 0.00230$ slug/ft^3, $V_2 = 747$ ft/sec, $T_2 = 494$°R, and $p_2 = 13.50$ psia. Note that in this case $p_2 = p_b$.

EXERCISES

13.8.1 Start with Eq. (13.40) and derive Eq. (13.43).

13.8.2 Differentiate Eq. (13.43) with respect to p_2/p_O and set to zero to find the value of p_2/p_O for which $\dot{m}$ is a maximum. The answer should correspond to Eq. (13.42).

13.8.3 Carbon dioxide within a tank at 40 psia and 80°F discharges through a convergent nozzle into a 14.2-psia atmosphere. Find the velocity, pressure, and temperature at the nozzle outlet. Assume isentropic conditions.

13.8.4 Carbon dioxide within a tank at 280 kPa abs and 25°C discharges through a convergent nozzle into a 98 kPA abs atmosphere. Find the velocity, pressure, and temperature at the nozzle outlet. Assume isentropic conditions.

13.8.5 In Exer. 13.8.3 if the pressure and temperature within the tank had been 20 psia and 100°F, what would have been the velocity, pressure, and temperature at the nozzle outlet? Assume isentropic conditions.

13.8.6 In Exer. 13.8.4 if the pressure and temperature within the tank had been 140 kPa abs and 40°C, what would have been the velocity, pressure, and temperature at the nozzle outlet? Assume isentropic conditions.

13.9 ISENTROPIC FLOW THROUGH A CONVERGING-DIVERGING NOZZLE

If a converging nozzle has attached to it a diverging section, it is possible to attain supersonic velocities in the diverging section. This will happen if sonic flow exists in the throat. In such a case the gas or vapor will continue to expand in the diverging section to lower pressures and the velocity will continue to increase. The flow through such a converging-diverging nozzle is shown in Fig. 13.6. If there is not enough pressure differential to attain sonic velocity at the throat, the gas or vapor will behave in much the same manner as a liquid, with acceleration in the region up to the throat and deceleration in the diverging section beyond the throat. A plot of the pressure in the flow for such a condition is shown by the dashed lines *ABD* of Fig. 13.6.

Suppose in Fig. 13.6 that we gradually reduce the back pressure p_b while p_O remains constant. In such a case $p_3 = p_b$, and the pressure at the throat p_2

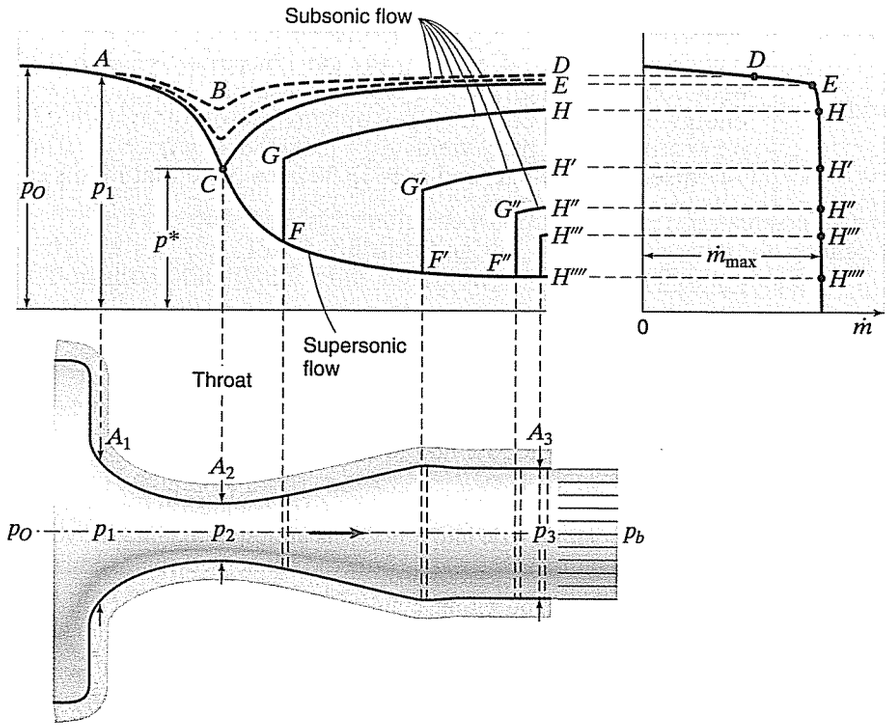


Figure 13.6
Flow of compressible fluid through a converging-diverging nozzle.

decreases while the velocity at the throat increases until the limiting sonic velocity is reached at the throat when the pressure plot is *ACE*. If we further reduce the back pressure to *H*, the pressure plot is *ACFGH*, the jump from *F* to *G* being a pressure shock, or ***normal shock wave*** (normal to the approaching flow, i.e., one-dimensional), which is analogous to the hydraulic jump, or standing wave, often seen in open channels conveying water (Secs. 10.18, 10.20). Through the shock wave the velocity is reduced abruptly from supersonic to subsonic, while at the same time the pressure jumps as shown by the lines *FG*, *F'G'*, and *F"G"*. The flow through a shock wave is *not isentropic,* since part of the kinetic energy is converted into heat.

Further reduction of the back pressure p_b causes the shock wave to move farther downstream until at some value given by H''' the shock wave is located at the downstream end of the nozzle. If we lower p_b below the level of H''', the shock wave occurs in the flow field downstream of the nozzle exit. Such flow fields are either two- or three-dimensional and cannot be described by the foregoing one-dimensional equations.

If we lower the back pressure to H'''', the flow will proceed isentropically to supersonic throughout the entire region downstream from the throat, the velocity will increase continuously from section 0 to its maximum value at section 3, and the pressure will drop continuously from 0 to 3. As long as p_b is above H'''' then $p_3 = p_b$; but if p_b drops below H'''', then $p_3 > p_b$ and supersonic flow occurs through the entire length of the divergent portion of the nozzle.

If the back pressure is above *E* in Fig. 13.6, the flow rate through the converging-diverging nozzle is given by Eq. (13.43). In this instance we must replace the p_2 of Eq. (13.43) by the p_b of Fig. 13.6. If the back pressure is below *E* in Fig. 13.6, critical pressure, as defined by Eq. (13.42), will occur at the throat and the flow rate will be given by Eq. (13.44) or Eq. (13.45).

If the stagnation pressure p_O increases, we can show that the sonic velocity remains unaltered, but since the density of the gas increases, the rate of discharge will be greater. The converging nozzle and the converging-diverging nozzle are alike as far as discharge capacity is concerned. The only difference is that with the converging-diverging nozzle, a supersonic velocity may be attained at discharge from the device, while with the converging nozzle, the sonic velocity is the maximum value possible.

SAMPLE PROBLEM 13.5 Air discharges from a large tank through a converging-diverging nozzle with a 2.0-in-diameter throat. Within the tank the pressure is 50 psia and the temperature is 80°F, while outside the tank the pressure is 13.5 psia. The nozzle is to operate with supersonic flow throughout its diverging section with a 13.5 psia pressure at its outlet. Find the required diameter of the nozzle outlet. Determine the mass flow rate and the velocities and temperatures at sections 2 and 3. Assume isentropic flow.

Solution

The pressure at the throat must be such that sonic velocity will occur there. Hence from Eq. (13.33), $p_2 = p_2^* = 0.528p_O = 0.528(50) = 26.4$ psia.

We can find the velocity V_3 at the outlet from Eq. (13.35) with $V_O = 0$:

$$\frac{V_3^2}{2} = RT_O \frac{k}{k-1}\left[1 - \left(\frac{p_3}{p_O}\right)^{(k-1)/k}\right] = 1715(540)\frac{1.4}{0.4}\left[1 - \left(\frac{13.5}{50}\right)^{0.4/1.4}\right]$$

$$= 1{,}012{,}000 \text{ ft·lb/slug (or ft}^2\text{/sec}^2\text{)}$$

from which $V_3 = 1420$ ft/sec **ANS**

Computing the mass flow rate from Eq. (13.45):

$$\dot{m}_{\max} = \frac{0.0218(50 \times 144)}{\sqrt{540}}\sqrt{\frac{1.4}{1715}\left(\frac{2}{2.4}\right)^{2.4/0.4}} = 0.1117 \text{ slug/sec} \quad \textbf{ANS}$$

We determine the temperature at 3 by using Eq. (13.23) with $V_O = 0$:

$$V_3^2 - V_O^2 = 2c_p(T_O - T_3)$$

$$(1420)^2 = 2(6000)(540 - T_3)$$

$$T_3 = 371°\text{R} = -89°\text{F} \quad \textbf{ANS}$$

From the perfect-gas law Eq. (2.4), $p_3/\rho_3 = RT_3$:

$$\rho_3 = \frac{13.5 \times 144}{1715(371)} = 0.00305 \text{ slug/ft}^3$$

We can assume isentropic flow between 2 and 3, since the shock wave does not occur within that region. Thus $p_2/\rho_2^{1.4} = p_3/\rho_3^{1.4}$:

$$\frac{26.4}{\rho_2^{1.4}} = \frac{13.5}{(0.00305)^{1.4}}$$

$$\rho_2 = 0.00493 \text{ slug/ft}^3$$

Now we can compute the velocity at 2:

$$V_2 = \frac{\dot{m}}{\rho_2 A_2} = \frac{0.1117}{0.00493(0.0218)} = 1040 \text{ ft/sec} \quad \textbf{ANS}$$

We can find the temperature at 2 from Eq. (13.23):

$$(1420)^2 - (1040)^2 = 2(6000)(T_2 - 371)$$

$$T_2 = 450°\text{R} = -10°\text{F} \quad \textbf{ANS}$$

The area at 3 is computed from

$$A_3 = \frac{\dot{m}}{\rho_3 V_3} = \frac{0.1117}{0.00305(1420)} = 0.0257 \text{ ft}^2$$

Finally, $D_3 = 0.1809$ ft $= 2.17$ in, the required outlet diameter. **ANS**

Check for sonic velocity at throat:

$$c_2 = \sqrt{kRT_2} = \sqrt{1.4(1715)450} = 1040 \text{ ft/sec} = V_2$$

With sonic velocity at the throat, if $D_3 < 2.17$ in, there will be supersonic flow throughout the divergent portion of the nozzle and a shock wave will occur in the flow field downstream of the nozzle exit. If $D_3 > 2.17$ in, with sonic flow in the throat, in order to satisfy pressure conditions, a shock wave will occur in the tube somewhere between the throat and the nozzle exit.

EXERCISES

13.9.1 Air enters a converging-diverging nozzle at a pressure of 120 psia and a temperature of 90°F. Neglecting the entrance velocity and assuming a frictionless process, find the Mach number at the cross section where the pressure is 35 psia.

13.9.2 Air enters a converging-diverging nozzle at a pressure of 830 kPa abs and a temperature of 32°C. Neglecting the entrance velocity and assuming a frictionless process, find the Mach number at the cross section where the pressure is 240 kPa abs.

13.9.3 Work Sample Prob. 13.5 with all data the same except for the pressure within the tank, which is 100 rather than 50 psia.

13.9.4 Air is to flow through a converging-diverging nozzle at 0.6 slug/sec. At the throat the pressure, temperature, and velocity are to be 20 psia, 100°F, and 500 ft/sec, respectively. At outlet the velocity is to be 200 ft/sec. Determine the throat diameter. Assume isentropic flow.

13.9.5 Air in a tank at a pressure of 140 psia and 70°F flows out into the atmosphere through a 1.00-in-diameter converging nozzle. (*a*) Find the mass flow rate. (*b*) If a diverging section with an outlet diameter of 1.50 in were attached to the converging nozzle, what then would be the flow rate? Neglect friction.

13.9.6 Air in a tank at a pressure of 950 kPa abs and 20°C flows out into the atmosphere through a 25-mm-diameter converging nozzle. (*a*) Find the mass flow rate. (*b*) If a diverging section with an outlet diameter of 40 mm were attached to the converging nozzle, what then would be the flow rate? Neglect friction.

13.10 ONE-DIMENSIONAL SHOCK WAVE

In Fig. 13.7 we see a one-dimensional shock wave where the approaching supersonic flow (subscript 1) changes to subsonic flow (subscript 2). This phenomenon is accompanied by a sudden rise in pressure, density, and temperature. Applying the momentum principle to the fluid in the shock wave, we get

$$\sum F_x = p_1A_1 - p_2A_2 = \dot{m}(V_2 - V_1) \tag{13.46}$$

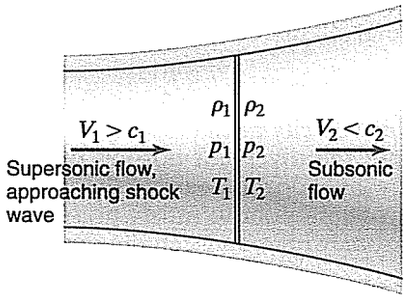


Figure 13.7
A one-dimensional normal shock wave.

Substituting the continuity conditions ($\dot{m} = \rho_1 A_1 V_1 = \rho_2 A_2 V_2$) and noting that $A_1 = A_2$, so that $\rho_1 V_1 = \rho_2 V_2$, we get

$$p_2 - p_1 = \rho_1 V_1^2 - \rho_2 V_2^2 = \rho_1 V_1 (V_1 - V_2) = \rho_2 V_2 (V_1 - V_2) \qquad (13.47)$$

which is the pressure jump across the wave.

We can consider the flow across the shock wave to be adiabatic, and can express it as

$$V_2^2 - V_1^2 = \frac{2k}{k-1}(p_1 v_1 - p_2 v_2) \qquad (13.48)$$

This is identical to Eq. (13.24). We suggest that the reader review the development of this equation, in Sec. 13.4.

Equations (13.47) and (13.48) may be solved simultaneously and rearranged algebraically to give some significant relationships. Several such relations are as follows:

$$\frac{p_2}{p_1} = \frac{2k\mathbf{M}_1^2 - (k-1)}{k+1} \qquad (13.49)$$

$$\frac{T_2}{T_1} = \frac{[2k\mathbf{M}_1^2 - (k-1)][2 + (k-1)\mathbf{M}_1^2]}{(k+1)^2 \mathbf{M}_1^2} \qquad (13.50)$$

$$\frac{V_2}{V_1} = \frac{\rho_1}{\rho_2} = \frac{(k-1)\mathbf{M}_1^2 + 2}{(k+1)\mathbf{M}_1^2} \qquad (13.51)$$

and

$$\mathbf{M}_2^2 = \frac{2 + (k-1)\mathbf{M}_1^2}{2k\mathbf{M}_1^2 - (k-1)} \qquad (13.52a)$$

or

$$\mathbf{M}_1^2 = \frac{2 + (k-1)\mathbf{M}_2^2}{2k\mathbf{M}_2^2 - (k-1)} \qquad (13.52b)$$

These equations allow us to find the physical properties of the flow on the two sides of the one-dimensional shock wave. These equations, of course, are

TABLE 13.2 **One-dimensional normal shock functions for an ideal gas with $k = 1.4$**

$\mathbf{M}_1$	$\mathbf{M}_2$	T_2/T_1	p_2/p_1	ρ_2/ρ_1
1.00	1.00000	1.00000	1.00000	1.00000
1.10	0.91177	1.06494	1.24500	1.16908
1.20	0.84217	1.12799	1.51333	1.34168
1.30	0.78596	1.19087	1.80500	1.51570
1.40	0.73971	1.25469	2.12000	1.68966
1.50	0.70109	1.32022	2.45833	1.86207
1.60	0.66844	1.38797	2.82000	2.03175
1.70	0.64055	1.45833	3.20500	2.19772
1.80	0.61650	1.53158	3.61333	2.35922
1.90	0.59562	1.60792	4.04500	2.51568
2.00	0.57735	1.68750	4.50000	2.66667
2.10	0.56128	1.77045	4.97833	2.81190
2.20	0.54706	1.85686	5.48000	2.95122
2.30	0.53441	1.94680	6.00500	3.08455
2.40	0.52312	2.04033	6.55333	3.21190
2.50	0.51299	2.13750	7.12500	3.33333
2.60	0.50387	2.23834	7.72000	3.44898
2.70	0.49563	2.34289	8.33833	3.55899
2.80	0.48817	2.45117	8.98000	3.66355
2.90	0.48138	2.56321	9.64500	3.76286
3.00	0.47519	2.67901	10.33333	3.85714
4.00	0.43496	4.04688	18.50000	4.57143
5.00	0.41523	5.80000	29.00000	5.00000

applicable only if $\mathbf{M}_1 > 1$; that is, the oncoming flow must be supersonic. We can see that the shock wave is analogous to the hydraulic jump in open-channel flow (Sec. 10.18). Properties across a shock are given numerically in Table 13.2 and presented qualitatively in Fig. 13.8, in terms of $\mathbf{M}_1$.

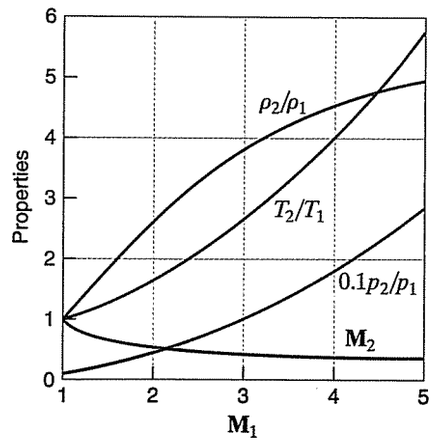


Figure 13.8
Properties across a shock wave for one-dimensional flow of a perfect gas with $k = 1.4$ (see also Table 13.2).

SAMPLE PROBLEM 13.6 A normal shock wave occurs in the flow of air where $p_1 = 10$ psia (70 Pa abs), $T_1 = 40°F$ (5°C), and $V_1 = 1400$ ft/sec (425 m/s). Find p_2, V_2, and T_2.

Solution (in BG units)

$$\rho_1 = \frac{p_1}{RT_1} = \frac{10(144)\ \text{lb/ft}^2\ \text{abs}}{1715\ \text{ft·lb/(slug·°R)}(460 + 40)°\text{R}} = 0.00168\ \text{slug/ft}^3$$

$$c_1 = \sqrt{kRT_1} = \sqrt{1.4 \times 1715 \times (460 + 40)} = 1096\ \text{ft/sec}$$

$$\mathbf{M}_1 = \frac{V_1}{c_1} = \frac{1400}{1096} = 1.28$$

From Eq. (13.49): $\dfrac{p_2}{p_1} = 1.75, \quad p_2 = 17.5$ psia

From Eq. (13.51): $\dfrac{V_2}{V_1} = 0.675, \quad V_2 = 945$ ft/sec ***ANS***

$$\rho_1 V_1 = \rho_2 V_2, \quad \rho_2 = \frac{0.00168}{0.675} = 0.00249\ \text{slug/ft}^3 \quad \textit{ANS}$$

$$pv = \frac{p}{\rho} = RT, \quad T_1 = \frac{p_2}{\rho_2 R} = \frac{17.5(144)}{0.00249(1715)} = 590°\text{R} = 130°\text{F} \quad \textit{ANS}$$

Solution (in SI units)

$$\rho_1 = \frac{p_1}{RT_1} = \frac{70\ \text{N/m}^2\ \text{abs}}{287\ \text{N·m/(kg·K)}(273 + 5)\ \text{K}} = 8.8 \times 10^{-4}\ \text{kg/m}^3$$

$$c = \sqrt{kRT_1} = \sqrt{1.4 \times 287 \times (273 + 5)} = 334\ \text{m/s}$$

$$\mathbf{M}_1 = \frac{V_1}{c_1} = \frac{425}{334} = 1.27$$

From Eq. (13.49): $\dfrac{p_2}{p_1} = 1.75, \quad p_2 = 122.5$ Pa (or N/m^2) abs

From Eq. (13.51): $\dfrac{V_2}{V_1} = 0.675, \quad V_2 = 286$ m/s ***ANS***

$$\rho_1 V_1 = \rho_2 V_2, \quad \rho_2 = \frac{8.8 \times 10^{-4}}{0.675} = 1.3 \times 10^{-3}\ \text{kg/m}^3 \quad \textit{ANS}$$

$$T_2 = \frac{p_2}{\rho_2 R} = \frac{122.5}{(1.3 \times 10^{-3})287} = 328\ \text{K} = 55°\text{C} \quad \textit{ANS}$$

EXERCISES

13.10.1 Just downstream of a normal shock wave the pressure, velocity, and temperature are 52 psia, 400 ft/sec and 120°F. Compute the Mach number upstream of the shock wave. Consider (*a*) air and (*b*) carbon dioxide as the working fluids.

13.10.2 Just downstream of a normal shock wave the pressure, velocity, and temperature are 360 kPa abs, 110 m/s and 50°C. Compute the Mach number upstream of the shock wave. Consider (*a*) air and (*b*) carbon dioxide as the working fluids.

13.11 THE OBLIQUE SHOCK WAVE

When the velocity of a body passing through any fluid, whether a liquid or a gas, exceeds that of a sound wave in the same fluid, the flow conditions are entirely different from those for subsonic flow. Thus, instead of streamlines such as are shown in Fig. 4.12, the conditions are as shown in Fig. 13.9, which is a schlieren photograph[3] of supersonic flow past a sharp-nosed model in a wind tunnel. It could also represent a projectile in flight through still air. A conical compression or shock wave extends backward from the tip, as we can see by the strong density gradient revealed as a bright shadow in the photograph. A streamline in the undisturbed fluid is unaffected by the solid boundary or by a moving projectile until it intersects a shock-wave front, when it is abruptly changed in direction, proceeding roughly parallel to the nose form. Where the conical nose is joined

[3] The schlieren method employs the fact that the refraction of light is disturbed by a pressure (density) gradient. Details of the optics of flow visualization are discussed in Sec. 11.6 of Ref. 75 in Appendix E.

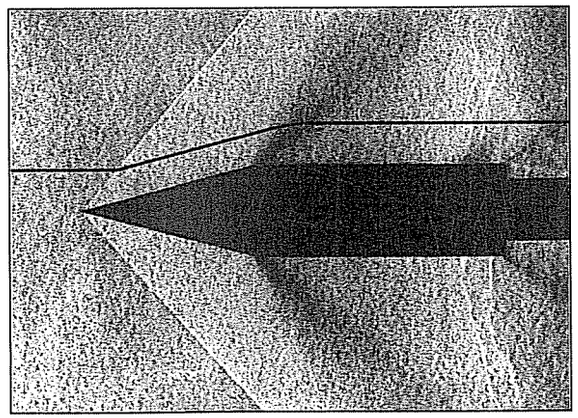


Figure 13.9
Schlieren photograph of the head wave on a 30° (total angle) cone at $\mathbf{M} = 1.88$. (Courtesy of the Archives, California Institute of Technology)

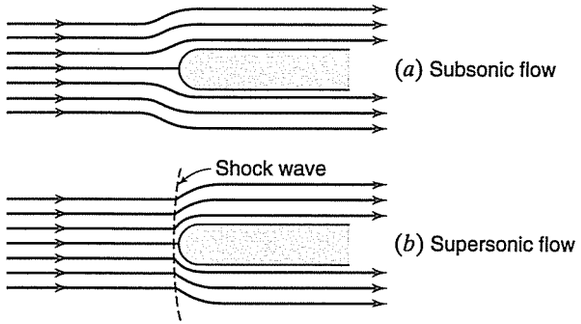


Figure 13.10
Flow pattern upstream of a body.

to the cylindrical body of the model, dark shadows may be seen, representing rarefaction waves. The streamlines are again changed in direction through this region, becoming parallel to the main flow again. A typical streamline has been superimposed on the photograph. Figure 13.10 shows the difference between a subsonic and supersonic flow pattern upstream of a body.

The reason why streamlines are unaffected in front of a projectile is that the body travels faster than the disturbance can be transmitted ahead. This is illustrated in Fig. 13.11. Consider a point source of an infinitesimal disturbance moving at a supersonic velocity V through a fluid. At the instant when this source passes through the point A_0, the disturbance commences to radiate in all directions with the velocity c of a sound wave in this medium. In successive instants the source passes through the points A_1, A_2, and A_3, the last of which can represent the position of the source at the instant of observation. While the source has covered the distance A_0A_3 with velocity V, the sound wave, traveling at the

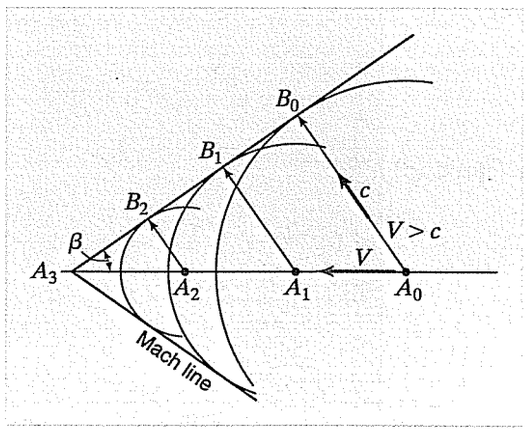


Figure 13.11
Schematic diagram of a disturbance moving with supersonic velocity.

slower acoustic velocity c, has progressed only as far as radius A_0B_0. Similar termini of disturbances emanating from A_1 and A_2 form a straight-line envelope, which is the shock wave. The angle β is called the ***Mach angle,*** and it is seen that

$$\sin\beta = \frac{A_0B_0}{A_0A_3} = \frac{c}{V} = \frac{1}{\mathbf{M}} \tag{13.53}$$

where **M**, the dimensionless velocity ratio V/c, is the Mach number.

In the case of the finite projectile of Fig. 13.9, because of the conical nose that follows the tip of the projectile, the shock wave leaves the tip at an angle with the main flow which exceeds the Mach angle. Appropriate corrections[4] can be applied, however, and the shock-wave angle from such a sharp-nosed object can serve as an accurate means of measuring supersonic velocities. For a blunt-nosed object (Fig. 13.10) the shock wave is a combination of partly normal and partly oblique shock waves.

EXERCISES

13.11.1 Assuming the tip of the model in Fig. 13.9 to be a point source of infinitesimal disturbance, find the air velocity if the temperature is −60°F and $k = 1.4$. If the actual Mach number is 1.38, what is the percentage error involved in the preceding assumption?

13.11.2 A schlieren photograph of a bullet shows a Mach angle of 40°. The air is at a pressure of 14 psia and 50°F. Find the approximate speed of the bullet.

13.11.3 A schlieren photograph of a bullet shows a Mach angle of 25°. The air is at a pressure of 9.5 kPa abs and 20°C. Find the approximate speed of the bullet.

13.12 ISOTHERMAL FLOW

Most of the flows discussed thus far in this chapter are characterized by substantial changes in temperature. In isentropic flow (pv^k = constant) through a nozzle, for example, large changes in temperature occur because the variation in cross-sectional flow area causes substantial changes in V, p, ρ, and T.

Let us now examine some aspects of isothermal (T = constant) flow. An example application is gas flow through buried pipelines, where the ground serves as a heat sink to maintain pipeline temperatures constant. In this case the internal energy $i_1 = i_2$, and $pv = RT$ = constant so that p/ρ = constant. As a result Eq. (5.32) dictates that $h_1 = h_2$ and Eq. (13.6) reduces to

$$\frac{V_1^2}{2} + q_H = \frac{V_2^2}{2} \tag{13.54}$$

where q_H is the heat interaction per unit mass.

[4] Donald A. Gilbrech, *Fluid Mechanics,* p. 405, Wadsworth Publishing Co., Belmont, California, 1965.

This equation is applicable to both real and ideal compressible fluids. For the case of isothermal compressible flow of a real fluid in a pipe of uniform diameter (Sec. 13.13) there is a drop in pressure along the pipe because of fluid friction, and thus a decrease in density. To maintain continuity, the velocity must increase along the pipe and thus the kinetic energy also increases. Reference to Eq. (13.54) shows that, with $A_1 = A_2$, heat energy must be added to the fluid to maintain isothermal flow in a pipe of uniform diameter.

13.13 ISOTHERMAL FLOW IN A CONSTANT-AREA DUCT

To obtain some understanding of the characteristics of compressible flow with friction, let us consider isothermal flow in a pipe of uniform diameter D. We can assume that the flow of a gas through a long, uninsulated pipe in an isothermal environment approximates isothermal flow, since the temperature rise created by fluid friction is very small. From energy Eq. (5.10) for an elemental length ds of pipe, we have

$$\frac{dp}{\rho} + g\,dz + V\,dV + \frac{\tau P}{\rho A}ds = 0$$

But from Eqs. (8.8) and (8.3) we can show that

$$\frac{\tau P}{\rho A} = g\frac{h_f}{L}$$

and eliminating h_f using the Darcy-Weisbach equation (8.13)

$$\frac{\tau P}{\rho A} = \frac{f}{D}\frac{V^2}{2}$$

So, substituting this into the energy equation, we obtain

$$\frac{dp}{\rho} + g\,dz + V\,dV + \frac{f\,ds}{D}\frac{V^2}{2} = 0$$

As we are here dealing with a gas, we may disregard variations in z, so that the energy equation becomes

$$\frac{dp}{\rho} + V\,dV + \frac{f\,ds}{D}\frac{V^2}{2} = 0 \tag{13.55}$$

Recalling from Eqs. (13.10) and (13.15) that the Mach number $\mathbf{M} = V/c$, where $c = \sqrt{kp/\rho} = \sqrt{kRT} =$ constant for isothermal flow, we can restate Eq. (13.55) as

$$\frac{dp}{p} + k\mathbf{M}^2\left(\frac{d\mathbf{M}}{\mathbf{M}} + f\frac{ds}{2D}\right) = 0 \tag{13.56}$$

Investigators have determined that the f values given in Fig. 8.11 are applicable to compressible flow if $\mathbf{M} < 1$. for $\mathbf{M} > 1$, the values for f are about one-half of those of Fig. 8.11.

From continuity $\rho AV = \dot{m} =$ constant, and from the perfect-gas law (Eq. 2.4) $\rho = p/RT$. By eliminating ρ, we get $pV = \dot{m}RT/A =$ constant for steady

isothermal flow in a constant-area duct. Substituting this expression ($pV =$ constant) into Eq. (13.56) and rearranging, we get for isothermal flow in a duct of constant cross section,

$$\frac{dp}{p} = -\frac{dV}{V} = -\frac{d\mathbf{M}}{\mathbf{M}} = -\frac{k\mathbf{M}^2}{1 - k\mathbf{M}^2}f\frac{ds}{2D} \tag{13.57}$$

This equation shows that, when $\mathbf{M} < 1/\sqrt{k}$, the pressure will decrease in the direction of flow. On the other hand, if $\mathbf{M} > 1/\sqrt{k}$, the pressure increases in the direction of flow.

The mass flow rate $\dot{m} = \rho AV$; hence we can express the Reynolds number as

$$\mathbf{R} = \frac{DV\rho}{\mu} = \frac{\dot{m}D}{\mu A} \tag{13.58}$$

At usual pressures the viscosity μ of a gas depends only on temperature and is constant for a given temperature; therefore for steady isothermal flow ($\dot{m} =$ constant and $T =$ constant), the Reynolds number (and therefore also f) is constant along the entire length of a uniform-diameter pipe for any given flow. From the perfect-gas law, $\rho = p/RT$, and from continuity, $V = \dot{m}/\rho A$. Hence $V = \dot{m}RT/pA$. Substituting these values into Eq. (13.55) and rearranging, we obtain

$$-\left(\frac{2A^2}{\dot{m}^2RT}\right)p\,dp = \frac{f}{D}ds + \frac{2\,dV}{V}$$

For isothermal flow in a pipe of constant diameter this equation is readily integrated for a length $L = s_2 - s_1$ to give

$$p_1^2 - p_2^2 = \frac{\dot{m}^2RT}{A^2}\left(f\frac{L}{D} + 2\ln\frac{p_1}{p_2}\right) \tag{13.59}$$

where p_1/p_2 has replaced V_2/V_1 (since $pV =$ constant). We can use Eq. (13.59) to find $\dot{m}$ if all other values are given, but if $\dot{m}$ and other values at (1) are given, we can find p_2 by successive approximation or with an equation solver (Appendixes C.1–C.3 and D.1–D.3). In most cases the last term is small compared with fL/D and we can neglect it as a first approximation. If it proves to be significant, a second solution, using an approximate value of p_1/p_2, can be made if greater accuracy is desired. Note that p and A involve the *same area units,* and so, for numerical work in BG units it is usually more convenient to use p in pounds per square inch and A in square inches.

There is a restriction to Eq. (13.59) similar to that discussed in Sec. 13.8 and inferred in Eq. (13.57). To illustrate, disregarding the logarithmic term, we can express Eq. (13.59) as $p_2^2 = p_1^2 - NL$, where N is a constant for the given conditions and L is the distance along the pipe from section 1 to section 2. According to the equation as L gets larger, p_2 will get smaller until it eventually drops to zero, which, of course, is physically impossible. Actually what happens is that as L gets larger, p gets smaller as does ρ; and since $\rho V =$ constant, V gets larger. However, there is a limit to how large V can get. This occurs when $\mathbf{M} = 1/\sqrt{k}$ as can be seen by examining Eq. (13.57). Thus Eq. (13.59) is applicable as long as $\mathbf{M} < 1/\sqrt{k}$. Another way of stating this is that for isothermal flow in a pipe of constant diameter there is a maximum length of pipe for which the

given isothermal flow will proceed continuously. If the pipe exceeds this limiting length, there is a ***choking*** of the flow that limits the mass flow rate.

Another way of expressing Eq. (13.59) is to divide it by p_1^2, substitute $\rho_1 A_1 V_1$ for $\dot{m}$, and note that $\mathbf{M}_1/\mathbf{M}_2 = p_2/p_1$ for this situation. Rearranging, we get

$$\frac{\mathbf{M}_1^2}{\mathbf{M}_2^2} = 1 - k\mathbf{M}_1^2\left(2\ln\frac{\mathbf{M}_2}{\mathbf{M}_1} + f\frac{L}{D}\right) \tag{13.60}$$

Equation (13.60) is a particularly useful form of the isothermal-flow equation, since we can handily employ it to determine the maximum length of pipe such that the maximum possible velocity (limiting condition) will occur at the downstream end of the pipe. This is demonstrated in Sample Prob. 13.7.

Although these derivations were for a pipe of constant diameter, we can obtain the same results for a constant-area duct. For the duct, we simply replace the pipe radius r by $0.5D_h = 2R_h$ (from Sec. 8.3), and we replace the pipe diameter D by $D_h = 4R_h$.

SAMPLE PROBLEM 13.7 Air flows isothermally at 65°F through a horizontal 10-in by 14-in rectangular duct at 3.0 slug/sec (Fig. S13.7). If the pressure at a section is 80 psia, find the pressure at a second section 500 ft downstream from the first. Assume the duct surface is very smooth; so use the lowest curve of Fig. 8.11 or Eq. (8.47) to determine f.

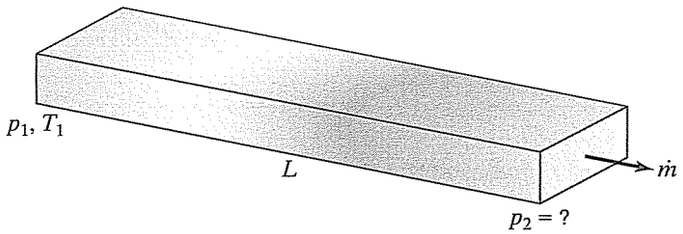


Figure S13.7

Solution

First determine the applicability of Eq. (13.59):

$$R_h = \frac{A}{P} = \frac{140}{48} = 2.92 \text{ in} = 0.243 \text{ ft}$$

Interpolating in Table A.2: Viscosity μ of air at 65°F is 3.78×10^{-7} lb·sec/ft^2.

$$\mathbf{R}_1 = \frac{DV\rho}{\mu} = \frac{\dot{m}D}{\mu A} = \frac{\dot{m}(4R_h)}{\mu A} = \frac{3.0(4 \times 0.243)}{3.78 \times 10^{-7}(10 \times 14)/144} = 7.93 \times 10^6$$

$$\mathbf{R}_2 = \mathbf{R}_1, \quad \text{since} \quad \rho_1 V_1 = \rho_2 V_2 \quad \text{and} \quad \mu_1 = \mu_2$$

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

From Fig. 8.11, $f = 0.0084$.

$$\rho_1 = \frac{p}{RT} = \frac{80 \times 144}{1715(460 + 65)} = 0.01279 \text{ slug/ft}^3$$

$$V_1 = \frac{\dot{m}}{\rho_1 A} = \frac{3.0}{(0.01279)\frac{140}{144}} = 241 \text{ ft/sec}$$

$$c = \sqrt{kRT} = \sqrt{1.4 \times 1715 \times 525} = 1123 \text{ ft/sec}$$

$$\mathbf{M}_1 = \frac{V_1}{c} = \frac{242}{1123} = 0.215$$

The limiting value of $\mathbf{M}_1$ is $1/\sqrt{1.4} = 0.845$. Substituting these into Eq. (13.60), while replacing D by $4R_h$ (Sec. 8.3),

$$\frac{(0.215)^2}{(0.845)^2} = 1 - 1.4(0.215)^2\left[2\ln\frac{0.845}{0.215} + 0.0084\frac{L}{4(0.243)}\right]$$

$$L = 1359 \text{ ft}$$

Thus Eq. (13.59) applies for all values of $L < 1359$ ft. Substituting $L = 500$ ft in Eq. (13.59) and neglecting the usually small logarithmic term,

$$(80 \times 144)^2 - p_2^2 = \frac{(3.0)^2(1715)(525)}{(140/144)^2}\left[0.0084\frac{500}{4(0.243)}\right]$$

from which $p_2 = 67.9$ psia. Substituting this value of p_2 into Eq. (13.59) and considering the logarithmic term yields $p_2 = 66.9$ psia. ***ANS***

Repeating the process again will give a more accurate answer. We can avoid iteration by using equation solving software (Appendixes C.2–C.3 and D.2–D.3) or a programmable calculator with an equation (root) solving capability (Appendixes C.1 and D.1).

SAMPLE PROBLEM 13.8 For the case of Sample Prob. 13.7 with a duct length of 500 ft, compute the thermal energy (heat) that must be added to the fluid to maintain isothermal conditions.

Solution

Since the flow is isothermal, $p_1/\rho_1 = p_2/\rho_2 = RT$ = constant; $p_1 = 80$ psia and from Sample Prob. 13.7, $p_2 = 66.9$ psia. Thus $\rho_1/\rho_2 = 80/66.9 = 1.195$ and $V_2/V_1 = 1.195$, since ρV = constant from continuity.

So $V_2 = 1.195(241) = 288$ ft/sec; applying Eq. (13.54), $q_H = (288)^2/2 - (241)^2/2 = 12{,}480$ ft·lb/slug or 16.04 Btu/slug of air. Since $\dot{m} = 3.0$ slug/sec, the rate at which heat must be added to the fluid is $3.0(16.04) = 48.1$ Btu/s. ***ANS***

Note: If $q_H > 16.04$ Btu/slug of air, then $T_2 > T_1$, and if $q_H < 16.04$ Btu/slug of air, then $T_2 < T_1$.

EXERCISES

13.13.1 Air flows isothermally through a long horizontal pipe of uniform diameter. At a section where the pressure is 100 psia, the velocity is 120 ft/sec. Because of fluid friction the pressure at a distant point is 40 psia. (*a*) What is the increase in kinetic energy per slug of air? (*b*) What is the amount of thermal energy in Btu per slug of air that must be transferred in order to maintain the temperature constant? (*c*) Is this heat transferred to the air in the pipe or removed from it? (*d*) If the temperature of the air is 100°F and the diameter of the pipe is 3 in find the total heat transferred in Btu per hour.

13.13.2 Air flows isothermally through a long horizontal pipe of uniform diameter. At a section where the pressure is 700 kPa abs, the velocity is 35 m/s. Because of fluid friction the pressure at a distant point is 280 kPa abs. (*a*) What is the increase in kinetic energy per kg of air? (*b*) What is the amount of thermal energy in J/kg of air that must be transferred in order to maintain the temperature constant? (*c*) Is this heat transferred to the air in the pipe or removed from it? (*d*) If the temperature of the air is 40°C and the diameter of the pipe is 75 mm find the total heat transferred in joules per hour.

13.14 ADIABATIC FLOW IN A CONSTANT-AREA DUCT

The flow of fluids with friction in well-insulated pipes is a case that approaches adiabatic flow, i.e., $q_H = 0$. The usual applications of insulation are to pipes conveying steam or refrigerating fluids, such as ammonia vapor. In some situations such pipes are short, the pressure drops are relatively small, and we can solve the problem as if the fluid were incompressible. However, there are situations where the effects of compressibility must be considered.

Let us consider the case of steady adiabatic flow with friction in a duct of uniform cross section. Substituting $V_1 = \dot{m}/\rho_1 A$ and $V_2 = \dot{m}/\rho_2 A$ into adiabatic flow Eq. (13.24), we get

$$\frac{\dot{m}^2}{\rho_2^2 A^2} - \frac{\dot{m}^2}{\rho_1^2 A^2} = \frac{2k}{k-1}(p_1 v_1 - p_2 v_2) \tag{13.61}$$

where $\dot{m}$ is the mass rate of flow and A is the cross-sectional area of the duct. This equation can be rewritten as

$$\frac{\dot{m}^2}{\rho_1^2 A^2} + \frac{2k}{k-1} p_1 v_1 = \frac{\dot{m}^2}{\rho_2^2 A^2} + \frac{2k}{k-1} p_2 v_2 = C \tag{13.62}$$

where C is a constant evaluated from known conditions at section 1. Thus, in general,

$$pv = \frac{p}{\rho} = \frac{k-1}{2k}\left(C - \frac{\dot{m}^2}{\rho^2 A^2}\right)$$

or

$$p = \frac{k-1}{2k}\left(C\rho - \frac{\dot{m}^2}{A^2 \rho}\right) \tag{13.63}$$

Hence

$$dp = \frac{k-1}{2k}\left(C + \frac{\dot{m}^2}{A^2\rho^2}\right)d\rho$$

Multiplying both sides by ρ and integrating,

$$\int_1^2 \rho\, dp = \frac{k-1}{2k}\left[C\left(\frac{\rho_2^2 - \rho_1^2}{2}\right) + \frac{\dot{m}^2}{A^2}\ln\frac{\rho_2}{\rho_1}\right] \tag{13.64}$$

We can obtain the value of $\int_1^2 \rho\, dp$ from Eq. (13.64) since we can find ρ for any flow $\dot{m}$ and any value of p by Eq. (13.63). Practically, it will be better to assume values of ρ and find the corresponding values of p by Eq. (13.62).

Next let us rewrite Eq. (13.55). Multiplying each term by ρ^2 so that the first term becomes $\rho\, dp$, and substituting $\rho V = \dot{m}/A$ and $\rho\, dV = -V d\rho$ (from the differential of ρAV = constant), we obtain

$$\rho\, dp - \left(\frac{\dot{m}}{A}\right)^2 \frac{d\rho}{\rho} + \frac{f}{2D}\left(\frac{\dot{m}}{A}\right)^2 dx = 0$$

Integrating this (assuming f to be constant)[5] and rearranging with $x_2 - x_1 = L$,

$$f\frac{L}{D}\frac{1}{2}\left(\frac{\dot{m}}{A}\right)^2 = -\int_1^2 \rho\, dp + \left(\frac{\dot{m}}{A}\right)^2 \ln\frac{\rho_2}{\rho_1} \tag{13.65}$$

Since $\int_1^2 \rho\, dp$ may be evaluated by Eq. (13.64) it is possible to solve Eq. (13.65) to obtain a value of f if p_1 and p_2 have been measured for a known distance L, or the value of L may be found for any assumed values of p_1 and p_2 (or preferably ρ_1 and ρ_2) if f is given or assumed.

Thus it is possible through successive calculations to plot a curve such as that in Fig. 13.12 for any assumed flow and initial conditions where p_2 represents any pressure along the pipe at any distance x_2. However, as in the case of isothermal flow, there is a minimum value of p_2 where the velocity has attained its maximum value. We can prove that in adiabatic flow, critical conditions occur when $\mathbf{M} = 1.0$.

The equations in this section (Eqs. 13.61–13.65) apply equally well to supersonic flow. However, the key characteristics of the flow are then reversed, as we will explain.

Adiabatic flow with friction is called ***Fanno flow,*** which we can represent on an enthalpy-entropy (*h*-*s*) diagram as in Fig. 13.13. We discussed enthalpy h and entropy s in Sec. 13.1.

In the subsonic region, the ***Fanno line*** approaches asymptotically the stagnation enthalpy line as the Mach number decreases. Thus the extreme left end of the Fanno line is nearly horizontal.

The point on the Fanno line that corresponds to maximum entropy represents conditions of sonic velocity and Mach 1. If a gas entering a duct is flowing

[5] In adiabatic flow in a duct of constant area the Reynolds number is not constant along the pipe because of changes in viscosity caused by variations in temperature. Thus, since $\mathbf{R}$ varies, f must vary. For most situations, however, we can assume f to have a constant value without introducing much error.

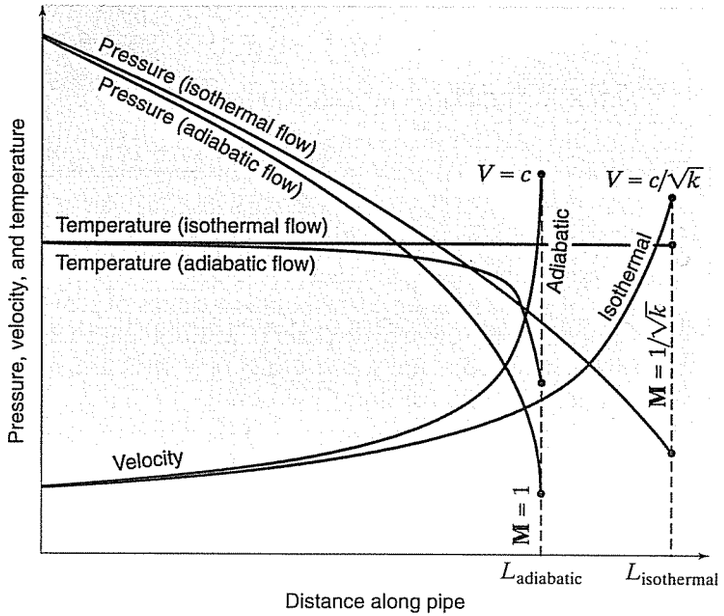


Figure 13.12
Subsonic flow of a compressible fluid in a constant-area duct.

at subsonic velocity, friction will have the effect of accelerating the flow so that sonic velocity is approached; likewise, if the flow at the entrance is supersonic, the gas will be decelerated, also approaching Mach 1. In each case, when Mach 1 is reached choking of the flow occurs.

All points on a Fanno line relate to the same stagnation temperature and to the same flow rate per unit area. If the cross-sectional area of a duct changes, or if the mass rate of flow changes in a duct, a different Fanno line then applies.

Flow of a fluid in the path described by a Fanno line always represents an entropy increase. In subsonic flow along a Fanno line there is a reduction in

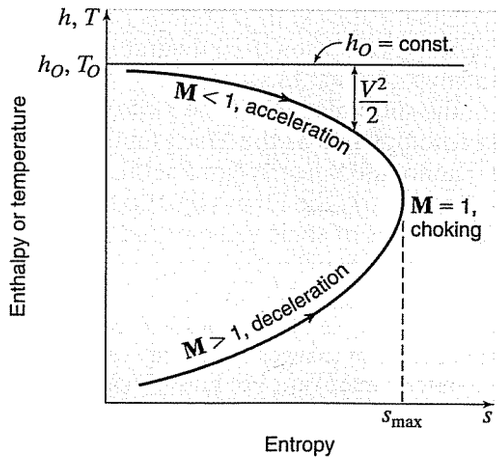


Figure 13.13
Fanno line on an *h-s* diagram.

TABLE 13.3 **Changes of flow properties in the downstream direction for adiabatic flow in a constant-area duct**

Flow	$d\mathbf{M}$	dh	dT	$d\rho$	dV	dp	dp_o	ds
Subsonic ($\mathbf{M} < 1$)	+	−	−	−	+	−	−	+
Supersonic ($\mathbf{M} > 1$)	−	+	+	+	−	+	−	+

\+ indicates increase
− indicates decrease

enthalpy, temperature, and density, and so an increase in velocity. Also, according to momentum principles, there must be a decrease in pressure. In supersonic flow along a Fanno line there is an increase in enthalpy, so that the velocity decreases, and therefore there is an increase in density and an increase in pressure. In both subsonic flow and supersonic flow, friction reduces the stagnation pressure. Table 13.3 summarizes the direction of changes in these properties. A fluid flowing at subsonic velocity tends to expand as a result of friction; on the other hand, a gas flowing supersonically tends to contract.

SAMPLE PROBLEM 13.9 Air flows adiabatically through a 10-in by 14-in rectangular duct at 3.0 slug/sec (Fig. S13.9). At a certain section the pressure is 80 psia and the temperature is 65°F (same data as Sample Prob. 13.7). Find the distance along the duct to the section where $\rho_2 = 0.80\rho_1$. Assume the duct surface is very smooth; so use the lowest curve of Fig. 8.11 or Eq. (8.47) to determine f.

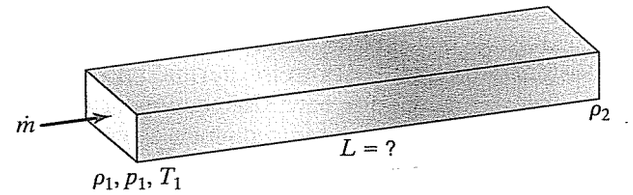


Figure S13.9

Solution

From Sample Prob. 13.7:

$$\rho_1 = 0.01279 \text{ slug/ft}^3, \quad V_1 = 241 \text{ ft/sec}, \quad \mathbf{R}_1 = 7.93 \times 10^6, \quad f = 0.0084$$

From Eq. (13.62):

$$C = \frac{3.0^2}{(0.01279 \times 140/144)^2} + \frac{2(1.4)}{0.4}\left(\frac{80 \times 144}{0.01279}\right) = 6{,}360{,}000 \text{ ft}^2/\text{sec}^2$$

Equation (13.63) yields

$$p_2 = \frac{0.4}{2.8}\left(6{,}360{,}000 \times 0.8 \times 0.01279 - \frac{3.0^2}{(140/144)^2(0.8 \times 0.01279)}\right)$$

$$= 9170 \text{ lb/ft}^2 \text{ abs} = 63.7 \text{ psia}$$

From Eq. (13.64), $\int_1^2 \rho\, dp = -27.1$ lb^2·sec^2/ft^6. Putting this value into Eq. (13.65) and substituting for f and (from Sec. 8.3) for $D = 4A/P = 0.972$ ft yields

$$L = 614 \text{ ft} \qquad \textit{ANS}$$

Thus at a section 614 ft downstream from the first section, $\rho_2 = 0.8\rho_1 = 0.01024$ slug/ft^3 and $p_2 = 63.7$ psia. Also,

$$V_2 = \frac{\dot{m}}{\rho_2 A} = \frac{3.0}{0.01024 \times 140/144} = 301 \text{ ft/sec}$$

$$T_2 = \frac{p_2}{\rho_2 R} = \frac{9170}{0.01024 \times 1715} = 522°\text{R} = 62°\text{F}$$

By assuming other values of ρ_2, we can get a complete picture of the flow at various sections along the pipe.

13.15 COMPARISON OF FLOW TYPES

The two types of compressible flow in pipelines that we have just discussed, isothermal and adiabatic flow, are special cases that are often approximated in practice and that are amenable to mathematical treatment. For fluids that do not follow the perfect-gas laws, such as wet steam, the preceding equations are only rough approximations. A more complicated thermodynamic treatment is necessary than is within the scope of this text.

In Fig. 13.12 we see curves plotted to scale for the flow of air through a constant-area duct for both isothermal and adiabatic conditions, assuming the same initial values for each. Inspection of this diagram shows that for small pressure drops from the initial, such as up to $\Delta p/p_1 = 0.10$ (or $p_2/p_1 = 0.90$), there is very little difference between the two curves. Thus, for such a situation where $p_2/p_1 > 0.90$, adiabatic flow in a pipe can be analyzed as isothermal flow without introducing much error. The flow of gas in a pipe is rarely either isothermal or adiabatic. Isothermal flow requires that heat be transferred into the flowing fluid from the surrounding atmosphere at just the right rate, and this rate must increase along the length of the pipe. If the rate of heat transfer is less than this required amount, the performance curves will lie between the isothermal and the adiabatic curves in Fig. 13.12. Heat transfer is proportional to the temperature difference between the fluid and the surrounding atmosphere. If we denote these temperatures as T_f and T_a, respectively, the heat transfer is some function of $T_a - T_f$. Isothermal flow is possible only if T_a is greater than T_f.

If the fluid in the pipe is very much colder than the surroundings, its absorption of heat might be such as to cause the pressure in the pipe to be higher than for isothermal flow. On the other hand, for example, air at a high temperature might be discharged directly from a compressor into a pipe so that T_f might

be greater than T_a, which would cause heat to flow from the fluid in the pipe to the surrounding atmosphere. In this case the pressure along the pipe would decrease even faster than for adiabatic flow.

Because the energy equation for compressible fluids does not contain a term for friction and the momentum equation which contains a term for friction[6] does not include any term for heat transfer, we see that there is no simple analytical solution of such cases. An approximate approach to such problems is to divide the entire length of pipe into short reaches and employ the equations of incompressible flow using average values of density and velocity within each reach. This step-by-step method will give results that are approximately correct if the lengths of the reaches are made small enough. Small reach lengths are particularly important in the regions where the curves of Fig. 13.12 are sharply curved.

13.16 CONCLUDING REMARKS

In this chapter we have seen that in order to solve problems of compressible flow, we must combine equations describing the flow process of the gas with the energy equation and the continuity principle. Hence thermodynamics is commonly involved in compressible-flow problems and the expressions relating the various physical parameters are generally quite complicated. In the discussion we have restricted ourselves to one-dimensional flow, and have not considered multidimensional flow, which is of course an extremely important topic, especially when dealing with aircraft and missiles. The intent here was merely to provide an introduction to the flow of compressible fluids. Among the references cited in Appendix E are some excellent treatments of compressible flow.

[6] Friction is accounted for as one of the forces acting on the fluid element in the equation $\Sigma F = \rho Q \Delta V$.

PROBLEMS

13.1 Using the data of Sample Prob. 13.2, determine the work done in compressing the air by finding the area under a pressure-vs-volume curve. Compute and tabulate volumes and pressures using volume increments that are 10% of the original volume.

13.2 Find the stagnation pressure and temperature in air flowing at 88 ft/sec if the static pressure and static temperature are 14.7 psia and 50°F, respectively.

13.3 Derive Eq. (13.35) for isentropic flow by integrating the one-dimensional Euler equation.

13.4 Carbon dioxide flows isentropically. At a point in the flow the velocity is 150 ft/sec, the temperature is 125°F, and the pressure is 20 psia. Determine the pressure and temperature on the nose of a streamlined object placed in the flow at that point.

13.5 Air flows at 150°F from a large tank through a 1.5-in-diameter converging nozzle (Fig. P13.5). Within the tank the pressure is 85 psia. Calculate the flow rate for back pressures of 10, 30, 50, and 70 psia. Assume isentropic conditions. Plot $\dot{m}$ as a function of p_b. Assume that the temperature within the tank is 150°F in all cases. Compute also the temperature at the nozzle outlet for each condition.

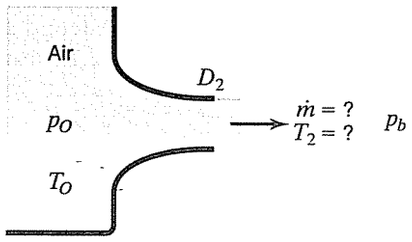


Figure P13.5

13.6 Air flows at 65°C from a large tank through a 40-mm-diameter converging nozzle (Fig. P 13.5). Within the tank the pressure is 600 kPa abs. Calculate the flow rate for back pressures of 50, 200, 350, and 500 kPa abs. Assume isentropic conditions. Plot $\dot{m}$ as a function of p_b. Assume that the temperature within the tank is 65°C in all cases. Compute also the temperature at the nozzle outlet for each condition.

13.7 Air flows at 25°C from a large tank through a 100-mm-diameter converging nozzle (Fig. P13.5). Within the tank the pressure is 50 kPa abs. Calculate the flow rate for pressures of 30, 20, and 10 kPa abs. Assume isentropic conditions. Plot $\dot{m}$ as a function of p_b. Assume that the temperature within the tank is 25°C in all cases. Compute also the temperature at the nozzle outlet for each condition.

13.8 Air within a tank at 120°F flows isentropically through a 2-in-diameter convergent nozzle into a 14.2-psia atmosphere. Find the mass flow rate for air pressures within the tank of 5, 10, 20, 40, and 50 psia.

13.9 Refer to Sample Prob. 13.4. If the pressure in the tank is 5 psig, confirm by calculation that $\dot{m} = 0.0374$ slug/sec, $p_2 = 13.50$ psia, and $T_2 = 494$°R.

13.10 Air discharges from a large tank through a converging-diverging nozzle (Fig. P13.10). The throat diameter is 3.0 in, and the exit diameter is 4.0 in. Within the tank the air pressure and temperature are 40 psia and 150°F, respectively. Calculate the mass flow rate for back pressures of 39, 38, 36, and 30 psia. Assume no friction.

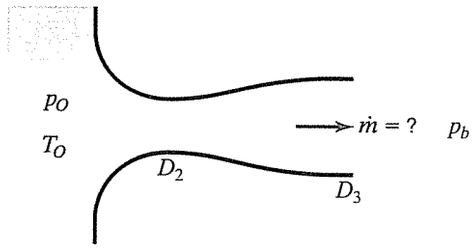


Figure P13.10

13.11 Air discharges from a large tank through a converging-diverging nozzle (Fig. P13.10). The throat diameter is 75 mm, and the exit diameter is 100 mm. Within the tank the air pressure and temperature are 290 kPa abs and 65°C, respectively. Calculate the flow rate for back pressures of 280, 270, 250, and 200 kPa abs. Assume no friction.

13.12 Repeat Exer. 13.9.5 for the case where the air within the tank is at 20 psia. Assume all other data to be the same.

13.13 Repeat Exer. 13.9.6 for the case where the air within the tank is at 140 kPa abs. Assume all other data to be the same.

13.14 Air discharges from a large tank through a converging-diverging nozzle with a 25-mm-diameter throat into the atmosphere. The gage pressure and temperature in the tank are 700 kPa and 40°C, respectively, the barometric pressure is 995 millibars. (*a*) Find the nozzle-tip diameter required for p_3 to be equal to the atmospheric pressure. For this case, what

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

are the flow velocity, sonic velocity, and Mach number at the nozzle exit? (*b*) Determine the value of p_b that will cause the shock wave to be located at the nozzle exit.

13.15 The pressure, velocity, and temperature just upstream of a normal shock wave in air are 10 psia, 2200 fps and 23°F. Determine the pressure, velocity, and temperature just downstream of the wave.

13.16 Air flows isothermally in a long pipe. At one section the pressure is 90 psia, the temperature is 80°F, and the velocity is 100 fps. At a second section some distance from the first the pressure is 15 psia. Find the energy head loss due to friction, and determine the thermal energy (ft·lb/lb) that must have been added to or taken from the fluid between the two sections. The diameter of the pipe is constant.

13.17 Air flows isothermally in a long pipe. At one section the pressure is 600 kPa abs, the temperature is 25°C, and the velocity is 30 m/s. At a second section some distance from the first the pressure is 100 kPa abs. Find the energy head loss due to friction, and determine the thermal energy (N·m/N) that must have been added to or taken from the fluid between the two sections. The diameter of the pipe is constant.

13.18 Refer to Sample Prob. 13.7. Neglecting the logarithm term in Eq. (13.59), find the pressures and velocities at sections 100, 300, and 800 ft downstream of the section where the pressure is 80 psia. Plot the pressure and velocity as a function of distance along the duct.

13.19 Carbon dioxide flows isothermally at 100°F through a horizontal 6-in-diameter pipe (Fig. P13.19). At this temperature $\mu = 4.0 \times 10^{-7}$ lb·sec/ft^2. The pressure changes from 150.0 to 140.0 psig in a 100-ft length of pipe. Determine the mass flow rate if the atmospheric pressure is 14.5 psia and e for the pipe is 0.002 ft.

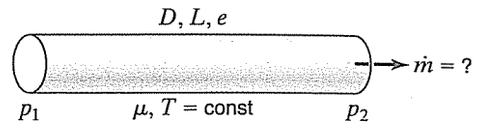


Figure P13.19

13.20 Carbon dioxide flows isothermally at 40°C through a horizontal 150-mm-diameter pipe (Fig. P13.19). At this temperature $\mu = 1.95 \times 10^{-5}$ N·s/m^2. The pressure changes from 1000 to 930 kPa gage in a 30-m length of pipe. Determine the mass flow rate if the atmospheric pressure is 100 kPa abs and e for the pipe is 0.60 mm.

13.21 Methane gas is to be pumped through a 24-in-diameter welded-steel pipe connecting two compressor stations 25 miles apart. At the upstream station the pressure is not to exceed 60 psia, and at the downstream station it is to be at least 20 psia. Determine the maximum possible rate of flow (in cubic feet per day at 60°F and 1 atm). Assume isothermal flow at 60°F.

13.22 Refer to Sample Prob. 13.9. Find the distance along the duct to (*a*) where $\rho_2 = 0.9\rho_1$; (*b*) where $\rho_2 = 0.7\rho_1$; (*c*) where subsonic adiabatic flow ends. Compute the corresponding values of p, V, T, and **M** and plot the first three as a function of distance along the duct.

13.23 Air flows adiabatically at 3.0 slug/sec in a 12-in-diameter horizontal pipe. At a certain section the pressure is 150 psia and the temperature is 140°F. Determine the distance along the pipe to the section where $\rho_2 = 0.80\rho_1$. Assume $e/D = 0.0004$.

13.24 Air flows adiabatically at 50 kg/s in a 300-mm-diameter horizontal pipe. At a certain section the pressure is 1000 kPa abs and the temperature 60°C. Determine the distance along the pipe to the section where $\rho_2 = 0.80\rho_1$. Assume $e/D = 0.0004$.

: Programmed computing aids (Appendix C) could help solve problems marked with this icon.

CHAPTER 14
Ideal Flow Mathematics

In this chapter we discuss various mathematical methods for describing the flow of imaginary ideal (frictionless) fluids. This subject is often referred to as ***hydrodynamics.*** It is a vast subject, so that the presentation here provides only an introduction, but it does give a good idea of the possibilities of a rigorous mathematical approach to flow problems.

Even though such an approach does not consider all the real properties of fluids, the results often closely approximate the behavior of real fluids. This is because there are numerous situations in which friction plays only a minor role. For example, in Chaps. 8 and 9, we noted, for fluids of low viscosity, that the viscosity affects only a thin region at the fluid boundaries. Also, in Chaps. 8–10 we often saw that turbulence and separation of the boundary layer occur far more readily with decelerating flows, and that accelerating flows generally have thin boundary layers. For such flows, mathematical analysis of ideal fluids yields results, often elegant, that can and do provide many useful and important insights into real fluid behavior.

To concentrate on fundamentals, after Sec. 14.1 we shall limit our discussions to constant-density fluids and to two-dimensional, steady flow fields. In Sec. 14.8 we shall see, rather interestingly, how the same methods can be applied to the flow of a *real* fluid through porous media.

14.1 DIFFERENTIAL EQUATION OF CONTINUITY

In Chap. 4 we presented a very practical, but special, form of the equation of continuity. For some purposes a more general three-dimensional form is desired. Also, in that chapter the concept of the flow net was explained largely on an intuitive basis. To reach a more fundamental understanding of the mechanics of the flow net, it is necessary to consider the differential equations of continuity and irrotationality (Sec. 14.2) that give rise to the orthogonal network of streamlines and equipotential lines.

Aside from application to the flow net, the differential form of the continuity equation has an important advantage over the one-dimensional form that we derived in Sec. 4.7 in that it is perfectly general for two- or three-dimensional

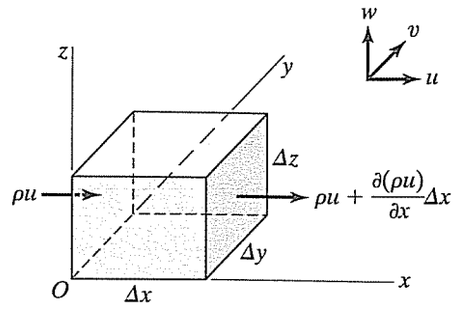


Figure 14.1

fluid space and for either steady or unsteady flow. Some of the equations in this section only will also be applicable to compressible flow.

Figure 14.1 shows three coordinate axes x, y, and z mutually perpendicular and fixed in space. Let the velocity components in these three directions be u, v, and w, respectively. Consider now a small parallelepiped, having sides Δx, Δy, and Δz. In the x direction the rate of mass flow into this box through the left face is approximately $\rho u\,\Delta y\,\Delta z$, this expression becoming exact in the limit as the box is shrunk to a point. The corresponding rate of mass flow out of the box through the right face is $\{\rho u + [\partial(\rho u)/\partial x]\Delta x\}\Delta y\,\Delta z$. Thus the net rate of mass flow into the box in the x direction is $-[\partial(\rho u)/\partial x]\,\Delta x\,\Delta y\,\Delta z$. Similar expressions may be obtained for the y and z directions. The sum of the rates of mass inflow in the three directions must equal the time rate of change of the mass in the box, or $(\partial\rho/\partial t)$ $\Delta x\,\Delta y\,\Delta z$. Summing up, applying the limiting process, and dividing both sides of the equation by the volume of the parallelepiped, which is common to all terms, we get

$$\text{Unsteady compressible flow:} \quad -\frac{\partial(\rho u)}{\partial x} - \frac{\partial(\rho v)}{\partial y} - \frac{\partial(\rho w)}{\partial z} = \frac{\partial \rho}{\partial t} \tag{14.1}$$

which is the equation of continuity in its most general form. This equation as well as the other equations in this section are, of course, valid regardless of whether the fluid is a real one or an ideal one. If the flow is steady, ρ does not vary with time, but it may vary in space. Since $\partial(\rho u)/\partial x = \rho(\partial u/\partial x) + u(\partial\rho/\partial x)$, it follows that for steady flow we can write the equation as

$$\text{Steady compressible flow:} \quad u\frac{\partial \rho}{\partial x} + v\frac{\partial \rho}{\partial y} + w\frac{\partial \rho}{\partial z} + \rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = 0 \tag{14.2}$$

In the case of an incompressible fluid (ρ = constant), whether the flow is steady or not, the equation of continuity becomes

$$\text{Continuity, for steady incompressible flow:} \quad \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{14.3}$$

For two-dimensional flow, application of the same procedure to an elemental volume in polar coordinates yields for steady flow these equations:

Steady compressible flow:
$$\frac{1}{r}(\rho v_r) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\partial}{r\partial\theta}(\rho v_t) = 0 \qquad (14.4)$$

Steady incompressible flow:
$$\frac{v_r}{r} + \frac{\partial v_r}{\partial r} + \frac{\partial v_t}{r\partial\theta} = 0 \qquad (14.5)$$

where v_r and v_t represent the velocities in the radial and tangential[1] directions, respectively.

SAMPLE PROBLEM 14.1 Assuming ρ to be constant, do the following flows satisfy continuity? (*a*) $u = -2y$, $v = 3x$; (*b*) $u = 0$, $v = 3xy$; (*c*) $u = 2x$, $v = -2y$.

Solution
From Eq. (14.3): Continuity for incompressible fluids is satisfied if $\partial u/\partial x + \partial v/\partial y = 0$.

(*a*) $\dfrac{\partial(-2y)}{\partial x} + \dfrac{\partial(3x)}{\partial y} = 0 + 0 = 0$ Continuity is satisfied ***ANS***

(*b*) $\dfrac{\partial(0)}{\partial x} + \dfrac{\partial(3xy)}{\partial y} = 0 + 3x \neq 0$ Continuity is not satisfied ***ANS***

(*c*) $\dfrac{\partial(2x)}{\partial x} + \dfrac{\partial(-2y)}{\partial y} = 2 - 2 = 0$ Continuity is satisfied ***ANS***

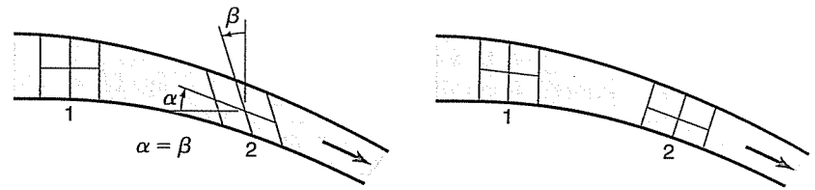
Note: If (*b*) did indeed describe a flow field, the fluid must be compressible.

EXERCISES

14.1.1 Which of the following incompressible flows satisfy continuity?
(*a*) $u = 2$
(*b*) $u = 2, \quad v = 3$
(*c*) $u = 2 + 3x, \quad v = 4$
(*d*) $u = 2y, \quad v = 3x$
(*e*) $u = 2y, \quad v = -3x$
(*f*) $u = 3xy, \quad v = 1.5x^2$

14.1.2 Which of the following incompressible flows satisfy continuity?
(*a*) $u = 3y, \quad v = 0$
(*b*) $u = 3x, \quad v = 3y$
(*c*) $u = 3x, \quad v = -3y$
(*d*) $u = 4 + 2x, \quad v = -6 - 2y$
(*e*) $u = 4xy + x^2, \quad v = -2xy - 2y^2$
(*f*) $u = -2xy + 2x^2, \quad v = 4xy - y^2$
(*g*) $u = -2xy - 2x^2 + 2y^2, \quad v = 4xy - x^2 + y^2$

[1] In this chapter we use v_t to represent the tangential component of velocity.



(*a*) Irrotational flow (*b*) Rotational flow

Figure 14.2
Two-dimensional flow along a curved path.

14.2 IRROTATIONAL FLOW

In the remainder of this chapter we will restrict our discussion to incompressible fluids. Briefly, we can describe ***irrotational flow*** as flow in which each element of the moving fluid suffers no *net* rotation from one instant to the next, with respect to a given frame of reference. The classic example of irrotational motion (although not a fluid) is that of the carriages on a Ferris wheel used for amusement rides. Each carriage describes a circular path as the wheel revolves, but does not rotate with respect to the earth. In irrotational flow, however, a fluid element may deform providing the axes of the element rotate equally toward or away from each other ($\alpha = \beta$ in Fig. 14.2*a*), so that there is no *net* rotation. As long as the algebraic average rotation (or angular velocity) is zero, the motion is irrotational.

In Fig. 14.2*b* we see an example of rotational flow. In this case both axes have rotated in a clockwise sense, so there is a net rotation of the fluid element. Actually, the deformation of the element in Fig. 14.2*b* is less than that of Fig. 14.2*a*.

Let us now express the condition of irrotationality in mathematical terms. It will help to restrict the discussion at first to two-dimensional motion in the xy plane. Consider a small fluid element moving as depicted in Fig. 14.3*a*. During a

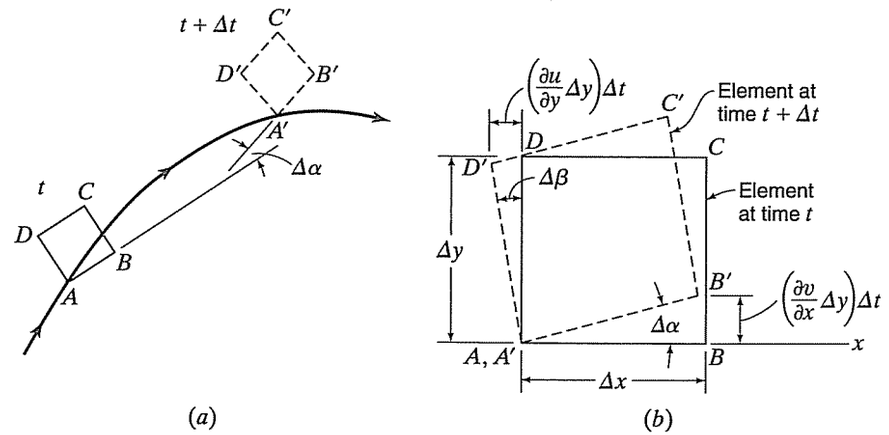


Figure 14.3

short time interval Δt, the element moves from one position to another and in the process it deforms as indicated. Superimposing A' on A, defining an x axis along AB, and enlarging the diagram, we get Fig. 14.3*b*. The angle $\Delta\alpha$ between AB and $A'B'$ can be expressed from Fig. 14.3*b* as

$$\Delta\alpha = \frac{BB'}{\Delta x} = \frac{[(\partial v/\partial x)\Delta x]\Delta t}{\Delta x} = \frac{\partial v}{\partial x}\Delta t$$

Hence the rate of rotation of the edge of the element that was originally aligned with AB is

$$\omega_\alpha = \frac{\Delta\alpha}{\Delta t} = \frac{\partial v}{\partial x}$$

Likewise $$\Delta\beta = \frac{DD'}{\Delta y} = \frac{[-(\partial u/\partial y)\Delta y]\Delta t}{\Delta y} = -\frac{\partial u}{\partial y}\Delta t$$

and the rate of rotation of the edge of the element that was originally aligned with AD is

$$\omega_\beta = \frac{\Delta\beta}{\Delta t} = -\frac{\partial u}{\partial y}$$

with the negative sign because $+u$ is directed to the right. The rate of rotation of the element about the z axis is now defined to be ω_z, the average of ω_α and ω_β; thus

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \tag{14.6}$$

But the criterion we originally stipulated for irrotational flow was that the rate of rotation be zero. Therefore we have

Irrotational flow in xy plane: $$\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \tag{14.7}$$

In three-dimensional flow there are corresponding expressions for the components of angular-deformation rates about the x and y axes. Finally, for the general case, irrotational flow is defined to be that for which

$$\omega_x = \omega_y = \omega_z = 0 \tag{14.8}$$

The rotation of incompressible fluid particles requires the application of torque, which in turn depends on shearing forces. Such forces are possible only in a viscous fluid. In ideal (inviscid or frictionless) fluids (Sec. 2.10) there can be no shears or torques. So it follows that all ideal incompressible flows are irrotational. However, some real flows can be irrotational too, such as in undisturbed regions.

In Sec. 14.6 we shall see that the primary significance of irrotational flow is that it corresponds to a velocity potential. Also, note that for irrotational flow, the Bernoulli equation (5.7) and (5.30) holds *throughout* the flow field (not just along a streamline).

SAMPLE PROBLEM 14.2 Determine whether the following flows are rotational or irrotational: (*a*) $u = -2y$, $v = 3x$; (*b*) $u = 0$, $v = 3xy$; (*c*) $u = 2x$, $v = -2y$.

Solution
Using Eq. (14.7):

(*a*) $\dfrac{\partial(3x)}{\partial x} - \dfrac{\partial(-2y)}{\partial y} = 3 + 2 \neq 0$ Flow is rotational ***ANS***

(*b*) $\dfrac{\partial(3xy)}{\partial x} - \dfrac{\partial(0)}{\partial y} = 3y - 0 \neq 0$ Flow is rotational ***ANS***

(*c*) $\dfrac{\partial(-2y)}{\partial x} - \dfrac{\partial(2x)}{\partial y} = 0 - 0 = 0$ Flow is irrotational ***ANS***

EXERCISES

14.2.1 Which of the flows of Exer. 14.1.1 are irrotational?

14.2.2 Which of the flows of Exer. 14.1.2 are irrotational?

14.3 CIRCULATION AND VORTICITY

To get a better understanding of the character of a flow field, we should acquaint ourselves with the concept of ***circulation.***[2] Let the streamlines of Fig. 14.4 represent a two-dimensional ***flow field,*** while L represents any closed path in this field. The circulation Γ (gamma) is defined mathematically as a ***line integral*** of

[2] In Secs. 9.10 and 9.11 we used the concept of circulation to develop an expression for lift force on an air foil.

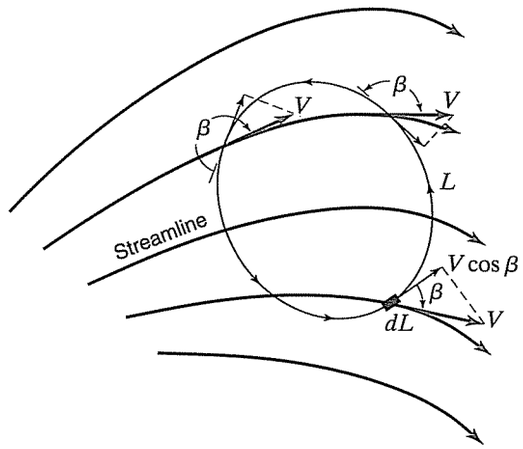


Figure 14.4 Circulation around a closed path in a two-dimensional field.

the velocity around a closed path. Thus

$$\Gamma = \oint_L \mathbf{V}\cdot d\mathbf{L} = \oint_L V\cos\beta\, dL \tag{14.9}$$

where $\mathbf{V}$ is the velocity in the flow field at the element $d\mathbf{L}$ of the path, and β is the angle between $\mathbf{V}$ and the tangent to the path (in the positive direction along the path) at that point. Equation (14.9) is analogous to the common equation in mechanics for work done as a body moves along a curved path while the force makes some angle with the path. The only difference here is the substitution of a velocity for a force.

Evaluation of Eq. (14.9) about any closed curve generally involves a tedious step-by-step integration. We acquire some valuable information, however, by evaluating the circulation of the two-dimensional flow field of Fig. 14.5 by taking the line integral around the boundary of the indicated element. Since the element is of differential size, the resulting circulation is also differential. Thus, starting at A and proceeding counterclockwise,

$$d\Gamma = \frac{u_A + u_B}{2}dx + \frac{v_B + v_C}{2}dy - \frac{u_C + u_D}{2}dx - \frac{v_D + v_A}{2}dy \tag{14.10}$$

where the values of u_A, u_B, u_C, and u_D and v_A, v_B, v_C, and v_D are as indicated in Fig. 14.5. Substituting these values into Eq. (14.10), expanding, combining terms, and disregarding those of higher order yields

$$d\Gamma = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)dx\,dy \tag{14.11}$$

The ***vorticity*** ξ (xi) is defined as the circulation per unit of enclosed area. Thus

$$\xi = \frac{d\Gamma}{dx\,dy} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \tag{14.12}$$

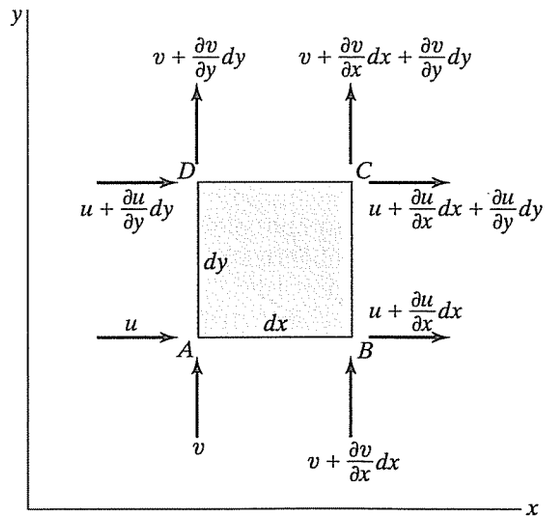


Figure 14.5

Comparing Eq. (14.12) with Eq. (14.7), we see that an irrotational flow is one for which the vorticity $\xi = 0$. Similarly, if the flow is rotational, $\xi \neq 0$.

Using a similar procedure for polar coordinates, we find

$$\xi = \frac{\partial v_t}{\partial r} + \frac{v_t}{r} - \frac{1}{r}\frac{\partial v_r}{\partial \theta} \tag{14.13}$$

SAMPLE PROBLEM 14.3 Check the following incompressible flows for continuity and determine the vorticity of each: (*a*) $v_t = 6r$, $v_r = 0$; (*b*) $v_t = 0$, $v_r = -5/r$

Solution

Applying Eqs. (14.5) and (14.13):

(*a*) $\dfrac{0}{r} + \dfrac{\partial(0)}{\partial r} + \dfrac{1}{r}\dfrac{\partial(6r)}{\partial \theta} = 0$ Continuity is satisfied ***ANS***

$$\xi = \frac{\partial(6r)}{\partial r} + \frac{6r}{r} - \frac{1}{r}\frac{\partial(0)}{\partial \theta} = 6 + 6 - 0 = 12 \quad \textit{ANS}$$

(Flow is rotational)

(*b*) $$-\frac{5/r}{r} + \frac{\partial(-5r^{-1})}{\partial r} + \frac{1}{r}\frac{\partial(0)}{\partial \theta} = -\frac{5}{r^2} + \frac{5}{r^2} + 0 = 0$$

Continuity is satisfied ***ANS***

$$\xi = \frac{\partial(0)}{\partial r} + \frac{0}{r} - \frac{1}{r}\frac{\partial(-5/r)}{\partial \theta} = 0 \quad \textit{ANS}$$

(Flow is irrotational)

EXERCISES

14.3.1 Find the vorticity of each of the flows in Exer. 14.1.1.

14.3.2 Find the vorticity of each of the flows in Exer. 14.1.2.

14.4 THE STREAM FUNCTION

The ***stream function*** ψ (psi), based on the continuity principle, is a mathematical expression that describes a flow field. In Fig. 14.6 are two adjacent ***streamlines*** (Sec. 4.4) of a two-dimensional flow field. Let $\psi(x, y)$ represent the streamline nearest the origin. Then $\psi + d\psi$ is representative of the second streamline. Since there is no flow across a streamline, we can let ψ be indicative of the flow carried through the area from the origin O to the first streamline. And thus $d\psi$ represents the flow carried between the two streamlines of Fig. 14.6. From continuity, referring to the triangular fluid element of Fig. 14.6, we see that for an incompressible fluid

$$d\psi = -v\,dx + u\,dy \tag{14.14}$$

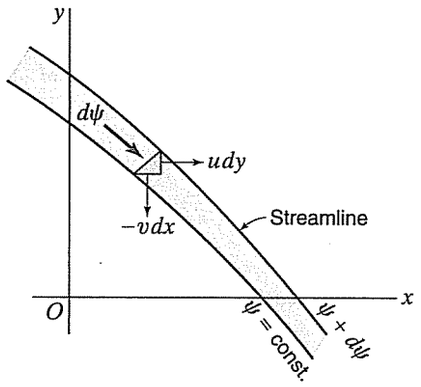


Figure 14.6
Stream function.

The total derivative $d\psi$ can also be expressed as

$$d\psi = \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy \tag{14.15}$$

Comparing these last two equations, we note that

$$u = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = -\frac{\partial \psi}{\partial x} \tag{14.16}$$

Thus, if we can express ψ as a function of x and y, we can find the velocity components (u and v) at any point of a two-dimensional flow field by application of Eqs. (14.16). Conversely, if we express u and v as functions of x and y, we can find ψ by integrating Eq. (14.14). Note however, that since the derivation of ψ is based on the principle of continuity, *continuity must be satisfied for the stream function to exist.* Also, since we did not consider vorticity in the derivation of ψ, *the flow need not be irrotational for the stream function to exist.*

The equation of continuity

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

can be expressed in terms of ψ by substituting the expressions for u and v from Eqs. (14.16); doing so, we get

$$\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial y}\right) - \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial x}\right) = 0, \quad \text{or} \quad \frac{\partial^2 \psi}{\partial x\, \partial y} = \frac{\partial^2 \psi}{\partial y\, \partial x}$$

which shows that, if $\psi = \psi(x, y)$, the derivatives taken in either order give the same result and that a flow described by a stream function automatically satisfies the continuity equation.

Note that the concept of the stream function ψ is limited to two-dimensional flow fields.

EXERCISES

14.4.1 The flow of an incompressible fluid is defined by $u = 2$, $v = 8x$. Does a stream function exist for this flow? If so, determine the expression for the stream function.

14.4.2 For each of the flows of Exer. 14.1.1, write an expression for the stream function if one exists.

14.4.3 For each of the flows of Exer. 14.1.2, write an expression for the stream function if one exists.

14.5 BASIC FLOW FIELDS

In this section we shall discuss several basic flow fields that are commonly encountered. Though these flow fields imply an ideal fluid, they closely depict the flow of a real fluid outside the zone of viscous influence provided there is no separation of the flow from the boundaries (see Sec. 4.10). The simplest of all flows is that in which the streamlines are straight, parallel, and evenly spaced (rectilinear) as in Fig. 14.7. In this case $v = 0$ and $u =$ constant. Thus, from Eq. (14.14), $d\psi = u\,dy$, and hence $\psi = Uy$, where U is the velocity of flow. If the distance between streamlines is a, the values of ψ for the streamlines are as indicated in Fig. 14.7.

Another flow field of general interest is that of a ***source*** or a ***sink.*** In the case of a source, the flow field consists of radial streamlines symmetrically spaced as shown in Fig. 14.8. If q is the ***source strength,*** or rate of flow from the

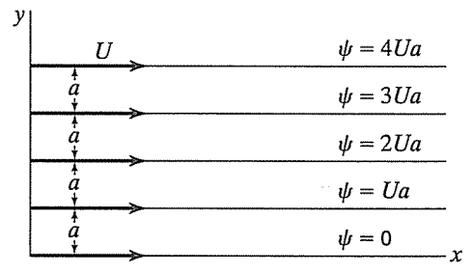


Figure 14.7
Rectilinear flow field.

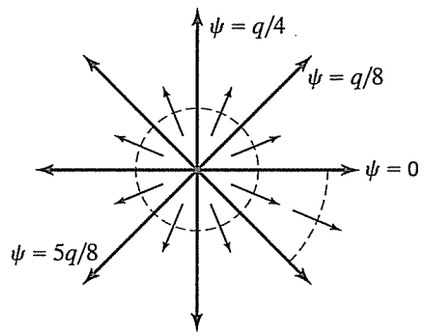


Figure 14.8
Source flow field.

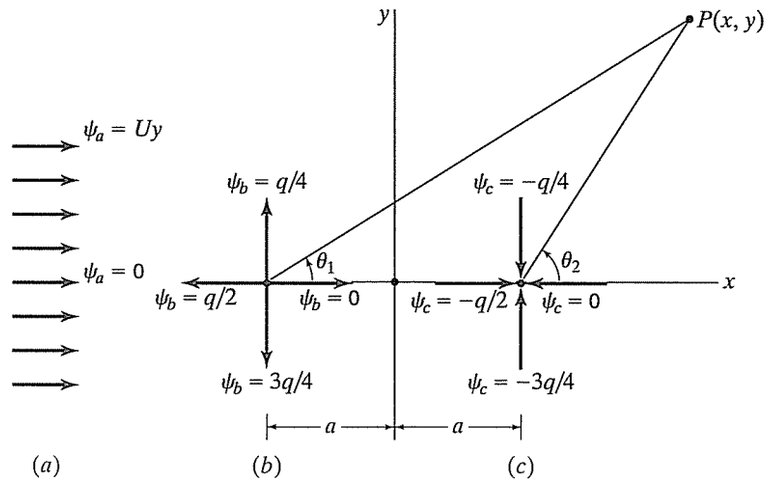


Figure 14.9
Superposition of flow fields. (*a*) Uniform rectilinear flow. (*b*) Source. (*c*) Sink. The source and sink are of equal strengths and a distance $2a$ apart along the x axis.

source, it is at once apparent that $\psi = q\theta/2\pi$. Customarily, for this case, we define the $\psi = 0$ streamline as that coincident with the direction of the x axis. From inspection of the flow field it is obvious that $v_t = 0$ and $v_r = q/2\pi r$. Thus $v_r \to 0$ as $r \to \infty$. For a sink (inward flow), the stream function is expressible as $\psi = -q\theta/2\pi$.

Flow fields can be combined by superposition to give other fields of importance. For example, let us combine a source and sink of equal strength with a rectilinear flow. Let $2a$ be the distance between the source and sink. Referring to Fig. 14.9 and defining θ_1 and θ_2 as shown, we can write for the combined field

$$\psi = Uy + \frac{q\theta_1}{2\pi} - \frac{q\theta_2}{2\pi} \tag{14.17}$$

Transforming the last two terms of this equation to cartesian coordinates by replacing the θ's with appropriate trigonometric functions, we get

$$\psi = Uy + \frac{q}{2\pi}\left(\arctan\frac{y}{x+a} - \arctan\frac{y}{x-a}\right) \tag{14.18}$$

This equation will permit us to plot streamlines by determining values of ψ at various points in the flow field having coordinates (x, y). Lines of constant ψ are streamlines. The resulting flow field for this case is shown in Sample Prob. 14.4. The $\psi = 0$ line produces a closed curve (oval), and thus the flow field represents ideal flow past a body of that shape. By using different values of a and different relationships between U and q, it is possible to describe a whole array of two-dimensional flow fields about ovals of various shapes.

As $2a$, the distance between the source and sink gets smaller, the oval approaches a circle. However, when $a = 0$, the flow field reduces mathematically to uniform rectilinear flow since the source and sink will cancel each other out. We can find the location of stagnation points S by differentiating Eq. (14.18) to obtain an expression for $u = \partial\psi/\partial y$ and then to determine the values of x for which $u = 0$.

The flow field of Sample Prob. 14.4 is for an ideal fluid and, of course, does not represent the flow picture for a real fluid, where there may be separation[3] with the formation of a wake on the downstream side of the body (Fig. 9.12). However, on the upstream side of the body where the boundary layer is thin, the flow of a real fluid is well represented by this example.

SAMPLE PROBLEM 14.4 A flow field for a source and sink of equal strength is combined with a uniform rectilinear flow. Given $U = 0.80$, $q = 2\pi$, and $a = 2$, plot the flow field.

Solution

Eq. (14.18): $$\psi = 0.80y + \arctan\frac{y}{x+2} - \arctan\frac{y}{x-2}$$

Letting $$A = \frac{y}{x+2} \quad \text{and} \quad B = \frac{y}{x-2}$$

we compute ψ for one of the symmetric quadrants:

				Degrees			Radians		
x	y	$\frac{y}{x+2} = A$	$\frac{y}{x-2} = B$	arctan A	arctan B	$0.8y$	arctan A	arctan B	ψ
0	2	$\frac{2}{2}$	$-\frac{2}{2}$	45°00′	135°00′	1.60	0.78	2.36	0.00
0	3	$\frac{3}{2}$	$-\frac{3}{2}$	56°19′	123°41′	2.40	0.98	2.16	1.22
0	4	$\frac{4}{2}$	$-\frac{4}{2}$	63°26′	116°34′	3.20	1.11	2.04	2.27
2	2	$\frac{2}{4}$	∞	26°34′	90°00′	1.60	0.46	1.57	0.49
2	3	$\frac{3}{4}$	∞	36°54′	90°00′	2.40	0.64	1.57	1.47
5	1	$\frac{1}{7}$	$\frac{1}{3}$	8°08′	18°26′	0.80	0.14	0.32	0.62
5	2	$\frac{2}{7}$	$\frac{2}{3}$	15°55′	33°42′	1.60	0.28	0.59	1.29
8	1	$\frac{1}{10}$	$\frac{1}{6}$	5°43′	9°28′	0.80	0.10	0.17	0.73
8	2	$\frac{2}{10}$	$\frac{2}{6}$	11°19′	18°26′	1.60	0.19	0.32	1.47

Plotting curves of constant ψ, we obtain Fig. S14.4 ***ANS***

[3] Refer to Sec. 9.6 for a discussion of the conditions under which separation will take place on the back side of a solid body.

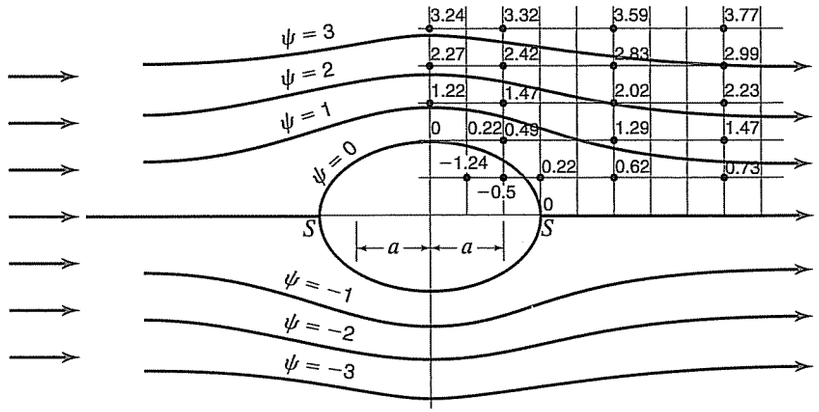


Figure S14.4
Flow field for source and sink of equal strength in a uniform rectilinear flow field. Note that point S represents a stagnation point.

Suppose we define a ***doublet*** as a source-sink combination for which $2qa = m$, a constant. Permitting a to approach zero, we see from the geometry of Fig. 14.9 that $r(\theta_2 - \theta_1) = 2a\sin\theta$. So $q(\theta_1 - \theta_2) = -(2aq\sin\theta)/r = -(m\sin\theta)/r$, which when substituted into Eq. (14.17) gives the stream function of the doublet imposed on the uniform field as

$$\psi = Uy - \frac{m\sin\theta}{2\pi r} \tag{14.19}$$

Taking $\psi = 0$ to determine the form of the closed-body contour and noting that $y = r\sin\theta$, we get

$$0 = Ur\sin\theta - \frac{m\sin\theta}{2\pi r}$$

or

$$r_{\psi=0} = R = \sqrt{\frac{m}{U2\pi}} = \text{constant}$$

Therefore the closed-body contour for this case is a circle with radius R, and Eq. (14.19) is the stream function for two-dimensional flow about a circular cylinder. We also discussed this flow in Sec. 9.10, with lift and circulation. Other basic flow fields of interest include the forced vortex ($\psi = \omega r^2/r$), the free vortex ($\psi = \Gamma\ln r/2\pi$), and various combinations such as the source in rectilinear flow (see Prob. 14.14), the source and sink of equal strength, and the doublet in rectilinear flow with circulation (see Fig. 9.18).

EXERCISES

14.5.1 A source discharging 25 m^3/s per m is located at the origin and a uniform flow with a velocity of 5 m/s from left to right is superimposed on the source flow. Determine the stream function of the flow (a) in polar and (b) in rectangular coordinates.

14.5.2 For the flow of Exer. 14.5.1, (a) find the location of the stagnation points and (b) find the velocity at $x = 3$ m, $y = 4$ m.

14.5.3 Refer to Exer. 14.5.1. Use Bernoulli's theorem to find the difference in pressure head between point A $(-14$ m, $0)$ and point B $(0, 2$ m$)$.

14.5.4 A source discharging 13 cfs/ft is at $(-1, 0)$ and a sink taking in 13 cfs/ft is at $(+1, 0)$. If a uniform flow with velocity 8 fps from left to right is superimposed on the source–sink combination, what is the length of the resulting closed body contour?

14.6 VELOCITY POTENTIAL

Let us define the potential

$$-d\phi = u\,dx + v\,dy \tag{14.20}$$

Mathematically, this is termed an "exact" differential, and therefore the function $\phi(x, y)$ exists, if

$$\frac{\partial u}{\partial y} = \frac{\partial v}{\partial x} \tag{14.21}$$

But the total derivative is defined to be

$$d\phi = \frac{\partial\phi}{\partial x}dx + \frac{\partial\phi}{\partial y}dy \tag{14.22}$$

By comparing (14.20) with (14.22) we see that in cartesian coordinates

$$u = -\frac{\partial\phi}{\partial x} \quad \text{and} \quad v = -\frac{\partial\phi}{\partial y} \tag{14.23}$$

For two-dimensional flow, we term ϕ (phi) with conditions (14.23) the ***velocity potential*** function. In polar coordinates, the corresponding expressions are

$$v_r = -\frac{\partial\phi}{\partial r} \quad \text{and} \quad v_t = -\frac{1}{r}\frac{\partial\phi}{\partial\theta} \tag{14.24}$$

When plotted graphically, lines of constant ϕ are called ***equipotential lines.***

The use of a minus sign in Eq. (14.20) led to the minus signs in the expressions (14.23), which indicate that the velocity potential decreases in the direction of flow, i.e., flow moves from areas of high potential to low potential. Some authors prefer the opposite, and so change these signs.

Differentiating Eqs. (14.23), we get

$$\frac{\partial u}{\partial y} = -\frac{\partial^2 \phi}{\partial y\,\partial x} \quad \text{and} \quad \frac{\partial v}{\partial x} = -\frac{\partial^2 \phi}{\partial x\,\partial y}$$

Since the right sides of these two last quantities are equal, this satisfies the requirement (14.21), which, from Eq. (14.12), proves that $\xi = 0$. Thus it follows that *if a flow is irrotational* (vorticity $\xi = 0$) *then a velocity potential exists, and vice versa.* Because of the existence of a velocity potential, irrotational and ideal flow is often called ***potential flow.***

If we substitute Eqs. (14.23) into the continuity Eq. (14.3), we get

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0 \tag{14.25}$$

This is the Laplace equation, named after the French mathematician and astronomer, Marquis Pierre Simon de Laplace (1749–1827). It is possibly the best known of all partial differential equations, important also in solid mechanics and thermodynamics. Because it is *linear* in ϕ (see Appendix B), using this equation gives us the important advantage that we can combine simple solutions to obtain solutions to more complex problems. For fluids, if a function ϕ satisfies Laplace's equation, the resulting flow must be irrotational.

The concept of the velocity potential function ϕ can be fully extended to three-dimensional flow fields.

EXERCISES

14.6.1 If $\phi = y + 2x^2$ is the velocity potential function for a two-dimensional flow, is it irrotational? Does it satisfy the Laplace equation? If not, suggest why.

14.6.2 Given the stream function $\psi = 10x - 7y$, is this a potential flow? If it is, determine the velocity potential function. Does it satisfy the Laplace equation?

14.6.3 For these stream functions, determine if a potential function exists, find the potential function if it does, and determine if the Laplace equation in ϕ is satisfied: (*a*) $\psi = 3xy + 2x$; (*b*) $\psi = 3xy + 2x^2$.

14.6.4 For these stream functions, determine if a potential function exists, find the potential function if it does, and determine if the Laplace equation in ϕ is satisfied: (*a*) $\psi = 6xy$; (*b*) $\psi = x \sin y$.

14.7 ORTHOGONALITY OF STREAMLINES AND EQUIPOTENTIAL LINES

From Eqs. (14.14) and (14.20), we have

$$d\psi = -v\,dx + u\,dy$$

and

$$d\phi = -u\,dx - v\,dy$$

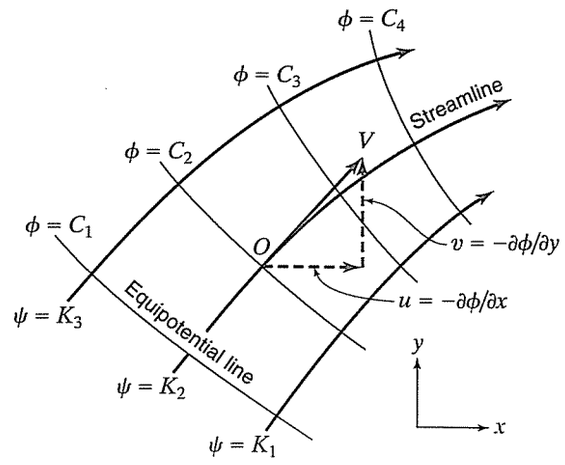


Figure 14.10
Flow net.

Along a streamline, ψ = constant, so $d\psi = 0$, and from the first equation we get $dy/dx = v/u$. Along an equipotential line, ϕ = constant, so $d\phi = 0$, and from the second equation we get $dy/dx = -u/v$. Geometrically, this tells us that the streamlines and equipotential lines are ***orthogonal***, or everywhere perpendicular to each other (the product of their slopes is -1). As a result, the stream function and the velocity potential are known as ***conjugate functions.***[4]

The equipotential lines $\phi = C_i$ and the streamlines $\psi = K_i$, where the C_i and the K_i have equal increments between adjacent lines, form a network of intersecting perpendicular lines that is called a ***flow net*** (Fig. 14.10; see also Secs. 4.9 and 4.10). The small quadrilaterals must evidently become squares as their size approaches zero, if the x and y scales are the same, since from Eqs. (14.16) and (14.23) $|u| = |\partial\phi/\partial x| = |\partial\psi/\partial y|$, or for finite increments, $|\Delta\phi/\Delta x| = |\Delta\psi/\Delta y|$. The difference in value of the stream function between adjacent streamlines is called the ***strength*** of the stream tube bounded by the two streamlines, and it represents the two-dimensional flow through the tube.[5]

Referring to Fig. 14.10, we see that the maximum velocity at any point O is tangential to the streamline. This velocity is given by $V = -\partial\phi/\partial s$, where s is measured along the streamline. Because ϕ does not vary along an equipotential

[4] Methods exist to solve for the conjugate functions of two-dimensional, incompressible, potential flow by using complex variables. In these, we define a ***complex variable potential***, $F(z) = \phi + i\psi$, where $z = x + iy$ and $i = \sqrt{-1}$.

[5] Consider flow in the direction of the x axis in a stream tube (of unit thickness perpendicular to the plane of the paper) bounded by $\psi = K_1$ and $\psi = K_2$. Let y_1 and y_2 represent the graphical locations of the two streamlines. The flow through the tube is then given by

$$Q = \int_{y_1}^{y_2} u\,dy = -\int_{y_1}^{y_2} \frac{\partial\psi}{\partial y}\,dy = -\int_{K_1}^{K_2} d\psi = K_1 - K_2$$

line, perpendicular to s, the partial derivative $\partial\phi/\partial s$ is equal to the total derivative $d\phi/ds$. The expression $\partial\phi/\partial s$ is also known as the ***gradient*** of the velocity potential. Thus we often write the velocity in convenient vector notation as $\mathbf{V} = -\text{grad}\,\phi$, which holds for either two- or three-dimensional flow.[6] We can always write the absolute velocity as the vector sum of its components; thus, in three dimensions,

$$V = |\mathbf{V}| = \sqrt{u^2 + v^2 + w^2} \tag{14.26}$$

Stream functions can exist in the absence of irrotationality, and potential functions are possible even though continuity is not satisfied. But, since lines of ϕ and ψ are required to form an orthogonal network, a flow net can only exist if irrotationality (the condition for the existence of ϕ) and continuity (the condition for the existence of ψ) are satisfied. The Laplace equation (Eq. 14.25) was derived assuming the existence of velocity potentials and the satisfaction of continuity. Thus, if a given flow satisfies the Laplace equation, a flow net can be constructed for that flow. Because of the irrotationality requirement such potential flows are usually those of ideal fluids.

Because the stream function ψ is limited in applicability to two-dimensional flow fields, so is the flow net.

SAMPLE PROBLEM 14.5 An incompressible flow is defined by $u = 2x$ and $v = -2y$. Find the stream function and potential function for this flow and plot the flow net.

Solution
Check continuity:

Eq. (14.3): $$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 2 - 2 = 0$$

Hence continuity is satisfied, and it is possible for a stream function to exist:

Eq. (14.14): $$d\psi = -v\,dx + u\,dy = 2y\,dx + 2x\,dy$$

Integrating: $$\psi = 2xy + C_1 \qquad \textbf{\textit{ANS}}$$

Check to see if the flow is irrotational:

Eq. (14.7): $$\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 - 0 = 0$$

Hence $\xi = 0$, the flow is irrotational, and a potential function exists:

[6] grad ϕ is in the direction of increasing ϕ.

Eq. (14.20): $$d\phi = -u\,dx - v\,dy = -2x\,dx + 2y\,dy$$

Integrating: $$\phi = -(x^2 - y^2) + C_2 \qquad \textbf{\textit{ANS}}$$

Letting $\psi = 0$ and $\phi = 0$ pass through the origin, we get $C_1 = C_2 = 0$.

The location of lines of equal ψ can be found by substituting values of ψ into the expression $\psi = 2xy$. Thus for $\psi = 60$, $x = 30/y$. This line is plotted (in the upper right-hand quadrant) on Fig. S14.5. In a similar fashion we can plot lines of equal potential. For example, for $\phi = -60$ we have $-(x^2 - y^2) = -60$ and $x = \pm\sqrt{y^2 + 60}$. This line is also plotted on the figure. The flow net depicts ***flow in a corner.*** Mathematically the net will plot symmetrically in all four quadrants.

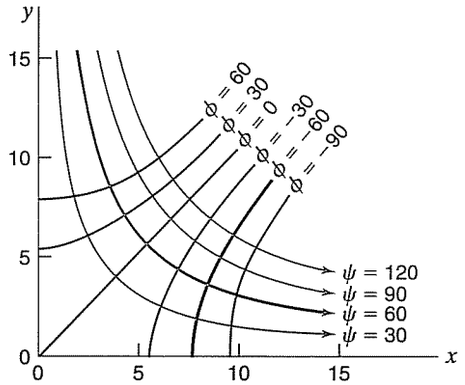


Figure S14.5 Flow net

EXERCISES

14.7.1 Which of the flows in Exer. 14.1.1 can be described by a flow net? Write expressions for the stream functions and the potential functions.

14.7.2 Which of the flows in Exer. 14.1.2 can be described by a flow net? Write expressions for the stream functions and the potential functions.

14.8 FLOW THROUGH POROUS MEDIA

Not only ideal flows satisfy the conditions for potential flow. An example where a real fluid satisfies them is that of ***laminar flow through porous media.*** In such a case, because the seepage velocities are so low, the velocity head is negligible and we can write the energy Eq. (5.28) as

$$\left(\frac{p_1}{\gamma} + z_1\right) - \left(\frac{p_2}{\gamma} + z_2\right) = h_L$$

TABLE 14.1 **Hydraulic conductivities of common geologic deposits**

	K, m/d		*K*, ft/day	
	Low	**High**	**Low**	**High**
Gravel	8.5×10^{1}	7.0×10^{4}	2.8×10^{2}	2.3×10^{5}
Sand	2.5×10^{-1}	8.5×10^{2}	8.5×10^{-1}	2.8×10^{3}
Silt	1.3×10^{-4}	1.8	4.2×10^{-4}	6.0
Clay	4.3×10^{-8}	1.7×10^{-4}	1.5×10^{-7}	5.5×10^{-4}

where the head loss h_L is directly proportional to velocity for laminar flow (see Secs. 8.1 and 8.7). Taking differentials, we get

$$-d\left(\frac{p}{\gamma} + z\right) = dh_L = \frac{V}{K} ds$$

where we define V as the flow per unit cross-sectional area of media, ds is the distance along the streamline, and $1/K$ is a constant of proportionality. So, using Eqs. (14.23),

$$V = -K\frac{d(p/\gamma + z)}{ds} = K\frac{dh_L}{ds} = -\frac{d\phi}{ds} \tag{14.27}$$

Thus we see for porous media flow that $\phi = K(p/\gamma + z)$. The constant K is known as the ***hydraulic conductivity;*** it has the units of velocity, and has high values for gravels and sands and low values for clays (see Table 14.1). The quantity $p/\gamma + z$ is called the ***hydraulic head,*** usually represented by the variable h, so $\phi = Kh$. Note that $(p/\gamma + z)$, h, and ϕ all decrease as s increases, accounting for the minus signs in Eq. (14.27); and $dh_L = -dh$. The most convenient and most used form of Eq. (14.27) is

$$V = -K\frac{dh}{ds} \tag{14.28}$$

where dh/ds is the ***hydraulic gradient,*** which is negative because h decreases in the direction of flow, although it is often given as a positive number.

Equation (14.27) is known as ***Darcy's law,*** in honor of Henry Darcy[7] (1803–1858), a French engineer. He was the first to propose such a relationship, in the form $V = K\,dh_L/ds$, published in 1856, based on his experiments with

[7] The frequently used spellings Henri and D'Arcy are incorrect. Darcy's law has also been frequently misrepresented as $V = -(K/\gamma)\,dp/ds$, which is clearly erroneous, compare with Eq. (14.27). His law has also often been incorrectly reported to be limited to one-dimensional flow, and to horizontal flow. For an interesting discussion of a major source of these errors, see M.K. Hubbert, Review and Discussion of "Physical Principles of Oil Production," *J. Geol.,* Vol. 58, No. 6, 1950, pp. 655–660.

vertical flow through sand used for water filtration. As a result, he is recognized as the father of groundwater hydraulics. Darcy's law today remains an empirical law, based only on experimental evidence. Subsequently it has been observed that the proportionality does not hold if the flow is not laminar,[8] which seldom occurs with groundwater flow.

We must make a distinction between the ***specific discharge*** (or apparent velocity or Darcy velocity) $V = Q/A$, where A is the total cross-sectional area of the media, and V_p, the ***average pore velocity.*** If n is the porosity of the medium, equal to the volume of the pores divided by the total volume, then the cross-sectional area of the pores $A_p = nA$, so that $V_p = Q/A_p = AV/nA$, resulting in

$$V_p = \frac{V}{n} \tag{14.29}$$

The top of the saturated groundwater zone, where not confined, is known as the ***water table.*** There, neglecting small surface tension effects, the pressure is atmospheric, or zero gage pressure, and so $p/\gamma + z = h = z$. From elevations of surface water bodies, and of water levels in unpumped wells, contours of water table elevations can be prepared. Since the elevations of the water table contours correspond to the hydraulic head h, it follows from the orthogonality principle that where the groundwater flow is nearly horizontal the flow will be perpendicular to the water table contours. This, then, is another useful application for flow nets, helping determine groundwater flow directions. Furthermore, from Eq. (14.27) and an estimate of K, groundwater flow velocities can be calculated.

Flow nets are used in many other situations to estimate flow rates through soils, such as through earth dams and under impervious dams.[9]

Seepage and groundwater hydraulics are important for economic, environmental, and other reasons. For example, contaminants in groundwater are transported by the groundwater flow, and because flow velocities are usually very slow, more or less in the range of a foot per year to a foot per day (0.3 to 110 m/y), it can take a long time to identify contamination, and to remedy it. Overpumping of groundwater, say for agricultural use, lowers the water table and increases the cost of pumping. In coastal areas it can lead to contamination by saltwater intrusion. Water table lowering is used beneficially to dewater construction sites.

Darcy's law is used in a number of other important areas besides the analysis of groundwater flow. These include the approximate analysis of unsaturated soil moisture movement, the flow of oil in petroleum reservoirs, and the design of chemical and ceramic filters and related processes.

[8] For flow through a granular medium, the flow is laminar if the Reynolds number has a value less than 1.0, where the Reynolds number is defined as $d_{10}V\rho/\mu$, where d_{10} is the effective particle size (the 10% finer than value), V is the Darcy velocity defined by Eq. (14.27), and ρ and μ are the properties of the fluid that is passing through the medium. At higher Reynolds numbers (larger particles, larger gradient, and lower fluid viscosity) the head loss tends to become proportional to V^2.

[9] See Ref. 99 in Appendix E.

SAMPLE PROBLEM 14.6 Groundwater is flowing through an aquifer, 100 ft thick and 5000 ft wide, that has a porosity of 0.2 and a hydraulic conductivity of 0.1 ft/day. If the hydraulic gradient is −0.035, what is the flow rate through the aquifer in ft³/day? To assess the speed of migration of contaminants, find how long on average it takes a particle of water to travel 100 ft?

Solution

Eq. (14.27): $V = -K(dh/ds) = -0.1(-0.035) = 0.0035$ ft/day

$$Q = AV = 100(5000)0.0035 = 1750 \text{ ft}^3/\text{day} \quad \textit{ANS}$$

Eq. (14.28): $V_p = V/n = 0.0035/0.2 = 0.0175$ ft/day

$$t = s/V_p = 100/0.0175 = 5714 \text{ days} = 15.6 \text{ yr} \quad \textit{ANS}$$

EXERCISES

14.8.1 The estimated average depth of saturated flow in a sloping water table aquifer (porosity 0.25, $K = 5$ ft/day) is 11.8 ft. The distance between 1-ft water table contours is 34 ft. What is the flow rate through a 1-ft width of aquifer, and what is the average pore velocity?

14.8.2 The estimated average depth of saturated flow in a sloping water table aquifer (porosity 0.23, $K = 1.6$ m/d) is 3.8 m. The distance between 1-m water table contours is 37 m. What is the flow rate through a 1-m width of aquifer, and what is the average pore velocity?

14.8.3 From dye tests between two wells, groundwater is estimated to travel at 0.034 ft/day. If the corresponding water table slope is 2.25 in per 100 ft, and the average porosity of the aquifer is 0.15, estimate its hydraulic conductivity.

14.8.4 From dye tests between two wells, groundwater is estimated to travel at 0.0087 m/d. If the corresponding water table slope is 175 mm per 100 m, and the average porosity of the aquifer is 0.19, estimate its hydraulic conductivity.

PROBLEMS

14.1 Given a flow defined by $u = 3 + 2x$. If this flow satisfies continuity, what can be said about the density of the fluid?

14.2 Why are Eqs. (14.2) and (14.3) applicable to real fluids as well as ideal fluids?

14.3 Derive Eqs. (14.4) and (14.5).

14.4 Sketch streamlines ($\psi = 0, 1, 2, 3$) for the following flow fields, note the values of u and v, and verify that continuity is satisfied in all cases: (*a*) $\psi = 10y$; (*b*) $\psi = -20x$; (*c*) $\psi = 10y - 20x$.

14.5 A flow field is described by the equation $\psi = 1.2xy$. Sketch the streamlines in one quadrant for $\psi = 0, 1, 2, 3, 4$.

14.6 Plot the streamlines in the upper right-hand quadrant for the flow defined by $\psi = 1.5x^2 + y^2$ and determine the value of the velocity at $x = 4$, $y = 2$.

14.7 The components of the velocities of a certain flow system are

$$u = -\frac{Q}{2\pi}\frac{x}{x^2 + y^2} + By + C \text{ and}$$

$$v = -A\frac{y}{x^2 + y^2} + Dx + E$$

(*a*) Calculate a value of A consistent with continuous flow. (*b*) Sketch the streamlines for this flow system, assuming $B = C = D = E = 0$.

14.8 A flow field is described by $\psi = x^2 - y$. Sketch the streamlines for $\psi = 0$, 1, and 2. Derive an expression for the velocity at any point in the flow field and determine the vorticity of the flow.

14.9 A source of strength 8π is located at (2, 0). Another source of strength 16π is located at (−3, 0). For the combined flow field produced by these two sources: (*a*) find the location of the stagnation point; (*b*) plot the $\psi = 0$, $\psi = 4\pi$, $\psi = 8\pi$ lines; (*c*) find the values of ψ at (0, 2) and at (3, −1); (*d*) find the velocity at (−2, 5).

14.10 Using the method described in Sample Prob. 14.4, plot the boundary of the body and a set of streamlines for a steady two-dimensional flow past a body such as that of Fig. 4.12, for $b = 15$ m using a scale of 1 cm = 2 m.

14.11 Combine the uniform flow defined by $U = 16$ ft/sec with the doublet $2qa = m$, where $q = 10$ cfs/ft and $a = 2$ in. Sketch the streamlines for $\psi = -3, -2, -1, -\frac{1}{2}$, $0, \frac{1}{2}, 1, 2$, and 3 cfs/ft. Use a scale of 1 in = 1 in.

14.12 A flow is defined by the stream function $\psi = 15r\sin\theta - 30\ln r - (20/r)\sin\theta$. Sketch this flow field. Calculate the velocities at $r = 3$ for $\theta = 0°, 45°, 90°, 150°, 210°$, and 315°.

14.13 Consider the two-dimensional flow described by $u = x^2 + 2x - 4y$, $v = -2xy - 2y$. (*a*) Does this satisfy continuity? (*b*) Compute the vorticity. (*c*) Plot the velocity vectors for $0 < x < 5$ and $0 < y < 4$ and sketch the general flow pattern. (*d*) Find the location of all stagnation points in the entire flow field. (*e*) Find the expression for the stream function.

14.14 The flow around the body of Fig. 4.12 may be considered as that due to the sum of two velocity potentials, $\phi_1 = -Ux$, representing an undisturbed flow of velocity U in the x direction, and $\phi_2 = -S\ln r$, representing the radial flow from a source located inside the body behind the stagnation point. To relate U and S, it is observed that the total flow $2\pi S$ from the source (which is hydrodynamically equivalent to the body itself) must be equal to the flow of the main stream that is not passing through the body of width b, or $2\pi S = Ub$. This gives

$$\phi_2 = -\frac{Ub}{2\pi}\ln r$$

(*a*) The distance from the stagnation point to the source is determined by setting the radial velocity from the source, $v_r = -\partial\phi/\partial r$, equal and opposite to the undisturbed velocity U. Prove that this establishes the source at a distance $b/2\pi$ behind the stagnation point. The absolute velocity at any point of the field may be determined by the vector sum of the components U and v_r.

There follows an ingenious method of plotting the boundary of such a streamlined body, as shown in Fig. P14.14. Suppose that the streamlines in the undisturbed flow are spaced a distance a apart, where $b/2a = n$, an integer. Next divide the upper half of the source into n radial sectors, each of angle α, that is,

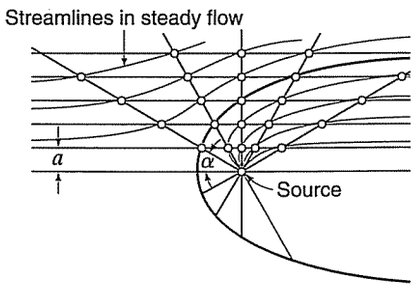


Figure P14.14

$n\alpha = \pi$. Then the undisturbed flow between the x axis and the first streamline is associated with the source flow in the first sector from the stagnation point. Thus the intersection of the first streamline with the first line must be a point on the boundary of the body, through which there can be no flow. Similarly, the intersection of the horizontal line at $2a$ with the radial line at 2α forms another point, and so on. Further streamlines can be plotted by connecting successive intersections of the original horizontal lines with the radial lines, recognizing that the same flow must exist between any adjacent pair of streamlines. Thus the intersection of a horizontal line ea above the axis with a radial line at $f\alpha$ from the stagnation point must lie on a streamline which is $(e - f)a$ distant from the axis in the undisturbed region, where e and f are integers.

(b) Assume a value of $U = 20$ ft/sec and a two-dimensional flow past a streamlined body for which $b = 36$ ft. Compute the distance from the source to the stagnation point and to the surface of the body at a radius 90° to the axis. What is the value of the source velocity at the latter point? (c) What is the magnitude of the velocity of the fluid along the surface at the 90° point? (Compare with the results of Sample Prob. 4.2). What is its direction relative to the axis?

14.15 Solve Prob. 14.14 in SI units; for parts (b) and (c) use $U = 5$ m/s and $b = 12$ m.

14.16 Find the distance to the surface of the body and the two velocities called for in Prob. 14.14 for an angle of 30°. (Compare with Sample Prob. 4.2).

14.17 Find the distance to the surface of the body and the two velocities called for in Prob. 14.15 for an angle of 30°. (See Sample Prob. 4.2).

14.18 An ideal fluid flows around a two-dimensional 90° bend (Fig. P14.18). The inner and outer radii of the bend are 0.45 and 1.50 ft. (a) Sketch the flow net and estimate the velocity at the inner and outer walls of the bend if the velocity in the 1.05-ft-wide straight section is 6 ft/sec. (b) Develop an analytic expression for the stream function, in this case noting that $v_t = -\partial\psi/\partial r$ and $v_r = r^{-1}\partial\psi/\partial\theta$. Determine the inner and outer velocities accurately.

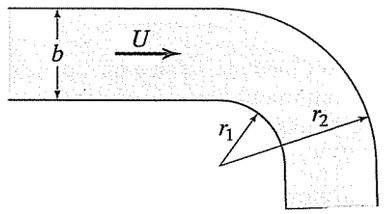


Figure P14.18

14.19 An ideal fluid flows around a two-dimensional 90° bend (Fig. P14.18). The inner and outer radii of the bend are 0.15 and 0.50 m. (a) Sketch the flow net and estimate the velocity at the inner and outer walls of the bend if the velocity in the 0.35-m-wide straight section is 2.8 m/s. (b) Develop an analytic expression for the stream function, in this case noting that $v_t = -\partial\psi/\partial r$ and $v_r = r^{-1}\partial\psi/\partial\theta$. Determine the inner and outer velocities accurately.

14.20 For the two-dimensional flow of a frictionless incompressible fluid against a flat plate normal to the initial velocity, the stream function is given by $\psi = -2axy$, while its conjugate function, the velocity potential, is

$$\phi = a(x^2 - y^2)$$

where a is a constant and the flow is symmetrical about the yz plane (Fig. 14.1). By direct differentiation, demonstrate that these functions satisfy Eq. (14.25). Using a scale of 1 in = 1 unit of distance, plot the streamlines given by $\psi = \pm 2a, \pm 4a, \pm 6a, \pm 8a$, and the equipotential lines given by $\phi = 0, \pm 2a, \pm 4a, \pm 6a, \pm 8a$. Observe that this flow net also gives the ideal flow around an inside square corner. Compare your results with Sample Prob. 14.5 and note the effect of changing the sign of ψ and ϕ.

14.21 In Prob. 14.20 determine the velocity components u and v, and demonstrate that they satisfy the differential equations for continuity and irrotational flow. In which direction is the flow? Prove that the absolute velocity is given by $V = 2ar$, where r is the radius to the point from the origin. Now assume that the linear scale is 1 in = 1 ft. Determine the constant a such that the flow net of Prob. 14.20 will represent a flow of 3 ft²/sec between any two adjacent streamlines. What are the dimensions of a? Draw curves of equal velocity for values of 3, 6, 9, and 12 ft/sec. How does the velocity vary along the surface of the plate?

14.22 A cylindrical drum with a 2-ft radius is securely held in position in an open channel of rectangular section. The channel is 10 ft wide, and the flow rate is 240 cfs. Water flows beneath the drum as shown in Fig. P14.22. Sketch the flow net, and determine from flow net measurements the pressure at the points indicated along the wetted drum surface. Neglect fluid friction. Sketch the pressure distribution, and by numerical integration determine an approximate value of the horizontal thrust on the cylinder. (Compare with Exer. 6.4.1).

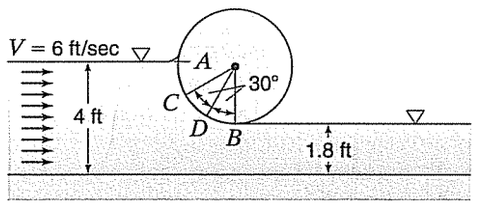


Figure P14.22

14.23 Refer to Sample Prob. 6.1. Sketch a flow net. Using the given dimensions in BG units through application of Bernoulli's principle, determine the approximate pressure distribution along the channel bottom and around the curved structure. By numerical integration estimate the magnitude of the horizontal and vertical components of the force of the water on the structure.

14.24 Work Prob. 14.23 using the dimension as given in SI units in Sample Prob. 6.1.

14.25 The three-dimensional counterpart of the flow in Probs. 14.20 and 14.21 is that of flow along the y axis approaching the plate in the xz plane. As the flow must be symmetrical about the y axis, the traces of stream and equipotential surfaces in the xy plane will be representative of those in all planes containing the y axis. The velocity potential is now given by $\phi = -a(0.5x^2 - y^2)$, and the stream function by $\psi = -ax^2y$. Notice that these functions no longer satisfy Eq. (14.25). Why not? Again plot streamlines and equipotential lines for the values given in Prob. 14.20. The velocities u and v may still be determined by Eq. (14.23). Prove that the absolute velocity for this case is given by $V = a\sqrt{x^2 + 4y^2}$. With the value of $a = 1.5$ sec^{-1} found in Prob. 14.21, draw curves of equal velocity for values of 3, 6, 9, and 12 ft/sec. How does the velocity vary along the surface? What is the total flow between any two adjacent stream surfaces?

14.26 For the two-dimensional flow around any angle α, the velocity potential and stream function are given in polar coordinates as $\phi = -ar^{\pi/\alpha}\cos(\pi\theta/\alpha)$ and $\psi = -ar^{\pi/\alpha}\sin(\pi\theta/\alpha)$, respectively. Prove that the functions given in Prob. 14.20 are a specialization of these expressions for $\alpha = \pi/2$. Take the case of $\alpha = 3\pi/2$, and plot streamlines and equipotential lines for the values given in Prob. 14.20. Compare the velocity at the corner with that at the corner in Prob. 14.20.

14.27 Superimpose a point source ($Q = 100$ cfs) on a rectilinear flow field ($U = 20$ ft/sec). Plot the body contour at $\theta = 30°, 60°, 90°, 120°, 150°$, and 180° using a scale of 1 in = 1 ft. Compute the velocities along the body contour at these points. Determine the pressures at these points assuming $\rho = 1.94$ slug/ft³ with zero pressure in the undisturbed rectilinear flow field. What is the velocity and pressure in the combined flow field at the following points?
Hint: Refer to Prob. 14.14.
(a) $\theta = 45°$, $r = 4.0$ ft
(b) $\theta = 90°$, $r = 2.0$ ft
(c) $\theta = 90°$, $r = 4.0$ ft
(d) $\theta = 135°$, $r = 2.0$ ft

14.28 Figure P14.28 represents a steady state flow condition through a timber crib dam. The crib is in two sections, each containing a different soil; $a = 8$ ft, $b = 14$ ft, $c = 5$ ft, and $d = 4$ ft. (*a*) Determine the hydraulic conductivity of the soil in Region 1 if the soil in Region 2 has a hydraulic conductivity of 300 gpd/ft^2. (*b*) Determine the seepage rate through the dam in gpd per foot of dam length (i.e., per foot perpendicular to the plane of the figure).

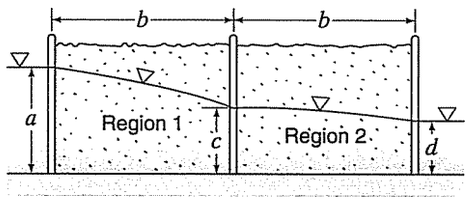


Figure P14.28

14.29 Figure P14.28 represents a steady state flow condition through a timber crib dam. The crib is in two sections, each containing a different soil; $a = 2.5$ m, $b = 4.0$ m, $c = 1.5$ m, and $d = 1.2$ m. (*a*) Determine the hydraulic conductivity of the soil in Region 1 if the soil in Region 2 has a hydraulic conductivity of 12 m/d. (*b*) Determine the seepage rate through the dam in L/h per meter of dam length (i.e., per meter perpendicular to the plane of the figure).

CHAPTER 15
Hydraulic Machinery—Pumps[1]

There are various types of fluid machinery. Among them are those that transfer fluid energy (***torque converters***), those that convert mechanical energy to fluid energy (***pumps***), and those that convert fluid energy to mechanical energy (***turbines***). The conversion of mechanical energy to fluid energy is accomplished by pumps for the case of incompressible fluids, and by ***blowers, fans,*** and ***compressors*** for the case of compressible fluids. In this chapter we shall confine our discussion to pumps. In Secs. 6.11–6.14 there is a discussion of flow through rotating machinery and rotating conduits; we suggest the reader review that material before proceeding further. Since pumps are much more prevalent than turbines, we discuss pumps first. We shall deal with ***centrifugal and axial-flow pumps,*** but shall not discuss positive displacement pumps such as reciprocating piston and rotary types.

In our discussion of hydraulic machinery, we will emphasize the selection of pumps (and turbines) for particular situations. To accomplish this, we shall first describe the various types of pumps (and turbines) and discuss their performance characteristics. Finally we will show how pumps (and turbines) fit into the hydraulics of the systems in which they operate. We shall find that cavitation plays an important role in the selection of pumps and turbines.

15.1 DESCRIPTION OF CENTRIFUGAL AND AXIAL-FLOW PUMPS

The rotating element of a centrifugal pump is called the ***impeller*** (Fig. 15.1). The impeller may be shaped to force water outward in a plane at right angles to its axis (***radial flow***), to give the water an axial as well as radial velocity (***mixed flow***), or to induce a spiral flow on coaxial cylinders in an axial direction

[1] In this chapter and the next, because of the difficulty of carrying everything in both sets of units, for the most part we shall deal with pumps and turbines using conventional English (BG) units in conformance with practice in the United States. However, a substantial number of sample problems, exercises and problems will be expressed in SI metric units.

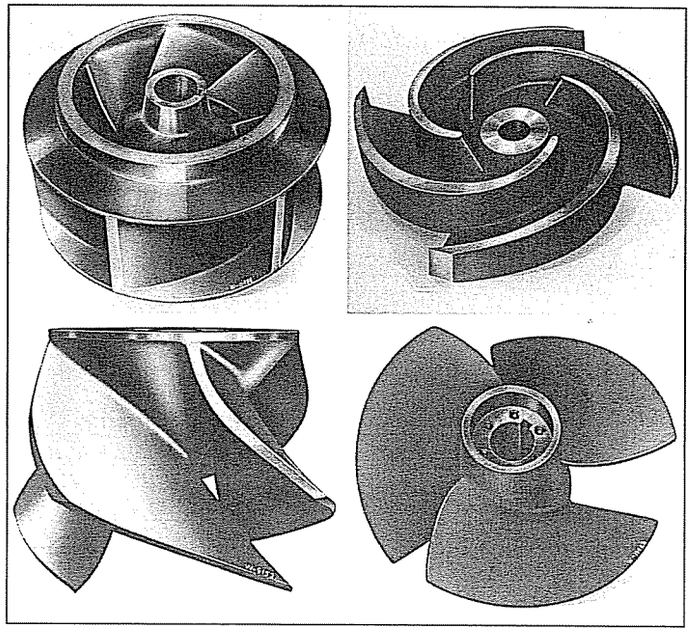


Figure 15.1
Types of pump impellers. Top left: closed or shrouded radial. Top right: open or unshrouded radial. Bottom left: mixed flow. Bottom right: propeller. (Worthington Pump Co.)

(***axial flow***). Radial-flow and mixed-flow machines are commonly referred to as centrifugal pumps, while axial-flow machines are called axial-flow pumps or ***propeller pumps.*** Radial- and mixed-flow impellers may be either open or closed. The open impeller consists of a hub to which vanes are attached, while the closed impeller has plates (or ***shrouds***) on each side of the vanes. The open impeller does not have as high an efficiency as the closed impeller, but it is less likely to become clogged and hence is suited to handling liquids containing solids.

Radial-flow pumps are provided with a ***spiral casing,*** often referred to as a ***volute casing*** (Fig. 15.2), which guides the flow from the impeller to the discharge pipe. The ever increasing flow cross section around the casing tends to maintain a constant velocity within the casing. This helps to provide relatively smooth flow conditions at exit from the impeller. Some pumps have diffuser vanes instead of a volute casing. Such pumps are known as ***turbine pumps.*** Some radial pumps are of the double-suction type. They have identical, mirror-image impellers placed back to back. Water enters the pump from both sides and is discharged into a volute casing or diffuser vanes. The advantage of the double-suction pump is the reduced mechanical friction that results because the thrust on the bearings is balanced.

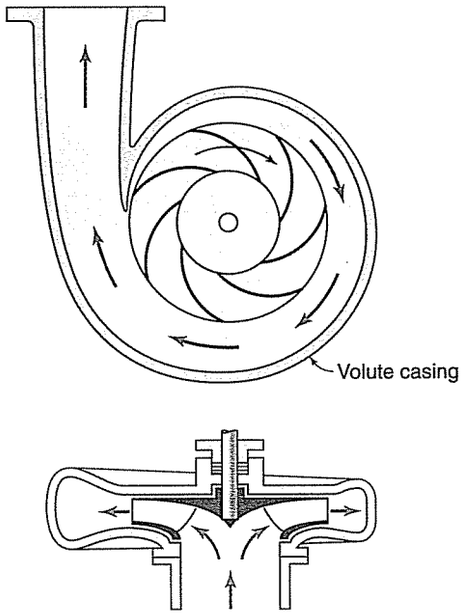


Figure 15.2
Radial-flow centrifugal pump with volute casing.

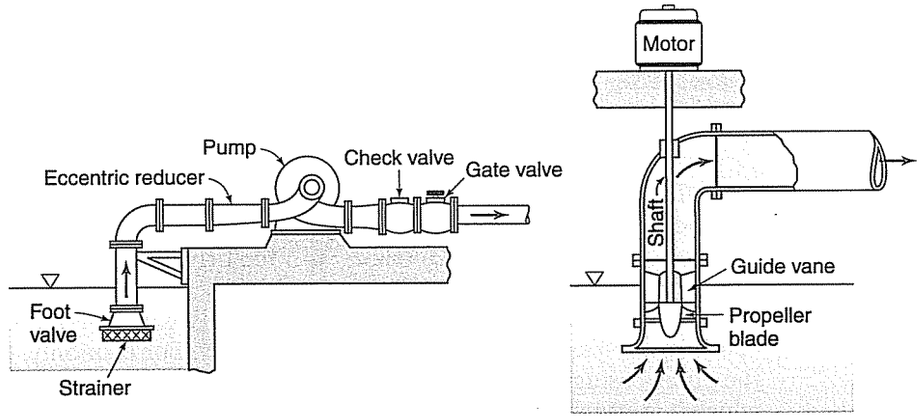


Figure 15.3
(*a*) Typical centrifugal pump installation. (*b*) Typical axial-flow pump installation.

Typical centrifugal-flow and axial-flow pump installations are shown in Fig. 15.3. Pumps can be ***single-stage*** or ***multistage.*** A single-stage pump has only one impeller, while a multistage has two or more impellers arranged in such a way that the discharge from one impeller enters the eye of the next impeller. Deep-well pumps (Fig. 15.4), a type of turbine pump, are usually multistage, having

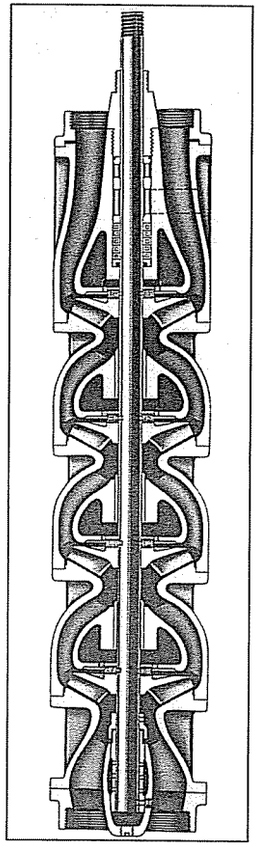


Figure 15.4
Deep-well multistage mixed-flow turbine pump. (Courtesy of Byron Jackson Company)

several impellers on a vertical shaft suspended from a prime mover, usually an electric motor, located at the ground surface. Each impeller discharges into a fixed-vane diffuser, or bowl, coaxial with the drive shaft, which directs water to the next impeller.

Proper arrangement of the suction and discharge piping is necessary if a centrifugal pump is to operate at best efficiency. For economy, the diameter of the pump casing at suction and discharge is often smaller than that of the pipe to which it is attached. If there is a horizontal reducer between the suction and the pump, we should use an ***eccentric reducer*** (Fig. 15.3*a*) to prevent air accumulation. A ***foot valve*** (check valve) can be installed in the suction pipe to prevent water from leaving the pump when it is stopped. The discharge pipe is usually provided with a check valve and a gate valve. The ***check valve*** prevents backflow through the pump if there is a power failure. Suction pipes taking water from a sump or reservoir are usually provided with a screen to prevent entrance of debris that might clog the pump.

Axial-flow pumps (Fig. 15.3*b*) usually have only two to four blades and, hence, large unobstructed passages that permit handling of water containing debris without clogging. The blades of some large axial-flow pumps are adjustable to permit setting the pitch for the best efficiency under existing conditions.

15.2 HEAD DEVELOPED BY A PUMP

The head h developed by a pump is determined by measuring the pressures on both the suction and discharge sides of the pump, computing the velocities by dividing the measured discharge by the respective cross-sectional areas, and noting the difference in elevation between the suction and discharge sides. The net head h delivered by the pump to the fluid is

$$h = H_d - H_s = \left(\frac{p_d}{\gamma} + \frac{V_d^2}{2g} + z_d\right) - \left(\frac{p_s}{\gamma} + \frac{V_s^2}{2g} + z_s\right) \tag{15.1}$$

where the subscripts d and s refer to the discharge and suction sides of the pump, as shown in Fig. 15.5. If the discharge and suction pipes are the same size, the velocity heads cancel out, but frequently the intake pipe is larger than the discharge pipe. Note that h, the head put into the fluid by the pump, we previously designated h_M in Sec. 5.5 and h_p in Sec. 8.28.

The official test code provides that the head developed by a pump be the difference between the total energy heads at the intake and discharge flanges. However, flow conditions at the discharge flange are usually too irregular for accurate pressure measurement, and we can more reliably measure the pressure at 10 or more pipe diameters away from the pump and add an estimated pipe friction head for that length of pipe. On the intake side, ***prerotation*** sometimes

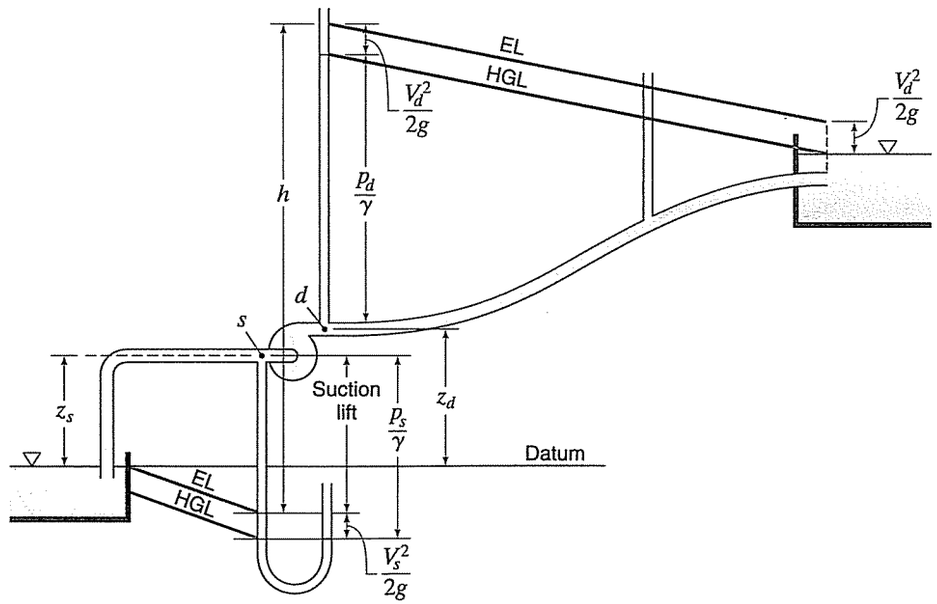


Figure 15.5
Head developed by a pump. In this case p_s/γ is negative.

exists in the pipe near the pump, and this will cause the pressure reading on a gage to be different from the true average pressure at that section.

15.3 PUMP EFFICIENCY

As liquid flows through a pump, only part of the energy imparted to the shaft of the impeller is transferred to the flowing liquid. There is friction in the bearings and packings, not all liquid passing through the pump is effectively acted upon by the impeller, and there is substantial loss of energy due to fluid friction which has a number of components including shock loss at entry to the impeller,[2] fluid friction as the fluid passes through the space between the vanes or blades, and head loss as the fluid leaves the impeller. The efficiency of a pump is quite sensitive to the conditions under which it is operated, as will be discussed in Sec. 15.5.

The power input to the pump, delivered to the pump shaft by the motor, is called the ***shaft power*** or the ***brake power.*** The power output from the pump, delivered to the fluid, usually water, is called the ***fluid power*** or the ***water power.*** From Eq. (5.42) then, the efficiency η (eta) of a pump is given by

$$\eta = \frac{\text{output power}}{\text{input power}} = \frac{\text{fluid power}}{\text{shaft (brake) power}} = \frac{\gamma Q h}{T\omega} \tag{15.2}$$

where γ, Q, and h are defined in the usual fashion; T is the torque exerted on the shaft of the pump by the motor that drives the shaft, and ω is the rate of rotation of the shaft in radians per second.

15.4 SIMILARITY LAWS FOR PUMPS

Similarity laws (Secs. 7.2–7.5) enable us to predict the performance of a prototype pump (or turbine) from the test of a scaled model. Moreover, and of particular value in pump selection, these laws make it possible to predict the performance of a given machine under different conditions of operation from those under which it has been tested.

Similarity laws are based on the concept that two geometrically similar machines (i.e., same scale change in all three dimensions) with similar velocity diagrams at entrance to and exit from the rotating element are ***homologous.*** This means that their streamline patterns will be geometrically similar, i.e., that their behaviors will bear a resemblance to one another.

We can derive similarity laws by dimensional analysis. The most significant variables[3] affecting the operation of a turbomachine are the head h, the

[2] Shock loss occurs when the flow does not enter the impeller smoothly. This results in separation of the flow from the impeller blade.

[3] If we wish to relate the operation of one pump to another with different fluids in each then kinematic viscosity is a significant variable.

discharge Q, the rotative speed n, the diameter of the rotor D, and the acceleration due to gravity g. Thus, from the Buckingham II-theorem (Sec. 7.7), since there are five dimensional variables and two fundamental dimensions (L and T), there will be three dimensionless groups. We have

$$f(h, Q, n, D, g) = 0$$

Upon grouping these variables into dimensionless quantities, we get

$$f'\left(\frac{Q}{nD^3}, \frac{g}{n^2D}, \frac{h}{D}\right) = 0$$

Laboratory tests on turbomachines have demonstrated that the second dimensionless quantity is inversely proportional to the third. These can be combined to give

$$\frac{g}{n^2D} = K\frac{D}{h} \quad \text{and} \quad K = \frac{gh}{n^2D^2}$$

Thus

$$f''\left(\frac{Q}{nD^3}, \frac{gh}{n^2D^2}\right) = 0 \tag{15.3}$$

From Eq. (15.3), we find

$$Q \propto nD^3 \quad \text{or} \quad Q = K_Q nD^3 \tag{15.4}$$

and, assuming g is constant,[4]

$$h \propto n^2D^2 \quad \text{or} \quad h = K_h n^2D^2 \tag{15.5}$$

Since power $P \propto Qh$ (Sec. 5.9), we get

$$P \propto n^3D^5 \quad \text{or} \quad P = K_P n^3D^5 \tag{15.6}$$

For any one design of a pump, K_Q, K_h, and K_P can be evaluated, preferably from test data, and then used to predict the performance of homologous pumps. Similarity laws are of great practical value, but care must be exercised when applying them. Thus, in comparing two machines of different sizes, the two must be homologous and the variation in the values of h, D, and n should not be too large. For example, a machine that operates satisfactorily at low speeds may cavitate at high speeds. The values of K in each of Eqs. (15.4)–(15.6) change somewhat as h, D, and n are varied, because the efficiencies of homologous machines are not identical. Large machines are usually more efficient than smaller ones, because their flow passages are larger. Also, efficiency usually increases with speed of rotation, because power output varies with the cube of the

[4] If we were on the moon, or in some gravitational field different than that of the earth, we must include g in Eqs. (15.5) and (15.6).

speed while mechanical losses increase only as the square of the speed. As a result, Eqs. (15.4)–(15.6) are approximate.

An empirical equation suggested by Moody,[5] originally intended for application to homologous reaction turbines (Sec. 16.9), that gives fairly reasonable results for estimating the efficiency of a prototype pump from the test of a geometrically similar (model) pump is

$$\frac{1-\eta_p}{1-\eta_m} \approx \left(\frac{D_m}{D_p}\right)^{1/5} \tag{15.7}$$

The effects of rotation speed n on the performance (Q, h, P) of a given pump are known as ***affinity laws.*** Equations (15.4)–(15.6) govern these effects, which generally change pump efficiency little. The effects of trimming the impeller outside diameter on the same performances are also known as affinity laws. In this case the same equations do *not* govern, instead $Q \propto D$, $h \propto D^2$, $P \propto D^3$. Because trimming increases losses through larger clearances, it can reduce efficiency appreciably and thereby affect performance, so we should only use these relations for small diameter changes.

SAMPLE PROBLEM 15.1 Develop a similarity equation for torque.

Solution

Power $= \gamma Qh = T\omega$

For water, with $g = 32.2$ ft/s^2, we can neglect γ.

Thus, since $Q = K_Q n D^3$ and $h = K_h n^2 D^2$, we get

$$(K_Q n D^3)(K_h n^2 D^2) = T\omega, \quad \text{but} \quad \omega = 2\pi n/60$$

Hence $$T = K_Q K_h \left(\frac{60}{2\pi n}\right) n^3 D^5 = K_T n^2 D^5 \qquad \textit{ANS}$$

SAMPLE PROBLEM 15.2 A small pump serving as a model has a diameter of 7.4 in. When tested in the laboratory at 3600 rpm, it delivered 3.0 cfs at a head of 125 ft. (*a*) If the efficiency of this model pump is 84%, what is the horsepower input to this pump? (*b*) Predict the speed, capacity, and horsepower input to the prototype pump if it is to develop the same head as the model pump and the model pump has a scale ratio (Sec. 7.2) of 1:10. Assume the efficiency of the prototype pump is 90%.

Solution

(*a*) From Eq. (15.2): $P_m = T\omega = \gamma Qh/550\eta$ hp

$$P_m = 62.4(3)125/[550(0.84)] = 50.6 \text{ hp} \qquad \textit{ANS}$$

[5] See J. H. T. Sun, "Hydraulic Machinery," page 21.26, in V. J. Zipparro and H. Hasen (Eds.), *Davis' Handbook of Applied Hydraulics*, 4th ed., McGraw-Hill, New York, 1993.

(*b*) From Eq. (15.5) and Sec. 7.2:

$$n_p = n_m(h_p/h_m)^{1/2}(D_m/D_p) = 3600(1)^{1/2}(1/10) = 360 \text{ rpm} \qquad \textit{ANS}$$

Eq. (15.4): $$Q_p = Q_m(D_p/D_m)^3 n_p/n_m = 3\times10^3 \times 1/10 = 300 \text{ cfs} \qquad \textit{ANS}$$

Eq. (15.6): $$P_p = 62.4(300)125/[550(0.90)] = 4730 \text{ hp} \qquad \textit{ANS}$$

Note: To satisfy homologous conditions (streamlines geometrically similar) the larger prototype pump must operate at a smaller rotative speed than the model pump.

EXERCISES

15.4.1 A model centrifugal pump has a scale ratio of 1:15. The model when tested at 3600 rpm, delivered 0.10 m^3/s of water at a head of 40 m with an efficiency of 80%. Assuming the prototype has an efficiency of 88%, what will be its speed, capacity, and power requirement at a head of 50 m?

15.4.2 A 19½-in-diameter centrifugal-pump impeller discharges 20 cfs at a head of 100 ft when running at 1200 rpm. (*a*) If its efficiency is 85%, what is the brake horsepower, i.e., what is the horsepower input to the shaft of the pump? (*b*) If the same pump were run at 1500 rpm, what would be h, Q, and the brake horsepower for homologous conditions?

15.4.3 An axial-flow pump delivers 300 L/s at a head of 6.0 m when rotating at 1800 rpm. (*a*) If its efficiency is 80%, how many kilowatts of power must the shaft deliver to the pump? (*b*) If this same pump were operated at 1500 rpm, what would be h, Q, and the power delivered by the shaft for homologous conditions?

15.4.4 In Sample Prob. 15.2 the model with a scale ratio of 1:10 has a maximum efficiency of 84%. Using Eq. (15.7), calculate the maximum efficiency of the prototype pump and compare the result with the data given in the sample problem. Note that Eq. (15.7) is empirical, and will provide only approximate values.

15.5 PERFORMANCE CHARACTERISTICS OF PUMPS AT CONSTANT SPEED

The efficiency of a pump varies considerably, depending upon the conditions under which it must operate. Because of this, when selecting a pump for a given situation, it is important for the pump selector to have information regarding the performance of various pumps among which the selection is to be made. The pump manufacturer usually has information of this type, as determined by laboratory tests, for what are called shelf items, i.e., standard pumps. Large-capacity pumps, however, are sometimes custom-made. Often a model of such a pump is made and tested before final design of the prototype pump.

TABLE 15.1 **Operating speeds of constant-speed electric motors**

	60-cycle		50-cycle	
Pairs of poles	**Synchronous**	**Induction[a]**	**Synchronous**	**Induction[a]**
1	3600 rpm	3500 rpm	3000 rpm	2900 rpm
2	1800	1750	1500	1450
3	1200	1160	1000	960
4	900	870	750	720
5	720	695	600	575
6	600	580	500	480
8	450	435	375	360
10	360	350	300	290
12	300	290	250	240

[a] These values for the operating speed of induction motors are approximate. The speed of induction motors is usually 2–3% lower than that of synchronous motors.

Though some centrifugal pumps are driven by variable-speed motors, the usual mode of operation of a pump is at constant speed. Typical speeds of constant-speed electric motors are given in Table 15.1. The ***pump characteristic curve*** (head versus capacity) and other performance curves for a typical mixed-flow centrifugal pump are shown in Fig. 15.6. This particular pump has a ***normal***

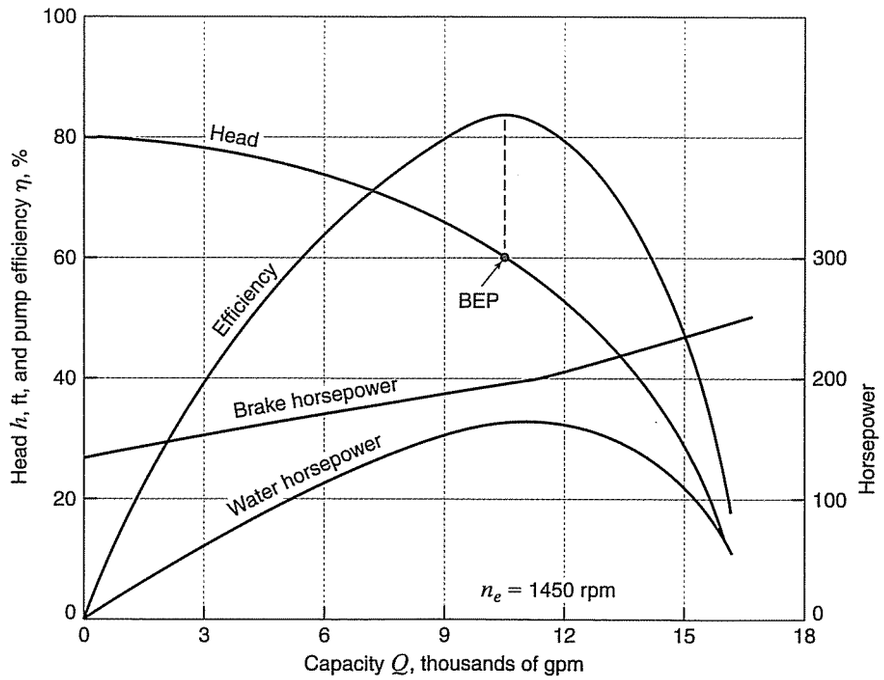


Figure 15.6
Characteristic curves for a typical mixed-flow centrifugal pump.

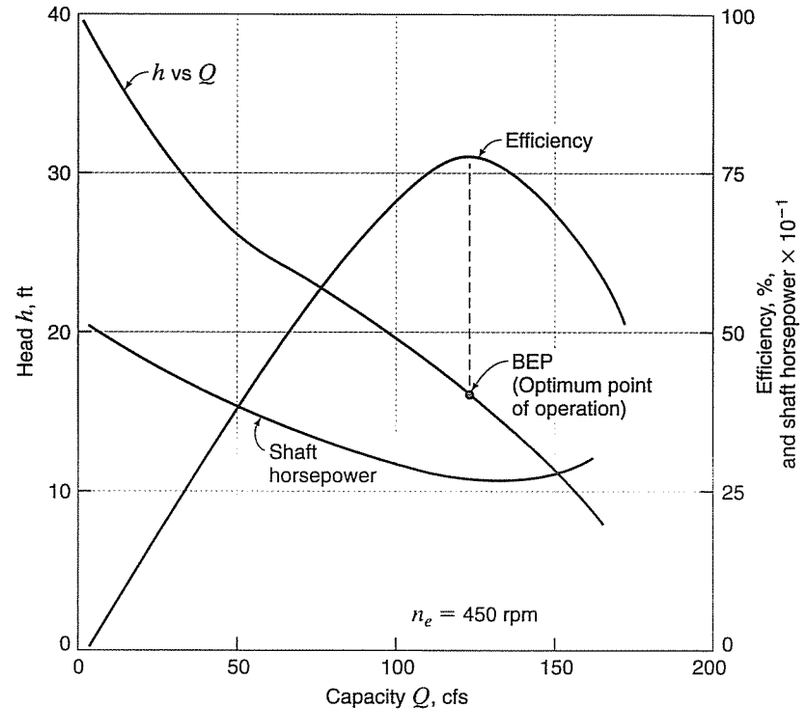


Figure 15.7
Characteristic curves for a typical axial-flow pump.

capacity or ***rated capacity*** of 10,500 gpm when developing a normal head of 60 ft at an operating speed of 1450 rpm. What we refer to as the "normal" or "rated" capacity corresponds to the ***point of optimum efficiency*** or ***BEP*** (best efficiency point); we shall denote rotative speeds at the BEP by a subscript *e*. Similar curves for a typical axial-flow pump are shown in Fig. 15.7. Curves such as those shown in Figs. 15.6 and 15.7 are usually determined by pump manufacturers through laboratory testing. By inspecting these two figures we see the remarkable difference in the characteristics of these two pumps. We observe that the efficiency of both pumps drops rather rapidly when the flow rate at which they are pumping exceeds the optimum. This is particularly true in the case of the axial-flow pump.

The shape of the impellers and vanes and their relationship to the pump casing cause variations in the intensity of shock loss, fluid friction, and turbulence. These vary with head and flow rate, and are responsible for the wide variation in pump characteristics. The ***shutoff head*** is that which is developed when there is no flow. In the case of the mixed-flow centrifugal pump (Fig. 15.6), the shutoff head is usually about 10% greater than the normal head, that which occurs at the point of optimum efficiency, while in the case of the axial-flow pump (Fig. 15.7), the shutoff head may be as much as three times the normal head.

15.6 PERFORMANCE CHARACTERISTICS AT DIFFERENT SPEEDS AND SIZES

The choice of a pump for a given situation will depend on the rotative speed of the motor used to drive the pump. If the characteristic curve for a pump at a given rotative speed is known, the relation between head and capacity at different rotative speeds can be derived approximately through use of Eqs. (15.4) and (15.5). For example, in Fig. 15.8, assume the characteristic curve (Curve 1) of a pump when operating at n_1 rpm is given. We can approximately derive the characteristic curves at different *speeds* of operation such as n_2 and n_3 by transferring points on Curve 1 to corresponding points on Curves 2 and 3, respectively. Thus, from Eqs. (15.4) and (15.5), we have for Curve 2, $Q_2 = Q_1(n_2/n_1)$ and $h_2 = h_1(n_2/n_1)^2$. Similar expressions can be used to develop Curve 3. Superimposed on Fig. 15.8 are lines of equal efficiency as determined by test. From this, we can see that efficiency drops off rather rapidly as one moves away from the BEP. We can also approximately derive characteristic curves for pumps of different *size,*

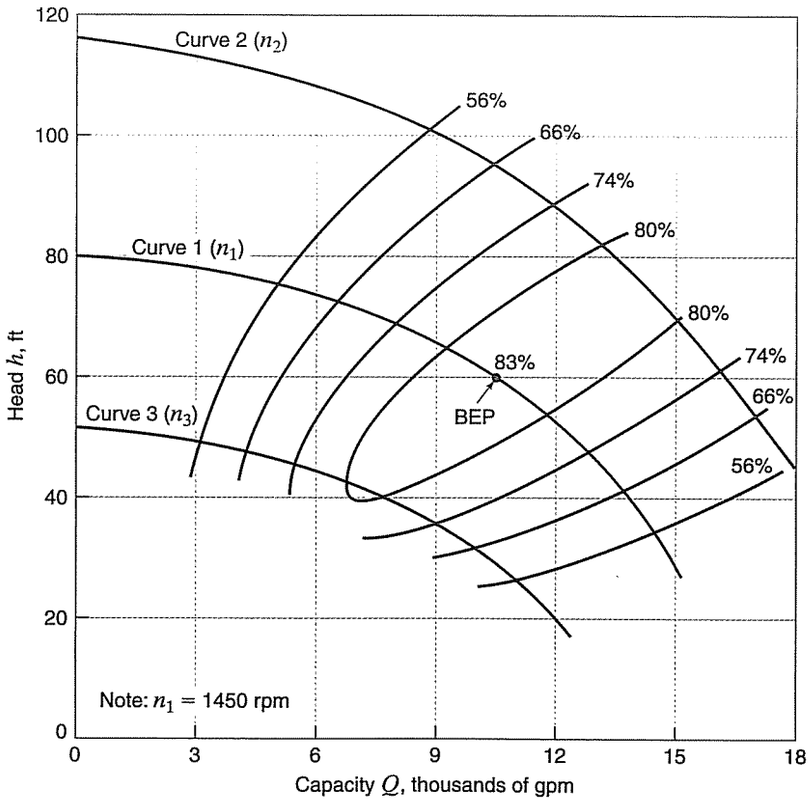


Figure 15.8
Characteristic performance curves of a typical mixed-flow centrifugal pump (Fig. 15.6) at various speeds of rotation with contours of equal efficiency.

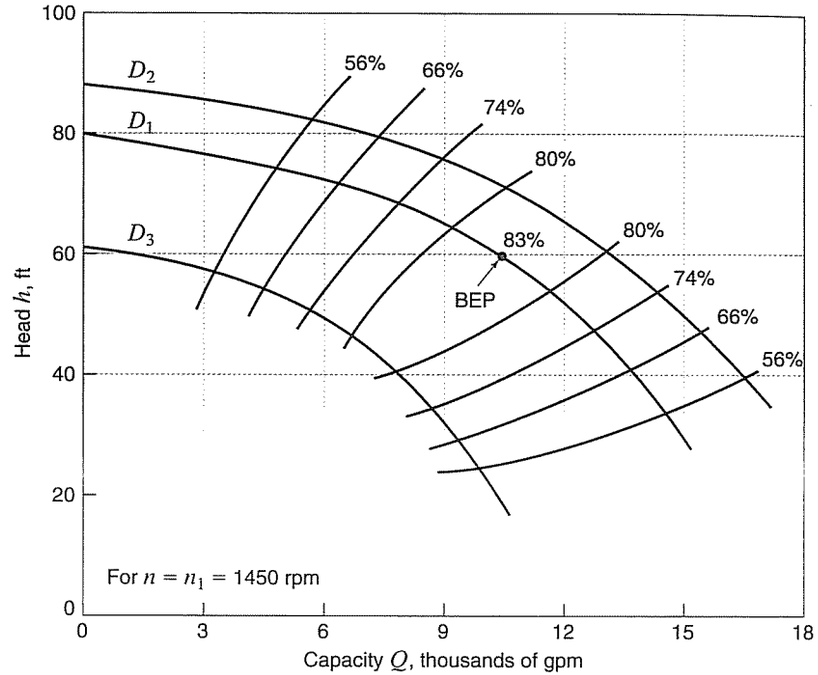


Figure 15.9
Characteristic performance curves with contours of equal efficiency for typical homologous mixed-flow centrifugal pumps having impellers of different size. D_1 is the same pump as in Fig. 15.6.

all operating at the same speed, using Eqs. (15.4) and (15.5). Such curves are shown in Fig. 15.9. Corresponding points were transferred through use of $Q_2 = Q_1(D_2/D_1)^3$ and $h_2 = h_1(D_2/D_1)^2$. Curves of equal efficiency have also been superimposed on Fig. 15.9.

We can approximately[6] develop characteristic curves for pumps having similar geometric shape, of both different *size* and operating at various constant *speeds,* using Eqs. (15.4) and (15.5), in which case $Q_2 = (n_2/n_1)(D_2/D_1)^3$ and $h_1 = (n_2/n_1)^2(D_2/D_1)^2$.

SAMPLE PROBLEM 15.3 In Fig. 15.8, $n_1 = 1450$ rpm. Estimate the rotative speeds n_2 and n_3.

Solution
Using values of h for $Q = 0$ gpm, we have $h_1 = 80$ ft. $h_2 \approx 117$ ft and $h_3 \approx 52$ ft.

Eq. (15.5): $h \propto n^2$

[6] The accuracy drops off with large variations in D and n.

Thus:

$$\frac{80}{117} = \left(\frac{1450}{n_2}\right)^2, \quad \text{from which} \quad n_2 \approx 1450\sqrt{\frac{117}{80}} = 1753 \approx 1750 \text{ rpm} \qquad \textit{ANS}$$

and

$$\frac{52}{80} = \left(\frac{n_3}{1450}\right)^2, \quad \text{from which} \quad n_3 \approx 1450\sqrt{\frac{52}{80}} = 1169 \approx 1170 \text{ rpm} \qquad \textit{ANS}$$

EXERCISES

15.6.1 Refer to Fig. 15.8. Transfer the point ($h = 70$ ft and $Q = 7500$ gpm) to a point on Curve 3 if $n_1 = 1450$ rpm and $n_3 = 1160$ rpm. Check your result with Fig. 15.8.

15.6.2 Refer to Fig. 15.9. Transfer the point ($h = 70$ ft and $Q = 7500$ gpm) to a point on the D_2 curve if $D_1 = 16.7$ in and $D_2 = 17.5$ in. Check your result with Fig. 15.9.

15.6.3 Plot h versus Q for the pump of Fig. 15.7 for the case where $n = 600$ rpm.

15.7 OPERATING POINT OF A PUMP

The manner in which a pump operates depends not only on the pump performance characteristics, but also on the characteristics of the system that it will operate in. In Fig. 15.10, for a particular pump under consideration, we show the pump operating characteristics (h versus Q) for a selected speed of operation, usually close to the speed that gives optimum efficiency. We also show the system

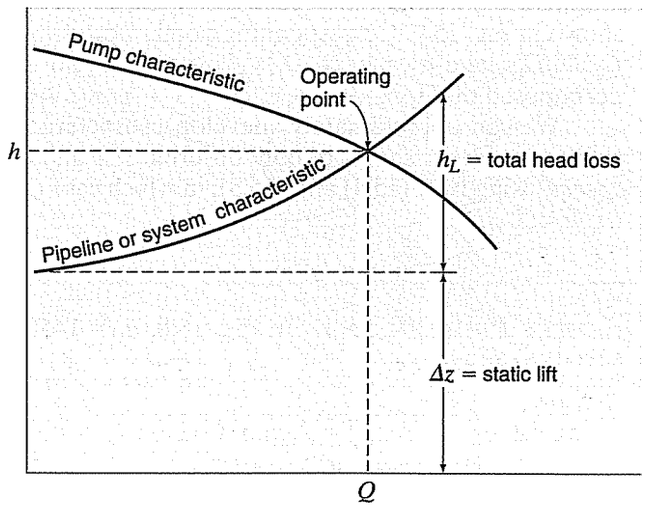


Figure 15.10
Graphical method for finding the operating point of a pump and pipeline.

characteristic curve (i.e., the required pumping head versus Q). In this case, the pump is delivering liquid through a piping system with a static lift of Δz. The head that the pump must develop is equal to the static lift plus the total head loss in the piping system (approximately proportional to Q^2). The actual ***pump-operating head*** and flow rate are determined by the intersection of the two curves.

The particular values of h and Q determined by this intersection may or may not be those for the maximum efficiency. If they are not, this means the pump is not exactly suited to the specific conditions. In Figs. 15.8 and 15.9 we see that efficiency drops off as one moves away from the BEP. Hence it is important to select a pump such that the pump performance and system characteristic curves intersect near the BEP. Changing the speed of the pump may help. Further discussion of the behavior of pumps and their relationship to the systems in which they operate is presented in Secs. 15.12 and 15.13.

SAMPLE PROBLEM 15.4 The pump whose characteristic curve is shown in Fig. 15.6 when operating at 1450 rpm is used to pump water from reservoir A to reservoir B through an 18-in-diameter pipe ($f = 0.032$) 500 ft long. Neglecting minor losses, find the flow rate for the following conditions: (a) reservoir water-surface elevations are identical; (b) water-surface elevation of reservoir B is 20 ft higher than that in reservoir A; (c) water-surface elevation of reservoir B is 65 ft higher than that in reservoir A. Assume f does not change with flow rate. Also find the efficiency by referring to Fig. 15.8.

Solution

The pump characteristic curve is given by Fig. 15.6. Note that this curve is the same as Curve 1 of Fig. 15.8.

Energy equation: $0 + h - h_L = \Delta z$

from which we find the equation of the system characteristic:

$$h = \Delta z + h_L = \Delta z + f\frac{L}{D}\frac{V^2}{2g}$$

$$= \Delta z + (0.032)\frac{500}{1.5}\frac{Q^2}{[\pi(1.5)^2/4]^2 2(32.2)}$$

$$= \Delta z + 0.0530Q^2 \qquad (Q \text{ expressed in cfs})$$

Coordinates of system curves:

Q, cfs	Q, gpm	(a) h_L, ft	(b) $20 + h_L$	(c) $65 + h_L$
0	0	0.0	20.0	65.0
10	4480	5.3	25.3	70.3
20	8970	21.2	41.2	86.2
30	13,450	47.8	67.8	(112.8)[7]

[7] For plotting purposes only. The pump cannot develop that much head when operating at 1450 rpm.

By plotting these system curves on Fig. 15.8 we find h and Q at the points of intersection with Curve 1. To find pump efficiency we note the location of the points of intersection and interpolate between contours of equal efficiency.

(*a*) $h \approx 45$ ft and $Q \approx 13{,}200$ gpm, $\eta \approx 71\%$
(*b*) $h \approx 55$ ft and $Q \approx 11{,}700$ gpm, $\eta \approx 81\%$
(*c*) $h \approx 73$ ft and $Q \approx 5800$ gpm, $\eta \approx 60\%$

ANS

EXERCISES

15.7.1 Repeat Sample Prob. 15.4 for the case where the length of the 18-in-diameter pipe is 1000 ft.

15.7.2 Repeat Sample Prob. 15.4 for the case where the pipe diameter is 12 in rather than 18 in, all other data remaining the same.

15.8 SPECIFIC SPEED OF PUMPS

Specific speed is a number that defines the type of pump (radial-flow, mixed-flow, or axial-flow). Traditionally in the United States, specific speed N_s of a pump has been expressed as

$$N_s = \left(\frac{n_e\sqrt{Q}}{h^{3/4}}\right)_{\text{BEP}} = \left(\frac{n_e\sqrt{\text{gpm}}}{h^{3/4}}\right)_{\text{BEP}} \qquad (15.8a)$$

where the values used for n_e (rpm), Q (gpm), and h (ft) are those that occur at the *point of optimum operating efficiency,* commonly referred to as the BEP (Sec. 15.5). Computed values of specific speed for a given pump throughout its entire operating range from zero discharge (shutoff head) to maximum discharge would give values from zero to some very large number. The only value that has any real significance is that corresponding to values of head, discharge, and speed at the BEP.

Equation (15.8*a*) is derived by eliminating D from Eqs. (15.4) and (15.5) in such a way that n becomes a term in the numerator of the resulting expression. It is left to the reader to work out this derivation (see Exer. 15.8.1). For large-capacity pumps, specific speed has sometimes been calculated using Q expressed in cfs rather than gpm, in which case $(N_s)_{\text{cfs}} = 0.0472(N_s)_{\text{gpm}}$.

In the SI, specific speed of pumps is defined by

$$(N_s)_{\text{SI}} = \left(\frac{\omega_e\sqrt{Q}}{(gh)^{3/4}}\right)_{\text{BEP}} \qquad (15.8b)$$

where the values of ω_e (rad/s), Q (m^3/s), and h (m) are those that occur at the BEP. The acceleration due to gravity g is 9.81 m/s^2. In this form, specific speed is dimensionless. However, rotative speed is expressed in radians per second, which is rather cumbersome. The relation between SI specific speed for pumps and the traditional mode of expressing specific speed is $(N_s)_{\text{SI}} = 0.000368(N_s)_{\text{gpm}}$.

Figure 15.11 shows several typical impellers in section and their corresponding specific speeds. Radial-flow impellers generally have specific speeds (Eq. 15.8*a*) between 500 and 5000, mixed-flow between 4000 and 10,000 and axial-flow from 9000 to 15,000. In the SI the corresponding ranges of specific speed are approximately 0.2–2.0 for radial-flow, 1.5–3.7 for mixed-flow, and

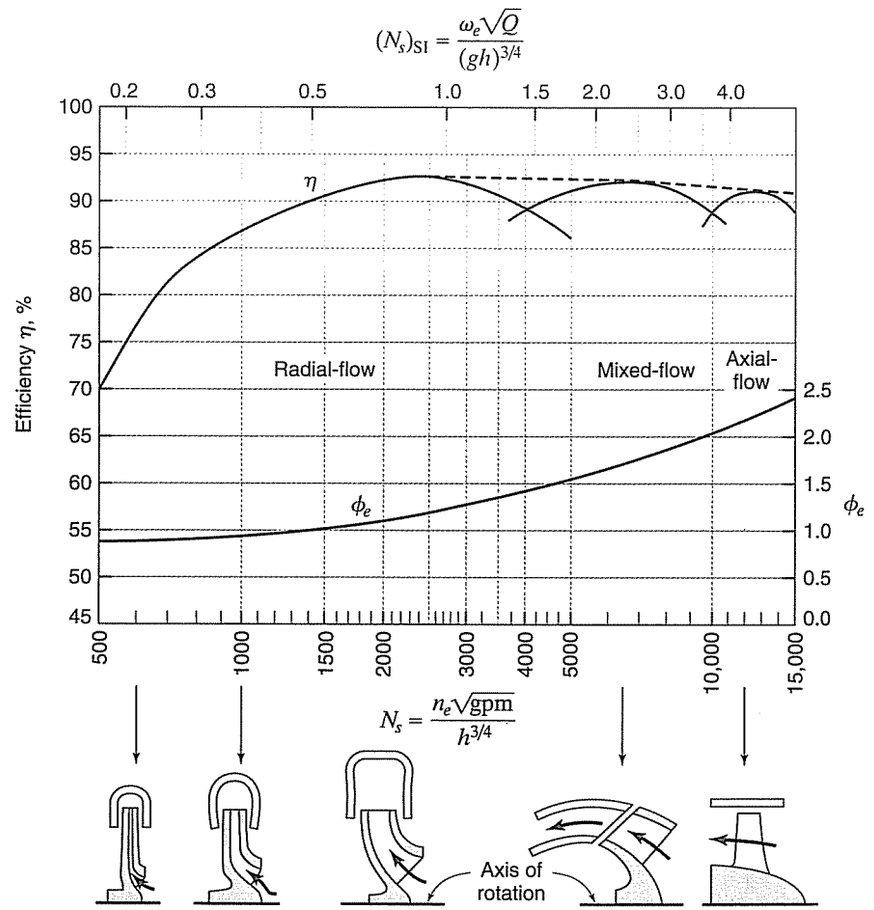


Figure 15.11
Optimum efficiency and typical values of ϕ_e for water pumps as a function of specific speed.

3.3–5.5 for axial-flow pumps. Two impellers having the same geometric shape have the same specific speed though their sizes may differ. The curves of Fig. 15.11 show the variation of peak efficiency with specific speed. Note that pumps with specific speed below 800 tend to be inefficient.

Equations (15.8) indicate that pumping against high heads requires a low-specific speed pump. We will discuss this further in Sec. 15.10. For very high heads and low discharges, the required specific speed may fall below the values for normal design and result in a pump with a low efficiency. To overcome this problem, we can distribute the head among a number of pumps in series, or we can use a multistage unit. The head per stage is generally limited to about 400 ft, although some pumps in use develop more than 600 ft of head per stage. A multistage pump is usually less expensive than a series of individual pumps, but the very high pressures developed in the multistage pump can offset this. We can avoid excessive pressures in the system by spacing several pumps more or less uniformly along a pipeline.

SAMPLE PROBLEM 15.5 What is the specific speed of the pump whose performance characteristics are given in Fig. 15.6? What type of pump is this?

Solution

At its BEP this pump has a capacity of 10,500 gpm while developing a head of 60 ft at a rotative speed of 1450 rpm.

Hence: $$N_s = \left(\frac{n_e\sqrt{\text{gpm}}}{h^{3/4}}\right)_{\text{BEP}} = \frac{1450\sqrt{10{,}500}}{60^{3/4}} = 6890 \qquad \textit{ANS}$$

From Fig. 15.11: This is a mixed-flow centrifugal pump. ***ANS***

EXERCISES

15.8.1 Derive the expression for the specific speed of a pump by eliminating D from Eqs. (15.4) and (15.5) in such a way that n becomes a term in the numerator.

15.8.2 What is the specific speed of the pump whose performance characteristics are given in Fig. 15.7? What type of pump is this?

15.8.3 What are the specific speeds of the pumps on the Colorado River Aqueduct at the Hayfield, Gene, and Iron Mountain Plants? (See Sec. 15.14.) Find also the corresponding specific speeds $(N_s)_{\text{cfs}}$.

15.8.4 Find the specific speed of a 10-stage pump that develops a total head of 600 ft at a capacity of 1600 gpm when operating at maximum efficiency at a rotative speed of 900 rpm.

15.8.5 A pump is to discharge 0.8 m³/s at a head of 40 m when running at 300 rpm. What type of pump will be required?

15.9 PERIPHERAL-VELOCITY FACTOR

For a pump impeller (Fig. 15.1) or a turbine runner (Fig. 16.7), the ratio of the peripheral velocity to $\sqrt{2gh}$ is referred to as the ***peripheral-velocity factor,*** denoted by ϕ (phi). Thus, for a pump,

$$u_2 = \phi\sqrt{2gh} \tag{15.9}$$

where u_2 is the peripheral speed of the impeller. For an axial-flow pump it is the vane-tip speed that is used in Eq. (15.9).

For any machine its peripheral velocity might be any value from zero up to some maximum under a given head, depending on the operating speed, and ϕ would consequently vary through a wide range. But the speed that is of most practical significance is that at which the efficiency is a maximum. We shall denote the value of ϕ at the speed of maximum efficiency by ϕ_e.

At maximum efficiency,

$$u_2 = \frac{2\pi r n_e}{60} = \frac{\pi D n_e}{60} = \phi_e\sqrt{2gh}$$

Thus, $$D = \frac{60\sqrt{2g}\phi_e\sqrt{h}}{n_e}$$

which in BG units reduces to

$$D = \frac{153.3\phi_e\sqrt{h}}{n_e} \tag{15.10}$$

Typical values of ϕ_e for various types of pumps as a function of specific speed are given in Fig. 15.11. We can use this curve, together with Eq. (15.10), to estimate the diameter of the impeller of a pump if the specific speed of the pump is known.

Because many exercises and problems on this and subsequent sections depend on values read from graphs, numerical answers will probably vary somewhat.

SAMPLE PROBLEM 15.6 Find the value of ϕ_e for the Eagle Mountain pumps mentioned in Sec. 15.14. How does this computed value of ϕ_e compare with the value given in Fig. 15.11?

Solution

Eq. (15.10): $$D = 153.3\phi_e\sqrt{h}/n_e$$

$$(81.6/12) = 153.3\phi_e\sqrt{440}/450$$

From which: $$\phi_e = 0.95 \qquad \textit{ANS}$$

$$\text{Specific speed } N_s = \frac{450\sqrt{200(448.8)}}{440^{3/4}} = 1404$$

From Fig. 15.11, $\phi_e \approx 1.0$, which is a close check. ***ANS***

SAMPLE PROBLEM 15.7 Estimate the diameter of the impeller of the pump whose operating characteristic is shown in Fig. 15.6. See Sample Prob. 15.5.

Solution

From Sample Prob. 15.5: Specific speed $N_s = 6890$

From Fig. 15.11 for $N_s = 6890$: $\phi_e = 1.7$

Eq. (15.10): $D = 155.3\phi_e\sqrt{h}/n$

$D = 153.3(1.7)\sqrt{60}/1450$

$D = 1.40 \text{ ft} = 16.70 \text{ in}$ ***ANS***

EXERCISES

15.9.1 Estimate the diameter of the impeller of the pump whose operating characteristics are shown in Fig. 15.7.

15.9.2 Estimate the diameter of the impeller of a pump whose BEP is defined by $Q = 5000$ gpm at a head of 82 ft when operating at 3000 rpm.

15.9.3 At maximum efficiency, a four-stage pump delivers 400 L/s against a head of 300 m at a rotative speed ω of 125.6 rad/s. Estimate the diameter of the impellers if all four impellers are identical. Calculate $(N_s)_{SI}$, convert to N_s, and then obtain an estimated value of D.

15.10 CAVITATION IN PUMPS

An important factor in the satisfactory operation of a pump is the avoidance of cavitation (Sec. 5.10), both for the sake of good efficiency and for the prevention of impeller damage. As liquid passes through the impeller of a pump, there is a change in pressure. If the absolute pressure of the liquid drops to the vapor pressure, cavitation will occur. The region of vaporization hinders the flow and places a limit on the capacity of the pump. As the fluid moves further into a region of higher pressure, the bubbles collapse and the implosion of the bubbles may cause pitting of the impeller.[8] Cavitation is most likely to occur near the point of discharge (periphery) of radial-flow and mixed-flow impellers, where the velocities are highest. It may also occur on the suction side of the impeller, where the pressures are the lowest. In the case of an axial-flow pump, the blade-tip is the most vulnerable to cavitation.

[8] The pitting of a turbine runner is shown in Fig. 16.15. Similar phenomena occur in pump impellers if cavitation takes place.

For pumps, a ***cavitation parameter*** σ (sigma) is defined as

$$\sigma = \frac{(p_s)_{abs}/\gamma + V_s^2/2g - p_v/\gamma}{h} = \frac{\text{NPSH}}{h} \tag{15.11}$$

where subscript s refers to values at the pump intake (i.e., suction side of the pump), h is the head developed by the pump, and p_v is the vapor pressure. As the latter is normally given in absolute units, it follows that p_s must also be absolute pressure. NPSH, the numerator of Eq. (15.11), is referred to as the ***net positive suction head.***

There is a critical value of σ, called the ***critical cavitation parameter*** and denoted by σ_c, below which there is a drop in pump efficiency indicative of the onset of cavitation. The value of σ_c depends on the type of pump and the conditions of operation. For a given pump, the value of σ_c varies with flow rate. Approximate values of σ_c for centrifugal pumps operating under normal conditions near optimum efficiency are shown in Fig. 15.12. In important installations the

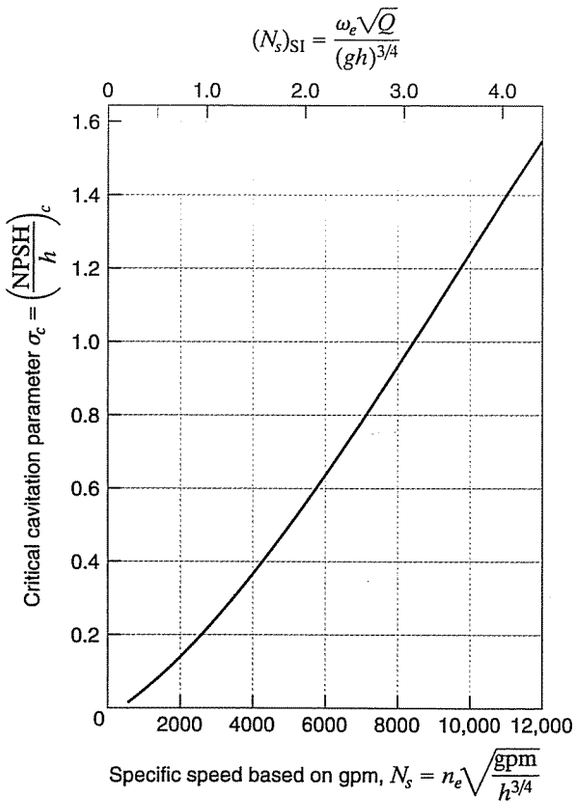


Figure 15.12
Approximate values of critical cavitation parameter σ_c as a function of specific speed.

value of σ_c should be determined experimentally in a model study. For modeling purposes, a useful parameter, the ***suction specific speed*** S, is defined as

$$S = \frac{n\sqrt{\text{gpm}}}{\text{NPSH}^{3/4}} \tag{15.12}$$

If model and prototype are operated at identical values of N_s and S, similarity of flow and cavitation is achieved provided the model and prototype are geometrically similar to one another.

To prevent the occurrence of cavitation, a pump must operate in such a manner that σ is larger than σ_c. We can accomplish this by selecting the proper type and size of pump and speed of operation and by setting the pump at the proper point and elevation in the system. Through inspection of Eq. (15.11), we can see that small values of σ result from large values of h, the head developed by the pump. Hence, for any given situation, there is a limiting value of h above which σ will be less than σ_c, resulting in cavitation.

By writing an energy equation for the situation depicted in Fig. 15.5 in terms of absolute pressure from the surface of the source reservoir to the suction side of the pump, we obtain

$$\frac{(p_0)_{\text{abs}}}{\gamma} - h_L = z_s + \frac{(p_s)_{\text{abs}}}{\gamma} + \frac{V_s^2}{2g}$$

or

$$\frac{(p_s)_{\text{abs}}}{\gamma} + \frac{V_s^2}{2g} = \frac{(p_0)_{\text{abs}}}{\gamma} - h_L - z_s$$

where $(p_0)_{\text{abs}}/\gamma$ is the absolute pressure on the surface of the source reservoir. If the liquid is drawn from a closed reservoir, $(p_0)_{\text{abs}}/\gamma$ could be either greater or less than the atmospheric pressure. In the usual case the reservoir is open to the atmosphere, and $(p_0)_{\text{abs}}/\gamma = p_{\text{atm}}/\gamma$.

Substituting the right side of the preceding expression into Eq. (15.11), we get

$$\sigma = \frac{\dfrac{(p_0)_{\text{abs}}}{\gamma} - h_L - z_s - \dfrac{p_v}{\gamma}}{h} \tag{15.13}$$

This expression indicates that σ will tend to be small (hence there will be a tendency toward cavitation) in the following situations: (*a*) high head; (*b*) low atmospheric pressure, i.e., high elevation; (*c*) large head loss between source reservoir and the pump; (*d*) large value of z_s, i.e., with the pump at a relatively high elevation compared with the elevation of the reservoir water surface; and (*e*) large value of vapor pressure, i.e., high temperature and/or a very volatile liquid such as gasoline being pumped. With (*a*), we can limit the required head by using multistage pumps. For a given liquid to be pumped at a certain elevation and temperature, we have no control over items (*b*) and (*e*). However, we can do something about (*c*) and (*d*). When we design the layout of a pumping system, we can reduce the tendency toward cavitation by minimizing h_L [item (*c*)]

by placing the pump close to the source reservoir, and by setting the pump at a low elevation [item (*e*)] relative to the reservoir water surface.

In Eq. (15.13), when σ equals the critical value σ_c then $z_s = (z_s)_{\max}$, the highest elevation at which we can safely set a pump and guard against cavitation. Rearranging, we get

$$(z_s)_{\max} = \frac{(p_0)_{\text{abs}}}{\gamma} - \frac{p_v}{\gamma} - \sigma_c h - h_L \tag{15.14}$$

As long as z_s is less than $(z_s)_{\max}$, there should be no problem with cavitation, assuming, of course, that an accurate value of σ_c is known for the given conditions of operation. Pump manufacturers often run tests on pumps to determine values of σ_c for different conditions of operation.

Based on experience with pumps, in Fig. 15.13 ***limiting heads*** are shown for the prevention of cavitation in single-stage, single-suction pumps as a function of specific speed and ***suction lift,*** the elevation difference between the energy line at suction and the center of the impeller as indicated in Fig. 15.5. A positive suction lift indicates that the impeller is above the energy line. The curves of Fig. 15.13 are applicable to water at 80°F under sea-level atmospheric pressure. These curves can readily be shifted for other conditions: different temperatures, different elevations, and different liquids.

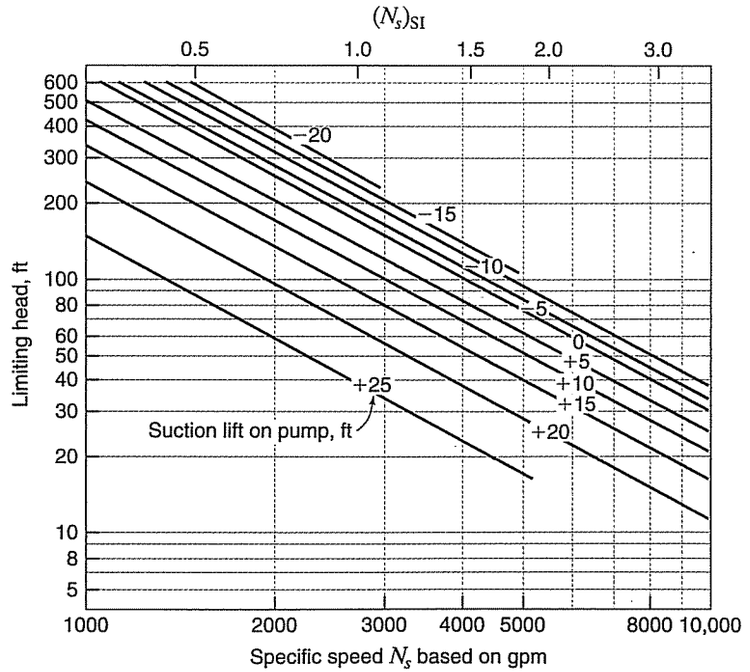


Figure 15.13
Recommended limiting heads for single-stage, single-suction pumps as a function of specific speed and suction lift, at sea level with water temperature of 80°F.

SAMPLE PROBLEM 15.8 A pump with a critical value of σ of 0.10 is to pump against a head of 500 ft. The barometric pressure is 14.3 psia, and the vapor pressure is 0.5 psia. Assume friction losses in the intake piping are 5 ft. Find the maximum allowable elevation of the pump relative to the water surface at intake.

Solution

$$p_{atm}/\gamma = (14.3)144/62.4 = 33.0 \text{ ft}$$

$$p_v/\gamma = (0.5)144/62.4 = 1.154 \text{ ft}$$

Eq. (15.14): $(z_s)_{max} = 33.0 - 1.154 - 0.10(500) - 5.0 = -23.2$ ft

The pump should be placed at least 23.2 ft below the reservoir water surface. ***ANS***

SAMPLE PROBLEM 15.9 A pump is delivering 7500 gpm at 140°F against a head of 240 ft when the barometric pressure is 13.8 psia. Determine the reading on a pressure gage in inches of mercury vacuum at the suction flange when cavitation is incipient if $\sigma_c = 0.085$. The suction pipe has a diameter of 2.0 ft.

Solution

Table A.1 at 140°F: $\gamma = 61.38$ pcf, $p_v/\gamma = 6.67$ ft

$$V_s = Q/A_s = (7500/449)/(\pi 2^2/4) = 5.32 \text{ ft/sec}$$

Let p = gage pressure at suction flange: $(p_s)_{abs} = p_{at} + p = (13.8 + p)$ psia

Eq. (15.11): $0.085 = [(13.8 + p)(144/61.38) + 5.32^2/2(32.2) - 6.67]/240$

$$p = -2.45 \text{ psi}$$

$$-2.45(29.9 \text{ inHg}/14.7 \text{ psia}) = 4.98 \text{ inHg vacuum} \quad \textit{ANS}$$

EXERCISES

15.10.1 A mixed-flow pump, located at sea level, with a specific speed of 6000 is to be used to pump 80°F water from a reservoir. The head to be developed is 40 ft. What is the greatest elevation above the reservoir water surface that the pump can be placed such that cavitation will not occur in the pump? Assume the head loss from the reservoir to the pump is 1.5 ft.

15.10.2 Solve Exer. 15.10.1 for the case where the pump is located at elevation 10,000 ft with a water temperature of 50°F. Assume all other data remain the same.

15.10.3 A pump with a critical value of $\sigma = 0.20$ is to pump against a head of 20 m. The barometric pressure is 98.5 kPa abs, and the vapor pressure is 5.2 kPa abs. Friction losses from the reservoir to the pump are 0.5 m. Find the maximum allowable height of the pump relative to the water surface in the reservoir to ensure that cavitation will not occur.

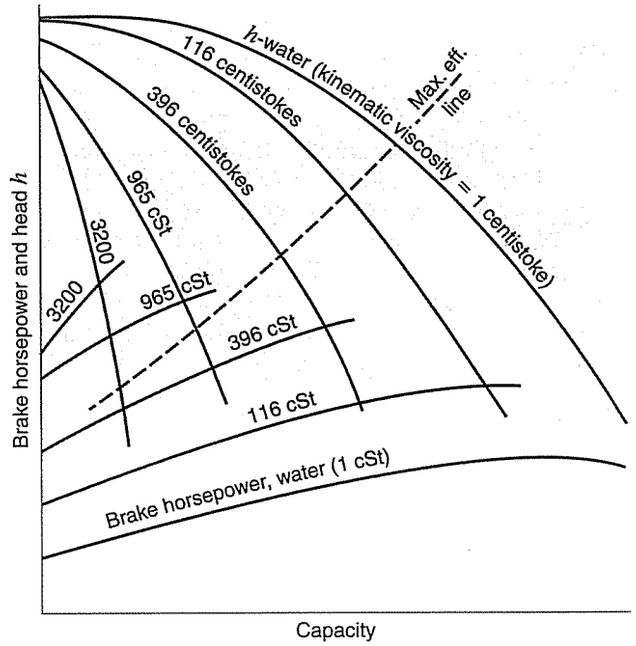


Figure 15.14
Centrifugal pump with viscous oils. (1 centistoke = 10^{-6} m^2/s.)

15.11 VISCOSITY EFFECT

Centrifugal pumps are also used to pump liquids with viscosities different from that of water. Figure 15.14 shows actual test curves of performance for a very extreme range of viscosities, from water to an oil with a kinematic viscosity 3200 times that of water. We see that, as the viscosity increases, the head-capacity curve becomes steeper and the power required increases. The dashed line indicates the points of maximum efficiency for each curve. We see that both the head and capacity at the point of maximum efficiency decrease with increasing viscosity. As these are accompanied by an increase in brake horsepower, there is a marked decrease in efficiency. For example, for the situation depicted in Fig. 15.14, if the optimum efficiency of the pump when pumping water is 0.85, its optimum efficiency is only 0.47 when pumping a liquid whose viscosity is 116 times that of water and only 0.18 when pumping a liquid whose viscosity is 396 times that of water.

15.12 SELECTION OF PUMPS

In selecting a pump for a given situation, we have a variety of pumps to choose among. We strive to select a pump that will operate at a relatively high efficiency for the given conditions of operation. Manufacturers provide pump performance information such as that in Figs. 15.6 and 15.7. The engineer's task is to

select the pump or pumps that best fits in with the system characteristics. Alternatives to be investigated include specific speed N_s, size D of the impeller, and speed n of operation. Other alternatives include the use of multistage pumps, pumps in series, pumps in parallel, etc. Even, under certain conditions, throttling the flow in the system can result in savings in energy.

In all cases the possibility of cavitation must be carefully investigated. Cavitation should be avoided at all costs. Problems with cavitation can be eliminated by limiting the head that the pump must develop, by selecting the proper type of pump, and by setting the pump at a low enough elevation.

The objective is to select a pump and its speed of operation such that the pump's performance characteristics relate to the system in which it operates in such a fashion that the operating point (Sec. 15.7) is close to the BEP (best operating point). This tends to optimize the pump efficiency, resulting in a minimization of energy expenditure.

The operating point can be shifted by changing the pump characteristic curve, by changing the system characteristic curve, or by changing both curves. The pump curve can be changed by changing the speed of operation of a given pump or by selecting a different pump with different performance characteristics (Fig. 15.15*a*). In some instances it is helpful to trim the impeller, i.e., reduce its diameter somewhat, perhaps 5% or so, by grinding it down (Sec. 15.4). The smaller impeller is installed in the original casing. The system characteristic curve can be changed by changing the pipe size or by throttling the flow (Fig. 15.15*b*).

A complication that often occurs is that the energy levels at the two ends of the system do not remain constant, such as that created by fluctuating reservoir

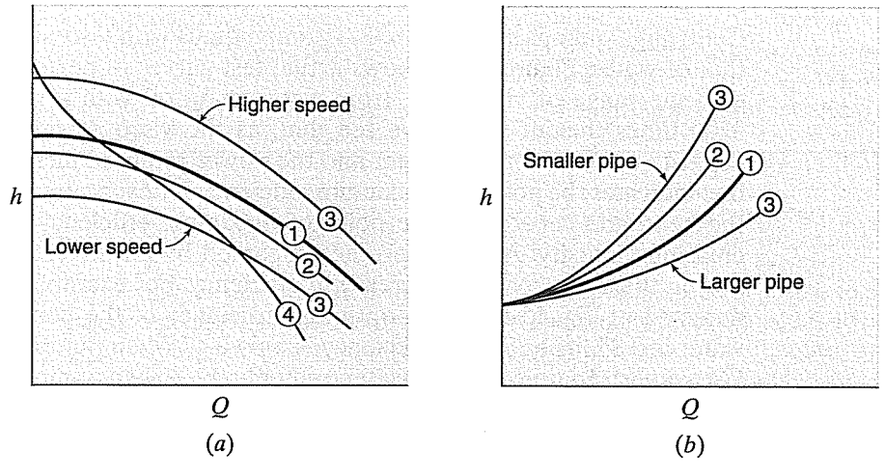


Figure 15.15
Changing the operating point by (*a*) changing the pump characteristic curve or (*b*) changing the system characteristic curve.

levels. In such a case it is difficult to achieve a high efficiency for all modes of operation. In extreme cases a variable-speed motor is sometimes used.

SAMPLE PROBLEM 15.10 Water at 50°F is to be pumped between reservoirs A and B whose water surfaces are at elevations 5910 ft and 6060 ft, respectively. The water is to be delivered at 20 cfs through a 24-in-diameter pipe of length 2000 ft. Assume $f = 0.03$. (*a*) Find the specific speed of the most suitable pump if the speed of operation is to be 600 rpm. (*b*) If this pump is installed at a point 300 ft from reservoir A at the same elevation as the water surface in reservoir A, will it be safe against cavitation?

Solution

(*a*)
$$V = Q/A = 20/\pi(1)^2 = 6.37 \text{ ft/sec}$$

$$h = (6060 - 5910) + 0.03\frac{2000}{2}\frac{(6.37)^2}{2(32.2)} = 168.9 \text{ ft}$$

Eq. (15.8*a*): $$N_s = \frac{n_e\sqrt{\text{gpm}}}{h^{3/4}} = \frac{600\sqrt{20 \times 449}}{(168.9)^{3/4}} = 1214 \qquad \textit{ANS}$$

(*b*) From Table A.3 at mean elevation 5985 ft, by interpolation:

$$\frac{(p_0)_{\text{abs}}}{\gamma} = \frac{11.81(144)}{62.4} = 27.25 \text{ ft}$$

Table A.1 at 50°F: $$\frac{p_v}{\gamma} = 0.41 \text{ ft}$$

Fig. 15.12: $$\sigma_c \approx 0.08, \quad \sigma_c h \approx 0.08(168.9) \approx 13.5 \text{ ft}$$

Eq. (8.13): $$h_L \approx h_f = f\frac{L}{D}\frac{V^2}{2g} = 0.03\frac{300}{2}\frac{(6.37)^2}{2(32.2)} = 2.83 \text{ ft}$$

Eq. (15.14): $$(z_s)_{\max} = \frac{(p_0)_{\text{abs}}}{\gamma} - \frac{p_v}{\gamma} - \sigma_c h - h_L$$

$$(z_s)_{\max} \approx 27.25 - 0.41 - 13.5 - 2.83 = 10.5 \text{ ft}$$

Yes, the pump will be safe from cavitation. It can be placed as much as 10.5 ft above the water-surface elevation in reservoir A. ***ANS***

SAMPLE PROBLEM 15.11 Repeat Sample Prob. 15.10 for a pump where the rotative speed is 1200 rpm rather than 600 rpm, with all other data remaining the same.

Solution

Eq. (15.8*a*): $N_s \propto n_e$; hence $N_s = (1200/600)1214 = 2428$

From Fig. 15.12, when $N_s = 2428$: $\sigma_c \approx 0.18$

Thus: $\sigma_c h \approx 0.18(168.9) \approx 30.4$ ft

And $(z_s)_{max} = 27.3 - 0.4 - 30.4 - 2.8 \approx -6.4$ ft

To prevent cavitation, the pump must be placed 6.4 ft or more below the water-surface elevation in reservoir A. ***ANS***

Notes: By comparing the results of Sample Probs. 15.10 and 15.11, we find that rotative speed can greatly influence the specific speed of the pump that should be selected at a given installation. We also observe that an increase in operating speed results in a pump with a higher specific speed that is more vulnerable to cavitation.

SAMPLE PROBLEM 15.12 Find the approximate diameters of the pumps of Sample Probs. 15.10 and 15.11.

Solution

Eq. (15.10): $$D = \frac{153.3\phi_e\sqrt{h}}{n_e}$$

Fig. 15.11: $N_s = 1214$, $\phi_e \approx 0.9$, and $N_s = 2428$, $\phi_e \approx 1.2$

for $N_s = 1214$: $$D = \frac{153.3(0.9)\sqrt{168.9}}{600} = 2.99 \text{ ft} = 35.9 \text{ in}$$ ***ANS***

for $N_s = 2428$: $$D = \frac{153.3(1.2)\sqrt{168.9}}{1200} = 1.99 \text{ ft} = 23.9 \text{ in}$$ ***ANS***

Note: The faster speed permits us to employ a smaller pump, but it is more vulnerable to cavitation (i.e., it must be set at a lower elevation).

EXERCISES

15.12.1 Repeat Sample Prob. 15.10 for the case where the reservoir water-surface elevations are 120 ft and 160 ft, respectively, with all other data remaining the same.

15.12.2 Repeat Sample Prob. 15.10 for the case where the flow rate is 50 cfs rather than 20 cfs, with all other data remaining the same.

15.12.3 Water flows at 80°C through a 100-mm-diameter pipe ($f = 0.024$) of length 300 m from a large pressurized tank ($p = 200$ kPa, water-surface elevation 60.0 m) to discharge into the atmosphere at elevation 68.0 m. If the flow rate is to be 20 L/s, find the head that a pump, rotating at 1200 rpm placed in the pipeline, must develop, and determine its specific speed. Assume the pressure in the tank remains constant and neglect minor losses. This pump is placed 6.0 m above the water level in the tank. Will cavitation occur if the head loss to the pump is 0.8 m?

15.13 PUMPS OPERATING IN SERIES AND IN PARALLEL

In Fig. 15.16*a*, *b*, and *c* we summarize our discussion thus far by showing: (*a*) the general shape of the characteristic curves for the three different types of pumps, (*b*) the effect on performance of changing the rotative speed of a given pump, and (*c*) the effect on performance of different sizes of pump impellers. Figures 15.16*d* and 15.16*e* show the performance curves for two identical pumps in series and for two identical pumps in parallel. By examining Figs. 15.16*d* and 15.16*e*, we see that pumps in series tend to increase head, while pumps in parallel tend to increase flow rate.

When pumps are installed in series or parallel, it is very important that they have reasonably similar, or better yet identical, head-capacity characteristics throughout their range of operation; otherwise, one pump will carry most of the load and, under certain conditions, all of the load, with the other pump acting as a hindrance rather than a help. In fact, in parallel, if the performance characteristics of the pumps are quite different, a condition of backflow can occur in one of the pumps. Finally, one must always be sure that the selected pump (or pumps) will not encounter cavitation problems over the full range of operating conditions.

The mode of operation for any pumping system is best determined by plotting the pumping characteristics and the pipe system characteristics on the same diagram (Fig. 15.10). The point at which the two curves intersect indicates what will take place. Several aspects of the relationship between the pump and system characteristics are demonstrated in the following sample problem.

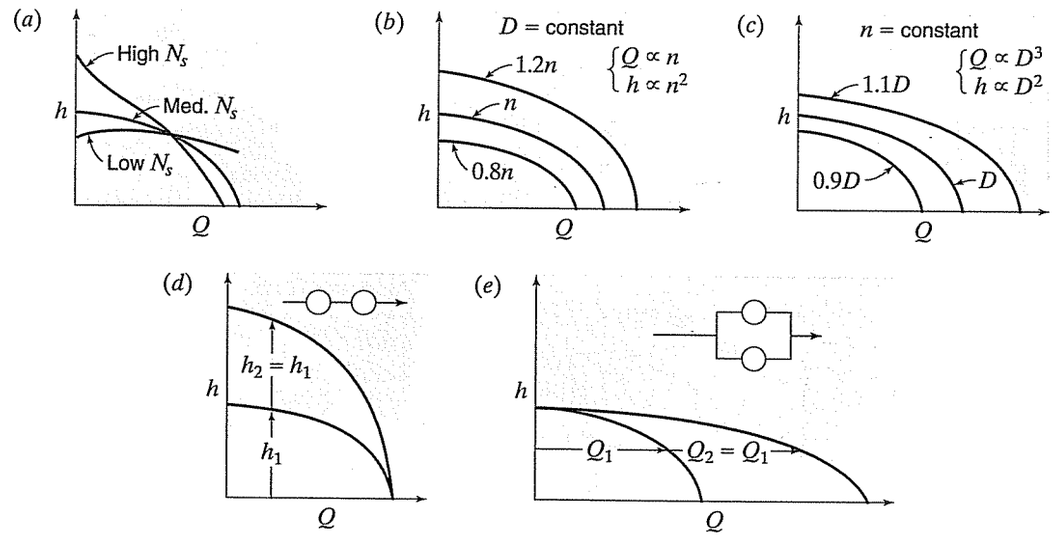


Figure 15.16
Pumping alternatives. (*a*) Different pumps with different characteristics. (*b*) A particular pump at different speeds. (*c*) Homologous pumps of different size. (*d*) Two identical pumps in series. (*e*) Two identical pumps in parallel. (*Note:* In series or parallel the pumps need not be identical, but their performance characteristics should be close to one another.)

SAMPLE PROBLEM 15.13 Two reservoirs A and B are connected with a long pipe that has characteristics such that the head loss through the pipe is expressible as $h_L = 20Q^2$, where h_L is in feet and Q is the flow rate in 100s of gpm. The water-surface elevation in reservoir B is 35 ft above that in reservoir A. Two identical pumps are available for use to pump the water from A to B. The characteristic curve of each pump when operating at 1800 rpm is given in the following table.

Operation at 1800 rpm	
Head, ft	Flow rate, gpm
100	0
90	110
80	180
60	250
40	300
20	340

At the optimum point of operation, the pump delivers 200 gpm at a head of 75 ft. Determine the specific speed N_s of the pump and find the rate of flow under the following conditions: (*a*) A single pump operating at 1800 rpm; (*b*) two pumps in series, each operating at 1800 rpm; (*c*) two pumps in parallel, each operating at 1800 rpm.

Solution

Eq. (15.7): $$N_s = \frac{n_e\sqrt{\text{gpm}}}{h^{3/4}} = \frac{1800(200)^{1/2}}{75^{3/4}} = 999 \qquad \textit{ANS}$$

We plot the head-capacity curves for the pumping alternatives (Fig. S15.13), and the h versus Q curve for the pipe system. In this case $h = \Delta z + h_L = 35 + 20Q^2$.

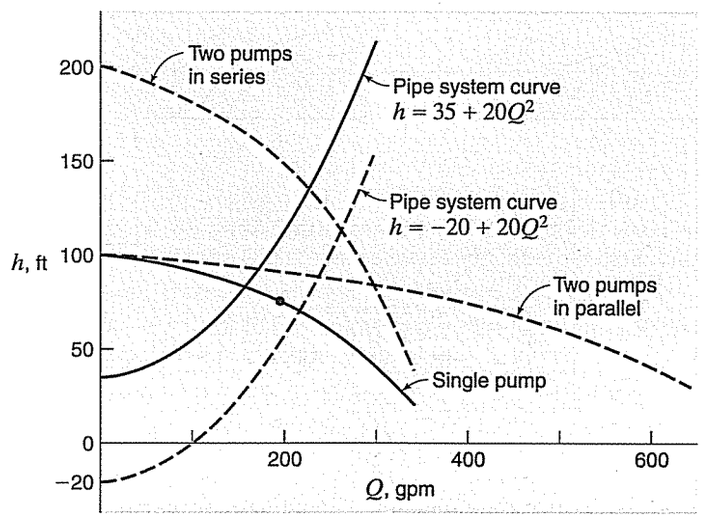


Figure S15.13

The answers are found at the points of intersection of the curves. They are as follows: (*a*) single pump, 156 gpm; (*b*) two pumps in series, 224 gpm; (*c*) two pumps in parallel, 170 gmp. ***ANS***

If Δz had been greater than 100 ft, neither the single pump nor the two pumps in parallel would have delivered any water. If Δz had been −20 ft (i.e., with the water-surface elevation in reservoir B 20 ft below that in A, the flows would have been: (*a*) 212 gpm; (*b*) 258 gpm; (*c*) 232 gpm.

EXERCISES

15.13.1 Suppose the pumps of Sample Prob. 15.13 operate at 1500 rpm. What then would be the flow rates for (*a*) a single pump; (*b*) two pumps in series; (*c*) two pumps in parallel. All other data to remain the same.

15.13.2 Two pumps whose characteristics are given in Sample Prob. 15.13 are to be used in parallel. They must develop a head $h = 35 + 20Q^2$ as in the sample problem. (*a*) What is the flow rate with both pumps operating at 1800 rpm? (*b*) The speed of one of the pumps is reduced until it no longer delivers water. At approximately what speed will this happen?

15.14 PUMP INSTALLATIONS

A few examples of pump installations are presented here as illustrations of modern practice.

The Byron Jackson Company has built pumps with as many as 54 stages. Water has been lifted to heights of several thousand feet by multistage pumps. Ingersoll-Rand produced a 6-in 10-stage pump operating at 3750 rpm that delivers 1600 gpm (101 L/s) at a head of 6000 ft (1830 m), the shutoff head being 7000 ft (2130 m). The Worthington Corporation installed a pump at Rocky River to deliver 279.5 cfs (7.915 m^3/s) at a head of 238.84 ft (72.798 m) when running at 327 rpm. The brake horsepower was 8259 (6.16 MW), giving an efficiency of 91.7%. The impeller diameter was approximately 7.54 ft (2.30 m), and the width at outlet approximately 0.72 ft (219 mm), with an ***eye*** (inlet) diameter of 4.24 ft (1.292 m).

On the Colorado River Aqueduct, which delivers water from the Colorado River to the metropolitan Los Angeles area, the Worthington Corporation built three pumps for the Hayfield plant to deliver 200 cfs (5.66 m^3/s) each at a head of 444 ft (135 m) when running at 450 rpm, and three similar pumps for Eagle Mountain with a head of 440 ft (134 m). The impeller diameters are approximately 81.6 in (2.07 m), and the eye diameters 34 in (864 mm). The Byron Jackson Company built three pumps for the Gene plant to deliver 200 cfs (5.66 m^3/s) each at a head of 310 ft (94.5 m) when running at 400 rpm. The impeller diameters are 78 in (1.98 m). At the Intake plant, where the head is 294 ft (89.6 m), the impeller diameters are 76 in (1.93 m). The Allis-Chalmers

Company built three pumps for the Iron Mountain plant to deliver 200 cfs (5.66 m^3/s) each at a head of 146 ft (44.5 m) when running at 300 rpm.

A typical moderate-sized large-capacity pumping plant is that at Cartersville, Georgia where the Johnston Pump Company installed a two-stage mixed-flow vertical-shaft pump that delivers 95,000 gpm (6.0 m^3/s) against a head of 95 ft (29 m) when operating at 394 rpm. At Marineland of the Pacific in Palos Verdes, California, three-stage Johnston vertical turbine pumps are used to pump salt water to tanks housing marine life. Because of the corrosive action of the seawater, the pump bowls are of iron with vitreous enamel coating. All moving parts that come in contact with the seawater are constructed of corrosion-resistant material.

A noteworthy pumping project is that at Grand Coulee on the Columbia River, where pumps with an impeller diameter of $167\frac{3}{8}$ in (4.251 m) operating at their BEP deliver 1250 cfs (35.4 m^3/s) against a head of 344 ft (105 m) when operating at 200 rpm. The maximum efficiency is 90.8%. The shutoff head is 422 ft (129 m). These pumps when operating at a head of 270 ft (82.3 m) deliver 1650 cfs (46.7 m^3/s) at an efficiency of 84%.

The Allis-Chalmers Company has built a combination reversible pump-turbine for the Hiwasee plant of the Tennessee Valley Authority, which as a pump will deliver 3900 cfs (110 m^3/s) at a head of 205 ft (62.5 m) at the point of maximum efficiency while requiring approximately 100,000 bhp (75 MW). The impeller diameter is 266 in (6.76 m), and it runs at 105.9 rpm. The maximum capacity is 5200 cfs (147 m^3/s) at 135 ft (41.1 m) head.

In Italy is a pump built in Switzerland that discharges 250,000 gpm, or 558 cfs (15.8 m^3/s), at a head of 787 ft (240 m) at 450 rpm. It requires 62,000 bhp (46 MW).

A hot-oil pump to deliver 875 gpm (55.2 L/s) at a head of 8600 ft (2620 m) with 19 stages has been built by the Byron Jackson Company; this is one of the highest head pumps in existence.

The Harvey O. Banks Delta Pumping Station, named after the first Director of Water Resources for the State of California, lifts water 244 ft (74.4 m) from the Sacramento-San Joaquin Delta to the California Aqueduct, which delivers water 350 miles (560 km) to Southern California. There are five units that deliver 1067 cfs (30.2 m^3/s) each and two units that deliver 350 cfs (9.9 m^3/s) each for a total plant capacity of over 6000 cfs (170 m^3/s).

One of the world's largest pumping installations is the Edmonston Pumping Plant of the State of California water project. This plant accepts water from the California Aqueduct, and lifts water over the Tehachapi Mountains. At this plant there are 14 four-stage vertical-shaft centrifugal pumps. Each is capable of delivering 315 cfs (8.92 m^3/s) against a head of 1970 ft (600 m) when rotating at 600 rpm. Their maximum efficiency is 92%. The maximum energy requirements for this plant are approximately 6×10^9 kWh per year.

At the opposite end of the scale, axial pumps with impeller diameters as small as 6 mm have been developed for installing in human arteries to help the heart pump blood. They deliver up to 4 L/min against a pressure head of 80 mmHg.

PROBLEMS

15.1 The diameter of the discharge side of a pump is 100 mm, and that of the intake pipe is 120 mm (Fig. P15.1). The pressure gage at discharge reads 240 kPa and the pressure gage at intake reads 60 kPa. If $Q = 45$ L/s of water and the pump efficiency is 0.82, find the power delivered to the pump by the drive shaft. The intake of the pump is 350 mm below the discharge side. Neglect the effects of prerotation in the entry pipe and assume smooth flow at discharge from the pump.

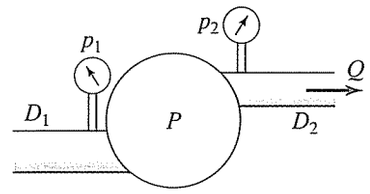


Figure P15.1

15.2 The diameter of the discharge side of a pump is 6 in, and that of the intake pipe is 8 in (Fig. P15.1). The pressure gage at discharge reads 30 psi, and the vacuum gage at intake reads 10 inHg. If $Q = 3.0$ cfs of water and the brake horsepower is 35.0, find the efficiency of the pump. The intake and discharge are at the same elevation.

15.3 When operating at its BEP, the pump of Fig. 15.6 develops 60 ft of head at a capacity of 10,500 gpm. (*a*) Calculate the horsepower delivered to the water by the pump. (*b*) If the brake horsepower is 192 hp, what is the efficiency of the pump at its BEP? (*c*) Does your answer to (*b*) agree with Fig. 15.6?

15.4 The pump of Fig. 15.6 delivers 29 ft of head at a capacity of 15,000 gpm. The pump efficiency under these conditions of operation is 47%. Calculate the water horsepower and the brake horsepower, and check to see if your answers agree with Fig. 15.6.

15.5 A centrifugal pump delivers 40 gpm of gasoline ($s = 0.72$) when developing a net head of 15 ft. If the shaft power is 0.126 hp, what is the efficiency of the pump?

15.6 A pump delivers 1000 L/s of water at a head of 8.5 m. If the efficiency of the pump is 68%, what is the shaft power?

15.7 A centrifugal pump with an 18-in-diameter impeller is rated at 25 cfs under a head of 100 ft when running at 1200 rpm. (*a*) If its efficiency is 85%, what is the brake horsepower? (*b*) If the same pump is run at 1800 rpm, what would be its rating in terms of h, Q, and the brake horsepower? Assume the efficiency does not change with the change in speed.

15.8 A centrifugal pump with a 300-mm-diameter impeller is rated at 40 L/s against a head of 25 m when rotating at 1750 rpm. What would be the rating of a pump of identical geometric shape with a 200-mm-diameter impeller? Assume pump efficiencies and pump speeds are identical.

15.9 At its optimum point of operation, a given centrifugal pump delivers 3.2 m^3/s of water against a head of 25 m when rotating at 1450 rpm. (*a*) If the efficiency is 82%, what is the brake power of the drive shaft? (*b*) If a homologous pump with an impeller diameter one half as large is rotating at 1200 rpm, what would be the discharge, head, and shaft power at its point of optimum efficiency? Assume both pumps operate at the same efficiency.

15.10 Using Eq. (15.7), find the approximate efficiency of the smaller pump of Prob. 15.9.

15.11 At its BEP, a given pump develops 30 ft of head when operating at 720 rpm. The flow rate is 4.2 cfs. Find the head and flow rate for a homologous pump operating at 600 rpm if its diameter is 90% that of the given pump.

15.12 Plot h versus Q for the pump of Fig. 15.6 for the case where $n = 1200$ rpm.

15.13 Plot h versus Q for the pump of Fig. 15.7 for the case where $n = 600$ rpm.

15.14 A centrifugal pump is installed to deliver water from a reservoir of water-surface elevation zero to another of elevation 300 ft (Fig. P15.14). The 12-in-diameter suction pipe ($f = 0.020$) is 100 ft long and the 10-in-diameter discharge pipe ($f = 0.026$) is 5000 ft long. The pump characteristic at 1200 rpm is defined by $h_p = 375 - 24Q^2$, where the pump head h_p is in ft and Q is in cfs. Calculate the rate at which this pump will deliver water, assuming the setting is low enough to avoid cavitation.

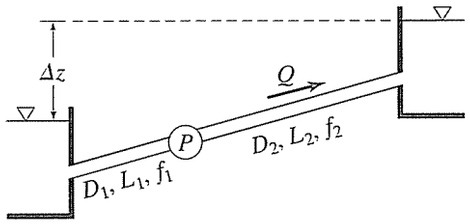


Figure P15.14

15.15 A pump whose characteristics are shown on Fig. 15.7 when operating at 450 rpm is used to pump water from reservoir A to reservoir B through a 24-in-diameter pipe ($f = 0.028$) of length 170 ft. Neglecting minor losses, find the flow rate and pump efficiency for the following conditions: (a) reservoir water-surface elevations identical; (b) water-surface in reservoir B is 10 ft higher than that in reservoir A; (c) water-surface elevation in reservoir B is 10 ft lower than that in reservoir A.

15.16 Repeat Prob. 15.15 using the same data except consider minor losses: square entrance and submerged discharge. Neglect bend losses.

15.17 A pump whose characteristics are shown in Fig. 15.8, operating at 1450 rpm, is placed in a 24-in-diameter pipe ($f = 0.032$) of length 5000 ft (Fig. P15.17). The discharge end of the pipe is 30.0 ft lower than the water-surface elevation in the reservoir. The water discharges freely into the atmosphere. (a) Find the flow rate. (b) What would be the flow rate if the pump had not been installed in the pipeline?

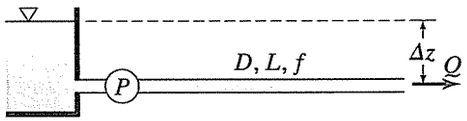


Figure P15.17

15.18 Repeat Prob. 15.17 with all data the same except operate the pump at 1160 rpm rather than 1450 rpm. Note the pump efficiency under this mode of operation.

15.19 The characteristic curve for a pump operating at 1200 rpm is given by the following:

Head, m	Q, L/s
20	0
15	165
10	250

This pump is installed in a pipeline to augment the flow from reservoir A to reservoir B (Fig. P15.19). When the water-surface elevation in reservoir A exceeds that in reservoir B by 15 m, the flow rate is 150 L/s. What will be the flow rate when the elevation difference is 3 m? Assume the system curve is parabolic.

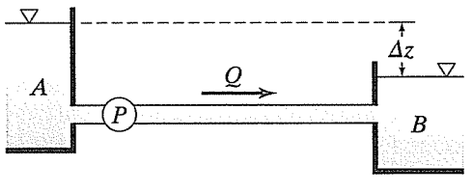


Figure P15.19

15.20 All dimensions of a model pump are one-third as large as the corresponding dimensions of a prototype pump. The model pump operating at 1800 rpm delivers 3.0 cfs at a head of 35 ft. (a) Find the speed and capacity of the prototype if it is to develop a head of 35 ft. (b) Find the specific speed of the model pump and the prototype. They should be the same. Assume the efficiency of the model and prototype are the same.

15.21 All dimensions of a model pump are one-third as large as the corresponding dimensions of a prototype pump. If the model pump operating at 1800 rpm delivers 3.0 cfs at a head of 35 ft, find (a) the speed and head of the prototype when it delivers 3.0 cfs. Find also (b) the specific speed of the model pump and the prototype. They should be the same. Assume the efficiency of the model and prototype are the same.

15.22 All dimensions of a model pump are one-third as large as the corresponding dimensions of the prototype pump. If the model operating at 1800 rpm delivers 3.0 cfs at a head of 35 ft, find (a) the speed and capacity of the prototype if is to develop a head of 140 ft. Assume the efficiency of the prototype and the model are the same. Find also (b) the specific speed of the model and the prototype.

15.23 A centrifugal pump is required to deliver 1600 gpm against a head of 350 ft at a rotative speed of 1150 rpm. Would a two-stage pump be more efficient than a single-stage?

15.24 A pump is to deliver 2.0 m³/s against a head of 160 m when operating at 300 rpm. What type of pump will be required?

15.25 At its optimum point of operation, a given centrifugal pump with an impeller diameter of 500 mm delivers 3.2 m³/s of water against a head of 25 m when rotating at 1450 rpm. (a) If its efficiency is 82%, what is the brake power of the drive shaft? (b) If a homologous pump with a diameter of 800 mm is rotating at 1200 rpm, what would be the discharge, head, and shaft power? Assume both pumps operate at the same efficiency. (c) Compute the specific speed of both pumps.

15.26 A three-stage pump is rated at 7500 gpm against a head of 600 ft at a speed of 1200 rpm. The three impellers are identical. Find the N_s of this pump and the approximate diameter of the impellers.

15.27 A four-stage pump is designed to deliver 65 L/s against a head of 120 m when operating at 1450 rpm. The four impellers are identical. Find the specific speed of this pump. What type of pump is it? Estimate the diameter of the impellers.

15.28 Estimate the diameter of the impeller of the model pump described in Exer. 15.4.1.

15.29 Approximately what would be the diameter of the impeller of a pump that at its BEP delivers 150 L/s of water at a head of 45 m when operating at 1750 rpm.

15.30 Approximately how much head will be developed by a pump when operating at its point of optimum efficiency if it delivers 6500 gpm when running at 1200 rpm? The diameter of the impeller is 15.0 in.

15.31 Calculate the values of ϕ_e for the pumps of Sample Prob. 15.2 and compare these values with those given in Fig. 15.11. Perform these calculations for both the model and the prototype, given the model diameter is 7.4 in.

15.32 At maximum efficiency, the pump at Rocky River delivers 280 cfs at a head of 238 ft when operating at 327 rpm. Calculate the specific speed of this pump and provide an estimate of the diameter of the impeller.

15.33 At the Grand Coulee project on the Columbia River there are several identical pumps, each with an impeller diameter of $167\frac{3}{8}$ in. The rotative speed of these pumps is 200 rpm and the maximum efficiency is 90.8% when discharging 1250 cfs at a head of 344 ft. Find the specific speed and calculate ϕ_e. How does the calculated value of ϕ_e compare with the value shown in Fig. 15.11?

15.34 Approximately what head would you expect a pump (impeller diameter = 10.0 in) to develop at 1200 rpm if its specific speed is 3500?

15.35 A given axial-flow pump delivers 300 L/s at a head of 6 m when operating at 1800 rpm. If this pump were operating on the moon at the same speed, what head would it develop when delivering water at 300 L/s? Assume the gravitational constant of the moon is one-sixth that of the earth. Assume cavitation does not occur.

15.36 Is the pump in Prob. 15.35 likely to cavitate? Why?

15.37 A pump with a critical value of σ_c of 0.20 is to pump against a head of 200 ft. The barometric pressure is 14.3 psia, and the vapor pressure of the water is 0.8 psia. Assume the friction losses in the intake piping are 4 ft. Find the maximum allowable height of the pump relative to the water surface at intake to assure that cavitation will not occur.

15.38 A boiler feed pump delivers water at 200°F, which it draws from an open hot well with a frictional loss of 2 ft in the intake pipe (Fig. P15.38). The barometric pressure is 29 inHg, and the value of σ_c for the pump is 0.16. What must be the elevation of the water surface in the hot well relative to that of the pump intake? The total pumping head is 140 ft.

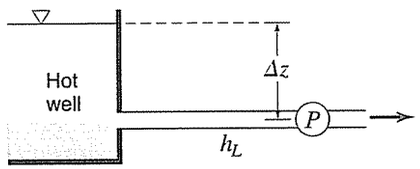


Figure P15.38

15.39 Repeat Prob. 15.38 with all data identical except for the total pumping head. Consider three cases as follows: total pumping heads of 40 ft, 100 ft, and 180 ft.

15.40 Suppose a pump was pumping water at a head of 30 ft with a water temperature of 100°F and a barometric pressure of 14.2 psia. At intake to the pump the pressure is 17 inHg vacuum and the pipeline velocity is 16 ft/sec. What are the values of NPSH and σ?

15.41 A centrifugal pump with $(N_s)_{SI} = 1.48$ is to pump 30 L/s of gasoline ($s = 0.82$, vapor pressure = 32 kPa) from an open tank (Fig. P15.41). The pump impeller rotates at 157 rad/s. Assuming the free surface in the tank is at elevation 100.0 m and the barometric pressure is 101 kPa abs, what is the highest elevation at which the pump centerline may be placed without cavitation problems? The suction pipe ($f = 0.022$) is 40 m long and has a diameter of 120 mm. Assume square entrance conditions where the pipe takes off from the tank. Express the answer in terms of h.

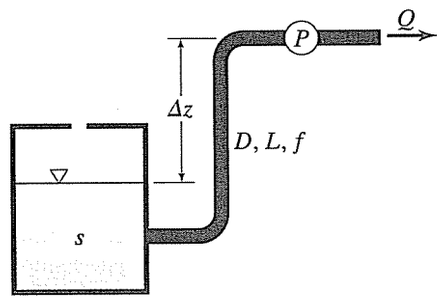


Figure P15.41

15.42 At rated capacity a four-stage centrifugal pump rotates at 1200 rpm and delivers 360 L/s of water against a head of 180 m. What diameter of impellers should be used? The suction and discharge flanges have inside diameters of 300 mm and 320 mm, respectively, and both are located 0.7 m below the pump centerline. At what gage pressure at the suction flange may cavitation be expected? Water temperature is 34°C and barometric pressure is 98 kPa abs.

15.43 Rework Prob. 15.42 for the case of a three-stage pump. Assume speed, flow rate, and total head are unchanged.

15.44 A pump with a critical value of $\sigma_c = 0.18$ is to pump against a head of 80 m. The barometric pressure is 98.5 kPa abs, and the vapor pressure is 5.4 kPa abs. Assume the friction losses in the intake are 1.2 m. Find the maximum allowable elevation of the pump relative to the water surface at intake.

15.45 Determine the specific speed of a centrifugal pump that is rated at 3500 gpm under a head of 70 ft at 1750 rpm. What would be the head and capacity of this pump if operated at 1160 rpm? For each rotative speed note the maximum tolerable suction lift as recommended in Fig. 15.13.

15.46 By scaling the necessary data off Fig. 15.14, find the approximate specific gravity of the liquid whose kinematic viscosity is 396 times that of water. Note that the maximum efficiency when pumping this liquid is 18%, and when pumping water it is 85%.

15.47 Water at 60°F flows by gravity between two reservoirs. The water surface in reservoir A is at elevation 5010 ft and that in reservoir B is at elevation 4980 ft. The water flows through a 24-in-diameter pipe ($f = 0.026$) of length 8000 ft (Fig. P15.47). (a) What is the flow rate? Neglect minor losses. (b) If one wishes to double the flow rate using a pump operating at 1800 rpm, what specific speed pump would you recommend? (c) Approximately what would be the diameter of the impeller of the pump? (d) If the pump were set at the upper end of the pipeline, at what elevation must the pump be set to safeguard against cavitation if the head loss from reservoir A to the pump is 1.5 ft?

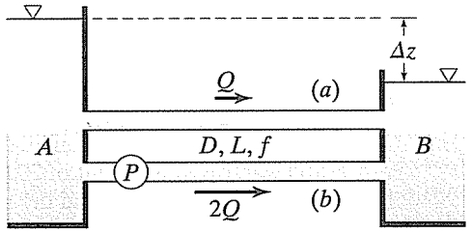


Figure P15.47

15.48 Repeat Prob. 15.47 for the case where the pump is to operate at 600 rpm rather than 1800 rpm, with all other data remaining the same.

15.49 Water at 170°F is pumped through a 15-in-diameter pipe ($f = 0.028$) 2000 ft long from a large pressurized tank ($p = 30$ psi) whose water surface is at elevation 200 ft to a point of discharge in the atmosphere at elevation 242 ft (Fig. P15.49). (a) If the flow rate is 5800 gpm, find the head that a pump, placed in the pipeline and rotating at 1200 rpm, must develop and determine the specific speed of the pump. Assume the pressure in the tank remains constant and neglect minor losses. (b) Approximately what will be the diameter of the impeller? (c) At what elevation must the pump be set to safeguard against cavitation if the head loss from the tank to the pump is 1.6 ft?

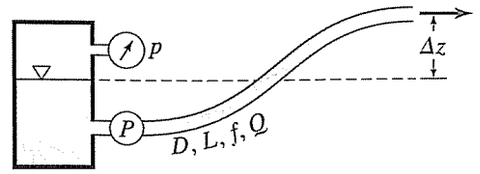


Figure P15.49

15.50 A pump manufacturer is asked to provide a pump that will deliver 84,500 gpm against a head of 225 ft. Several speeds of operation are considered, namely 225 rpm, 600 rpm, 1200 rpm, and 1800 rpm. Determine the specific speeds of pumps that will operate at these speeds and estimate the diameter of each of their impellers. What factors might one consider in making a choice among them?

15.51 You are asked to select a single-stage or multistage pump to deliver 10 cfs against a head of 300 ft when operating at 450 rpm. Specify the number of stages, the specific speed, and the approximate diameter of the impeller.

15.52 You are asked to select a single-stage or multistage pump to deliver 300 cfs against a head of 10 ft when operating at 450 rpm. Specify the number of stages, the specific speed and the approximate diameter of the impeller.

15.53 A submersible centrifugal pump in a water well is to be set 120 ft below the land surface. The steel discharge pipe from the pump to the surface has a diameter of 6.0 in. The water surface in the well at maximum drawdown is 106 ft below the land surface. How many stages ($N_s = 2650$) would you recommend for the pump if the discharge from the well at maximum drawdown is 800 gpm and the pump operates at 1450 rpm?

15.54 A pump is required to deliver 830 L/s against a head of 230 m. If the minimum desirable specific speed from an efficiency standpoint is $(N_s)_{SI} = 0.35$ and the motor speed is 1500 rpm, how many stages would you recommend? Also, what is the maximum permissible suction lift?

15.55 In planning a pumping station an engineer decided to use two identical pumps operating in parallel. These pumps, when both are operating, must deliver a total of 200 cfs against a head of 40 ft. Calculate the specific speed of each of these pumps and estimate the diameter of the impellers.

15.56 In planning a pumping station, an engineer decided to use two identical pumps operating in series. These pumps, when both are operating, must deliver 200 cfs against a total head of 40 ft. Calculate the specific speed of each of these pumps and estimate the diameter of the impellers.

15.57 A very long pipe connects two reservoirs whose water levels are the same. Identical pumps A and B are connected to the pipe in parallel. The operating characteristics of each of the pumps are shown in Fig. 15.6. The pumps are operated at 1450 rpm. When only one pump is operating, the flow rate was found to be 13,200 gpm. (*a*) How much flow will there be when both pumps operate? (*b*) Find also the rate of delivery (with one and two pumps) if the pump speeds are reduced to 1200 rpm?

15.58 Select the specific speed of the pump or pumps required to convey 1.0 cfs of water against a head of 115 ft. The pump rotative speed is 1750 rpm. Consider the following cases: a single pump; two pumps in parallel; three pumps in parallel; two pumps in series; three pumps in series.

15.59 The pump of Fig. 15.6 is placed in an 8-in-diameter pipe ($f = 0.030$) 150 ft long that is used to lift water from one pond to another (Fig. P15.59). The difference in water-surface elevations between the two ponds varies from 20 ft to 50 ft. Plot a curve showing delivery rate versus water-surface elevation difference.

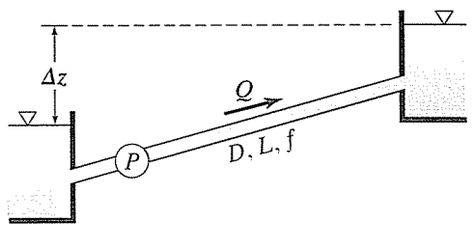


Figure P15.59

15.60 Under normal operating conditions, a centrifugal pump with an impeller diameter of 20.0 in delivers water at a head of 170 ft with an efficiency of 70% at 1800 rpm. Compute the peripheral velocity of the impeller, the specific speed, and the flow rate.

15.61 A centrifugal pump driven by an electric motor lifts water a total height of 200 ft. The pump efficiency is 78% and the motor efficiency is 88%. The lift is through 1220 ft of 6-in-diameter pipe and the pumping rate is 350 gpm. If $f = 0.022$ and power costs 7 ¢/kWh, what is the cost of pumping a million gallons of water?

15.62 What is the specific speed of the four-stage pumps mentioned in the last-but-one paragraph of Sec. 15.14? Approximately what is the diameter of the impellers?

CHAPTER 16
Hydraulic Machinery—Turbines

In this chapter we shall deal with hydraulic (i.e., water-driven) turbines. We shall not deal with steam or gas turbines, since their theory is outside the scope of this book. Water-driven turbines are used primarily for the development of hydroelectric energy. Turbines extract energy from flowing water and convert it to mechanical energy to drive electric generators. ***Hydroelectric power,*** as it is called, is an important source of energy. In the United States about 15% of the electric energy is derived from hydropower, though in some countries, such as Norway and Brazil, over 70% of the electric energy is developed at hydroelectric plants.

16.1 HYDRAULIC TURBINES

There are two basic types of hydraulic turbines. In the ***impulse turbine*** a free jet of water impinges on the revolving element of the machine, which is exposed to atmospheric pressure. In a ***reaction turbine,*** flow takes place under pressure in a closed chamber. Although the energy delivered to an impulse turbine is all kinetic, while the reaction turbine utilizes pressure energy as well as kinetic energy, the action of both turbines depends on a change in the momentum of the water so that a dynamic force is exerted on the rotating element, or ***runner.*** The runner of a reaction turbine is similar in design, but not identical to a pump impeller. By rotating either rotor (i.e., centrifugal pump or reaction turbine) in the reverse direction, the machine assumes the function of its counterpart. The efficiency, however, will be small in the reverse mode because the geometric shape of the rotating element and its surrounding casing will not be optimum.

Turbines are operated at constant speed. In the United States, 60-cycle (cycles/sec or Hz) electric current is most common, and under such conditions the rotative speed n of a turbine in revolutions per minute is given by $N = 7200/n$, where N is the number of poles in the generator and must be an even integer. Most 60-Hz generators have from 12 to 96 poles. In many parts of the world, 50-cycle current is used, in which case $N = 6000/n$. The power demand of an electric distribution system varies throughout the day; it is usually higher during daylight hours and lower at night. Consequently there is a variation in the "load" on the system. In a large system the variation in load can be

accommodated by varying the number of generators in operation. Even so, the load on a single generator may vary with time. Therefore, if the generator is driven by a hydraulic turbine, in order to maintain constant speed, there must be some way in which the flow passing through the turbine can be adjusted to regulate the power output of the turbine. Speed regulation will be discussed briefly in Secs. 16.2 and 16.6.

16.2 IMPULSE TURBINES

The impulse turbine (Fig. 16.1) is sometimes called an ***impulse wheel*** or ***Pelton wheel,*** so called in honor of Lester A. Pelton (1829–1908), who contributed much to its development in the early gold-mining days in California. The wheel (or runner) has a series of split buckets located around its periphery (Fig. 16.2). The buckets may be individually cast and bolted to the central spider, or, more commonly, the entire runner is cast as a single unit. When the jet strikes the dividing ridge of the bucket, it is split into two parts that discharge at both sides of the bucket. Each split bucket has a notch that enables the bucket to attain a position nearly tangent to the direction of the jet before the bucket lip intercepts the jet. Only one jet is used on small turbines, but two or more jets impinging at different points around the wheel are often used on large units. The jets are usually produced by a needle nozzle (Sec. 16.5). Some jet velocities exceed 500 fps (150 m/s).

Since water hammer (Sec. 12.6) might occur in the supply pipe if the nozzle were closed too rapidly, some nozzles are provided with a bypass valve that opens whenever the needle valve is closed quickly. The same effect can be obtained with a jet deflector, whose position can be adjusted to deflect the jet away from the wheel when the load drops. A governor is required to actuate the nozzle and bypass or deflector units. The governing mechanism is provided with a compensating arrangement that prevents over-travel of the governor to provide very sensitive speed regulation.

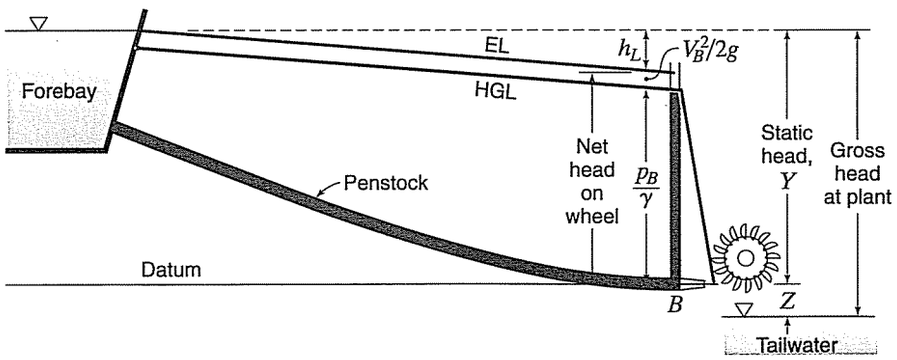


Figure 16.1
Definition sketch for the impulse turbine.

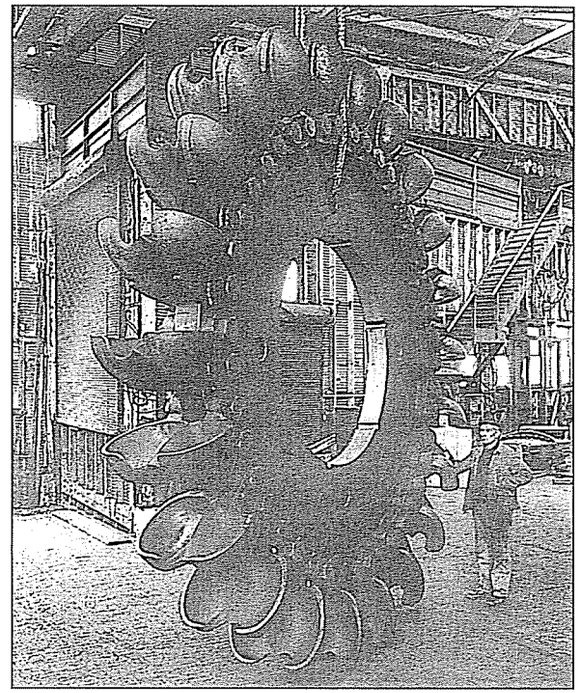


Figure 16.2
Impulse wheel (runner) at Big Creek-2A plant of Southern California Edison Co. Static head = 2418 ft (737 m), net head = 2200 ft (670 m), n = 250 rpm, pitch diameter = 162 in (4.11 m). (This original runner for 50-cycle generation has been replaced by a 300-rpm runner for 60-cycle generation.)

Because of the risk of greatly increased pressures and stresses in the supply pipe due to water hammer (Sec. 12.6), in addition to the protective measures just discussed, engineers specially design and strengthen these pipes. We call such a specially designed supply pipe, for an impulse turbine or a higher-head reaction turbine, a ***penstock.*** To maximize energy delivered, we strive to minimize energy losses in penstocks, so they are usually large in diameter.

The generator rotor is usually mounted on a horizontal shaft between two bearings with the runner installed on the projecting end of the shaft. This is known as a ***single-overhung*** installation. Often runners are installed on both sides of the generator (***double-overhung*** construction; Fig. 16.3) to equalize the bearing loads. Impulse turbines are provided with housings to prevent splashing, but the air within the housing is substantially at atmospheric pressure. Some modern wheels are mounted on a vertical axis below the generator and are driven by jets from several nozzles spaced uniformly around the periphery of the wheel. This arrangement simplifies shaft and bearing design as well as details of the jets and jet deflectors, though piping for the multiple nozzles becomes complicated.

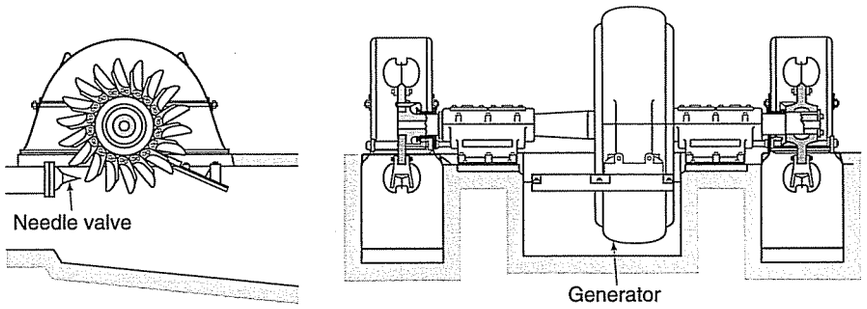


Figure 16.3
Double-overhung impulse wheel installation.

For good efficiency the width of the bucket should be 3–4 times the jet diameter, and the wheel diameter is usually 15–20 times the jet diameter.[1] The ***wheel diameter,*** also referred to as the ***pitch diameter,*** is the diameter of the ***pitch circle,*** the circle to which the centerline of the jet is tangent. The diameters of impulse turbines range up to about 15 ft (5 m). Theoretically maximum efficiency would result if a bucket completely reversed the relative velocity of the jet. This is not possible because the water must be deflected to one side to avoid interfering with the following bucket, and the bucket exit angle β_2 is usually about 165° (Fig. 16.4*b*).

[1] The ratio of wheel diameter to jet diameter varies over a wide range, from 10 to 80 or more, depending on the spacing of the buckets.

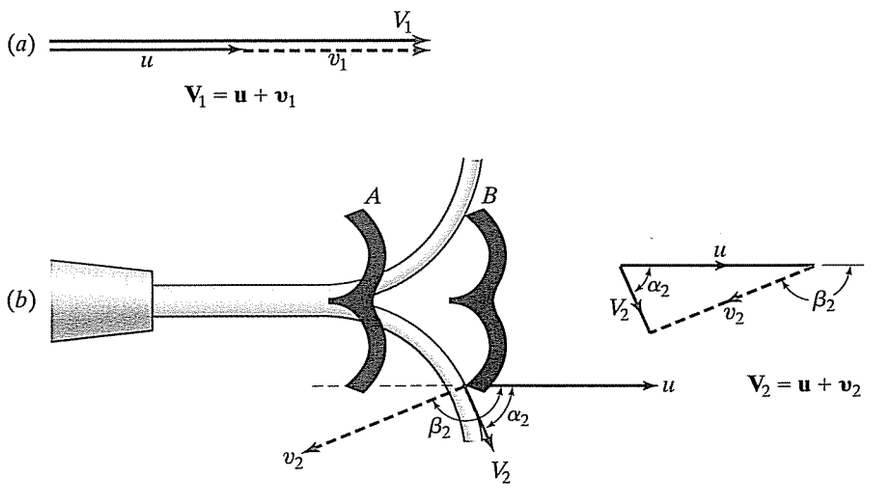


Figure 16.4
Velocity vector diagrams at (*a*) entrance to the bucket and (*b*) at exit from the bucket.

16.3 ACTION OF THE IMPULSE TURBINE

Figure 16.4*a* shows a portion of a rotating impulse turbine with a velocity u at the centerline of the buckets. A plan view of one of the buckets is shown in Fig. 16.4*b*. Position A represents the instant of entry of water into the bucket, while position B indicates the moment the water leaves the bucket. The true path of the water is shown, and its velocity changes from V_1 at entry to V_2 at exit. A vector diagram of the velocities at entry is shown in Fig. 16.4*a*. The vector $\boldsymbol{v}$ represents the velocity of the water relative to the bucket. Assuming friction to be negligible, the magnitude v of the water velocity relative to the bucket remains constant, but its direction at discharge must be tangential to the bucket (Fig. 16.4*b*). Applying the momentum principle, from Eq. (6.24) the force exerted by the water on the bucket in the direction of motion in terms of absolute velocities is

$$F = \rho Q \Delta V = \rho Q(V_1 - V_2 \cos\alpha_2) \tag{16.1}$$

where Q is the discharge of the nozzle. In terms of relative velocities (Sec. 6.7),

$$F = \rho Q \Delta v = \rho Q(v - v\cos\beta_2) = \rho Q(V_1 - u)(1 - \cos\beta_2) \tag{16.2}$$

since the relative velocity $v = V_1 - u$.

The power transmitted to the buckets from the water is the product of the force and the velocity of the body on which the force is acting. Hence,

$$P = Fu = \rho Q(V_1 - u)(1 - \cos\beta_2)u \tag{16.3}$$

From Eq. (16.3), we observe that there is no power developed when $u = 0$ or when $u = V_1$. For a given turbine and jet the maximum power occurs at an intermediate u, which can be found by differentiating Eq. (16.3) and equating to zero. Thus

$$\frac{dP}{du} = \rho Q(1 - \cos\beta_2)(V_1 - u - u) = 0 \tag{16.4}$$

from which $u = V_1/2$. Thus the greatest hydraulic efficiency (neglecting fluid friction) occurs when the peripheral speed of the wheel is half of the jet velocity. Tests of impulse turbines show that, because of energy losses, the best operating conditions occur when u/V_1 is between 0.43 and 0.48.

In the preceding analysis we have assumed that the buckets are shaped at the split in such a manner that the bucket angle at entrance is 0°; i.e., the water enters the bucket tangentially to the bucket. In actuality, there is usually a small bucket angle of less than 10°, in which case the V_1 in Eq. (16.1) should be replaced by $V_1\cos\alpha_1$. A small error, usually less than 2% is introduced by assuming $\alpha_1 = 0°$; hence we commonly make that assumption. Of somewhat greater importance is the fact that in the preceding analysis, which is idealized, we neglected fluid friction as the flow passes through the buckets. Because of fluid friction, the velocity v of the water relative to the buckets gets smaller as the water flows through the buckets. The velocity v_2 of the water relative to the

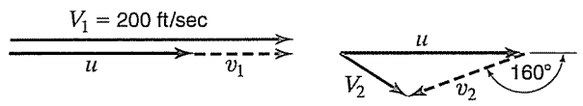
buckets at exit is usually between $0.8v_1$ and $0.9v_1$. In the development of Eqs. (16.1)–(16.4) we assumed $v = v_1 = v_2$. This assumption leads to substantial inaccuracy. To achieve accurate results, the drop in v as it passes through the buckets should be taken into consideration. Sample Probs. 16.1 and 16.2 illustrate the effect of neglecting the decrease in velocity as the water flows through the buckets.

SAMPLE PROBLEM 16.1 Consider an impulse turbine with a pitch diameter of 10.0 ft and a bucket angle β_2 of 160°. If the jet velocity V_1 is 200 ft/sec, the jet diameter 2.0 in and the rotating speed 240 rpm, find (*a*) the force on the buckets, (*b*) the torque on the runner, and (*c*) the horsepower transferred to the runner. Assume that the velocity relative to the bucket does not change (i.e., assume $v_2 = v_1$).

Solution

$$u = \omega r = 2\pi\left(\frac{240}{60}\right)5 = 125.6 \text{ ft/sec}$$

$$V_1 = 200 \text{ ft/sec} = v_1 + u = v_1 + 125.6 \text{ ft/sec}; \quad v_1 = 74.4 \text{ ft/sec}$$



$$v_2 = v_1 = 74.4 \text{ ft/sec}, \quad v_{2u} = v_2\cos 160° = -69.9 \text{ ft/sec}$$

So $\quad \Delta v_u = v_{2u} - v_{1u} = -69.9 - 74.4 = -144.3$ ft/sec

Eq. (6.17): $\quad V_{2u} = u + v_2\cos 160° = 125.6 - 69.9 = 55.7$ ft/sec

Thus $\quad \Delta V_u = V_{2u} - V_{1u} = 55.7 - 200.0 = -144.3$ ft/sec

So $\quad \Delta V_u = \Delta v_u$

(*a*) From Eq. (6.24): $\quad F_u = \rho Q(\Delta V_u) = \rho Q(\Delta v_u)$,

where $\quad Q = A_j V_j = \pi\left(\frac{1}{12}\right)^2 \times 200 = 4.36 \text{ ft}^3/\text{sec}$

Thus $\quad F_u = 1.940(4.36)144.3 = 1220$ lb $\quad$ ***ANS***

(*b*) $\quad T = F_u r = (1220)5 = 6100$ ft·lb $\quad$ ***ANS***

(*c*) From Eq. (5.38): $\quad P = \frac{T\omega}{550} = \frac{(6100)240(2\pi/60)}{550} = 278.6$ hp $\quad$ ***ANS***

or $\quad P = \frac{F_u u}{550} = \frac{(1220)125.6}{550} = 278.6$ hp

SAMPLE PROBLEM 16.2 Repeat Sample Prob. 16.1 for the case where $v_2 = 0.9v_1$. Calculate the percent error made by assuming $v_2 = v_1$ in Sample Prob. 16.1.

Solution

From the solution to Sample Prob. 16.1:

$$u = 125.6 \text{ ft/sec}, \quad v_1 = 74.4 \text{ ft/sec}, \quad \text{and} \quad Q = 4.36 \text{ cfs}$$

$$v_2 = 0.9v_1 = 67.0 \text{ ft/sec}$$

$$\Delta v_u = v_{2u} - v_{1u} = 67.0\cos 160° - 74.4 = -137.4 \text{ ft/sec}$$

(*a*) $\quad F_u = 1.940(4.36)137.4 = 1162$ lb $\quad$ ***ANS***

(*b*) $\quad T = F_u r = (1162)5 = 5810$ ft·lb $\quad$ ***ANS***

(*c*) $\quad P = F_u u/550 = (1162)125.6/550 = 265.4$ hp $\quad$ ***ANS***

(*d*) $\quad \text{Error} = \frac{278.6 - 265.4}{265.4} = 0.050$, or 5.0% $\quad$ ***ANS***

EXERCISES

16.3.1 In Sample Prob. 16.1 subtract the horsepower of the absolute velocity of the water at bucket exit from the horsepower of the jet at bucket entrance. What does the result represent? Why?

16.3.2 Repeat Sample Prob. 16.1 for the case where $v_2 = 0.8v_1$ and calculate the percent error in the horsepower made by assuming $v_2 = v_1$ in Sample Prob. 16.1.

16.3.3 Consider an impulse turbine with a pitch diameter of 2.0 m and a bucket angle β_2 of 165°. If the jet velocity is 86 m/s, the jet diameter 108 mm, and the rotative speed 375 rpm, find (*a*) the force on the buckets, (*b*) the torque on the runner, and (*c*) the power transferred to the runner. Assume that the velocity relative to the bucket does not change (i.e., assume $v_2 = v_1$).

16.4 HEAD ON AN IMPULSE TURBINE AND EFFICIENCY

The ***gross head*** for a power plant is the difference in elevation between headwater and tailwater, or $Y + Z$ in Fig. 16.1. The pressure within the case of an impulse wheel is atmospheric, and consequently this is the pressure at which the jet is discharged. Thus the portion Z is unavailable; so that part of the gross head available to the wheel itself is Y only. This is also called the ***static head.*** Setting the wheel too near the surface of the tailwater is not practical because it might be submerged with a rise in tailwater level.

We consider the nozzle an integral part of the impulse turbine. Hence the ***net head,*** or ***effective head*** h on the turbine (including nozzle), i.e., the head at the base B (entrance; Sec. 11.6) of the nozzle in Fig. 16.1, is the static head minus

the pipe friction losses. Thus, for net head, we have

$$h = \frac{p_B}{\gamma} + \frac{V_B^2}{2g} = Y - h_L \tag{16.5}$$

The energy, or head, supplied at the base of the nozzle is expended in four ways. Some energy is lost in fluid friction in the nozzle, known as the nozzle loss (Sec. 8.23 and Eq. 11.14); a portion is expended in fluid friction over the buckets, causing v_2 to be less than v_1; kinetic energy is carried away in the water discharged from the buckets ($V_2^2/2g$); and the rest is available to the buckets. Thus

$$h = \left(\frac{1}{C_v^2} - 1\right)\left[1 - \left(\frac{A_j}{A_B}\right)^2\right]\frac{V_j^2}{2g} + k\frac{v_2^2}{2g} + \frac{V_2^2}{2g} + h'' \tag{16.6}$$

where C_v = coefficient of velocity of the nozzle (Sec. 11.6)
A_j = cross-sectional flow area of the jet
A_B = cross-sectional flow area of the pipe upstream of the nozzle
V_j = jet velocity (this is the same as V_1 in Fig. 16.4)
v_2 = velocity of water relative to bucket at exit from bucket
V_2 = absolute velocity of water leaving the bucket
h'' = energy head directly available to the buckets (i.e., transferred to the runner)

The nozzle loss is often written as $k_n V_j^2/2g$. Typical values of k, the ***bucket friction loss coefficient,*** vary from 0.2 to about 0.6. The greater part of the energy delivered to the buckets is used in driving the generator, but some of it is used in overcoming mechanical friction in the bearings and in windage (air friction) loss. Some energy may also be lost if not all the water issuing from a nozzle acts effectively on the buckets (known as volumetric losses).

Since the nozzle is considered an integral part of the impulse turbine, the ***hydraulic efficiency*** η' (eta prime) of the impulse turbine is the ratio of the power transferred directly to the turbine buckets to the power in the flow at the base of the nozzle. Thus, for impulse turbines:

$$\eta' = \frac{\gamma Q h''}{\gamma Q h} = \frac{h''}{h} \tag{16.7}$$

The ***overall efficiency*** (or simply ***efficiency***) η of an impulse turbine is less than the hydraulic efficiency η' because of that part of the energy delivered to the buckets that is lost in the mechanical friction in the bearings and the windage, further reducing the energy delivered to the output shaft. The efficiency of an impulse turbine is given by

$$\eta = \frac{\text{Output (shaft) power}}{\text{Input power}} = \frac{T\omega}{\gamma Q h} \tag{16.8}$$

where h is the head as defined by Eqs. (16.5) and (16.6).

SAMPLE PROBLEM 16.3 Demonstrate that when taking account of fluid friction in the buckets of an impulse wheel by using $kv_2^2/2g$ with values of k ranging from 0.2 to 0.6, this is nearly equivalent to $v_2 = 0.8$–0.9 times v_1.

Solution

$$\frac{v_1^2}{2g} - h_L = \frac{v_2^2}{2g}, \quad \text{where} \quad h_L = k\frac{v_2^2}{2g}$$

Thus: $$\frac{v_1^2}{2g} - k\frac{v_2^2}{2g} = \frac{v_2^2}{2g} \quad \text{and} \quad v_1^2 - kv_2^2 = v_2^2$$

from which: $$v_2 = \frac{v_1}{\sqrt{1+k}}$$

$k = 0.2$, $v_2 = 0.91v_1$ ***ANS***

$k = 0.6$, $v_2 = 0.79v_1$ ***ANS***

SAMPLE PROBLEM 16.4 Refer to Fig. 16.1. Water is delivered from a reservoir through a 4-ft-diameter pipe ($f = 0.025$) 10,000 ft long to a nozzle that emits an 8-in-diameter jet that impinges on an impulse wheel. The surface of the reservoir is at an elevation 2420 ft higher than the nozzle. The impulse wheel is connected to a 20-pole generator in a 60-cycle system. The wheel has a diameter of 106 in and a bucket angle $\beta_2 = 165°$. Assuming the head loss through the nozzle can be expressed as $0.05V_j^2/2g$, where V_j is the jet velocity, find the following: velocity of flow in the pipe, jet velocity, flow rate, speed of the bucket, value of u/V_1, net head, force F on the bucket, torque on the wheel, shaft horsepower, horsepower at the base of the nozzle, and overall turbine efficiency. Assume $v_2 = v_1$, neglect minor losses, bearing friction, and windage.

Solution

Energy equation from surface of reservoir to jet neglecting minor loss at pipe entrance:

$$2420 - 0.025\frac{10{,}000}{4}\frac{V^2}{2g} - 0.05\frac{V_j^2}{2g} = \frac{V_j^2}{2g}$$

Continuity: $$V_j = (48/8)^2 V, \qquad V_j = 36V$$

$$2420 - 62.5\frac{V^2}{2g} - 0.05\frac{(36V)^2}{2g} = \frac{(36V)^2}{2g}$$

$$2420 = \frac{V^2}{2g}(62.5 + 64.8 + 1296) = \frac{V^2}{2g}(1423.3)$$

$V = \sqrt{2g(2420/1423)} = 10.47$ ft/sec, velocity in the pipe. ***ANS***

$V_j = 36(10.47) = 377$ ft/sec ***ANS***

$Q = AV = \pi 2^2(10.47) = 131.5$ cfs ***ANS***

$$N = 7200/n \quad \text{gives} \quad n = 7200/N = 7200/20 = 360 \text{ rpm}$$

$$\omega = \frac{2\pi n}{60} = \frac{6.28(360)}{60} = 37.7 \text{ rad/sec}$$

$$u = \omega r = 37.7\left(\frac{106/2}{12}\right) = 166.5 \text{ ft/sec} \qquad \textit{ANS}$$

Hence: $u/V_1 = 166.5/377 = 0.44$ ***ANS***

Eq. (16.5) for net head:

$$h = \frac{p_B}{\gamma} + \frac{V_B^2}{2g} = Y - h_L = 2420 - f\frac{L}{D}\frac{V^2}{2g}$$

$$= 2420 - 62.5\frac{10.47^2}{2(32.2)} = 2314 \text{ ft} \qquad \textit{ANS}$$

Force on bucket

$$F = \rho Q(V_1 - u)(1 - \cos\beta); \qquad \beta = 165°, \qquad \cos\beta = -0.966$$
$$= 1.94(131.5)(377 - 166.5)(1 + 0.966) = 105{,}600 \text{ lb} \qquad \textit{ANS}$$

Torque on wheel

$$T = Fr = 105{,}600(53/12) = 466{,}300 \text{ ft·lb} \qquad \textit{ANS}$$

Shaft power $P = \dfrac{T\omega}{550} = \dfrac{466{,}300(37.7)}{550} = 31{,}960$ hp ***ANS***

Power at base of nozzle (i.e., power input to entire turbine, including nozzle)

$$P = \frac{\gamma Qh}{550} = \frac{62.4(131.5)2314}{550} = 34{,}520 \text{ hp} \qquad \textit{ANS}$$

Overall efficiency

$$\eta = \frac{\text{Output (or shaft) power}}{\text{Input power}} = \frac{T\omega}{\gamma Qh} = \frac{31{,}960}{34{,}520} = 0.926 = 92.6\% \qquad \textit{ANS}$$

EXERCISES

16.4.1 Refer to Sample Prob. 16.4. (*a*) Find the head h'' delivered to the buckets of the turbine if the absolute velocity at discharge from the buckets is 60 ft/sec. Assume a bucket loss coefficient of 0.30. (*b*) Calculate the hydraulic efficiency of the flow through the turbine buckets. (*c*) What is the overall efficiency of the turbine?

16.4.2 Repeat Sample Prob. 16.4 for the case of a 4-in-diameter jet, with all other data remaining the same.

16.4.3 Refer to Sample Prob. 16.1. Calculate the hydraulic efficiency of the flow through the turbine buckets. Note, this is different than the hydraulic efficiency η' of the turbine because the nozzle is considered an integral part of the turbine.

16.5 NOZZLES FOR IMPULSE TURBINES

In any turbine, in order to maintain a constant speed of rotation, it is necessary that the flow rate be varied in proportion to the load on the machine and for the impulse wheel this is done by varying the size of the jet. This is accomplished by varying the position of the needle in the needle nozzle. The shape of both nozzle tip and needle should be such as to cause a minimum friction loss for all positions of the needle and also such as to avoid cavitation damage to the needle at any position.

An important feature in attaining high efficiency in an impulse wheel is that the jet be uniform, the ideal being to have all particles of water moving in parallel lines with equal velocities and with no spreading out of the jet. Air friction retards the water on the outside of the jet, and the needle causes the velocity in the center to be slightly reduced. Careful design of both nozzle tip and needle will minimize these effects, and a gain in turbine efficiency of several percent has been made by improved nozzle design producing better jets. Values of C_v (Sec. 11.6) for needle nozzles vary from about 0.95 when partly closed to permit one-half of maximum flow to 0.99 at the fully opened position.

For a given pipeline there is a unique jet diameter that will deliver maximum power to a jet. Refer to Fig. 16.1 and note that the power of the jet issuing from the nozzle (Sec. 5.9) may be expressed as

$$P_{\text{jet}} = \gamma Q\frac{V_j^2}{2g} \tag{16.9}$$

where V_j is the jet velocity (equal to V_1 of Eq. 16.1). As the size of the nozzle opening is increased, the flow rate Q gets larger while the jet velocity V_j gets smaller. Hence, from Eq. (16.9) and from the preceding statement, we must conclude that there is some intermediate size of nozzle opening (and hence of jet diameter) that will provide maximum power to the jet. This is best illustrated by a sample problem.

SAMPLE PROBLEM 16.5 A 6-in-diameter pipe ($f = 0.020$) 1000 ft long delivers water from a reservoir with a water-surface elevation of 500 ft to a nozzle at elevation 300 ft (Fig. S16.5). The jet from the nozzle is used to drive a small impulse turbine. If the head loss through the nozzle can be expressed as $0.04V_j^2/2g$ (Eq. 11.14), find the jet diameter that will result in maximum power in the jet. Neglect the head loss at the entrance to the pipe from the reservoir. Evaluate the power in the jet.

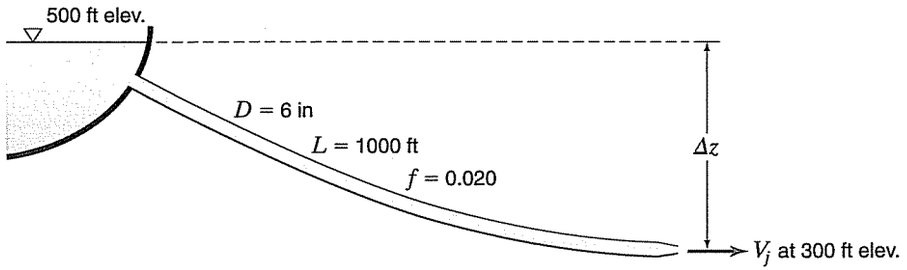


Figure S16.5

Solution
Energy equation (5.28) from reservoir surface to jet:

$$0 + 500 + 0 - 0.02\frac{1000}{0.5}\frac{V_p^2}{2g} - 0.04\frac{V_j^2}{2g} = 0 + 300 + \frac{V_j^2}{2g}$$

If we define the pipe diameter and velocity as D_p and V_p and the jet diameter and velocity as D_j and V_j, from continuity we get

$$A_pV_p = A_jV_j, \quad D_p^2V_p = D_j^2V_j$$

Since the pipe diameter $D_p = 0.50$ ft,

$$0.25V_p = D_j^2V_j \quad \text{and} \quad V_p = 4D_j^2V_j$$

Substituting this expression for V_p in the energy equation gives

$$200 = \frac{V_j^2}{2g}\left(1.04 + 640D_j^4\right)$$

Assuming different values for D_j, we can compute corresponding values of V_j and Q, and then the jet power can be computed using Eq. (16.9). The results are as follows:

D_j, in	D_j, ft	V_j, fps	A_j, ft^2	$Q = A_jV_j$, cfs	P_{jet}, hp
1.0	0.083	111	0.0054	0.60	12.8
1.5	0.125	105	0.0122	1.28	24.2
2.0	0.167	91	0.0218	2.00	29.8
2.5	0.208	76	0.0338	2.57	26.2
3.0	0.250	60	0.0491	2.94	18.8
4.0	0.333	38	0.0873	3.29	8.4
6.0	0.500	18	0.1964	3.49	1.9

Thus a 2-in-diameter jet is the optimum; it will have about 30 hp. ***ANS***

Note: An alternate procedure for solving this problem is to set up an algebraic expression for the power of the jet, P_{jet}, as a function of the jet diameter, D_j, and differentiate P_{jet} with respect to D_j and equate to zero to find the value of D_j for which P_{jet} is a maximum.

EXERCISES

16.5.1 Repeat Sample Prob. 16.5 for the case where the pipe length is 10,000 ft. All other data are to remain the same.

16.5.2 Repeat Sample Prob. 16.5 for the case where the pipe diameter is 12 in. All other data are to remain the same.

16.6 REACTION TURBINES

As mentioned in Sec. 16.1, a reaction turbine is one in which flow takes place in a closed chamber under pressure. The flow through a reaction turbine may be radially inward, axial, or mixed (partially radial and partially axial). There are two types of reaction turbines in general use, the ***Francis turbine*** and the ***axial-flow*** (or ***propeller***) ***turbine.*** The Francis turbine is named after James B. Francis (1815–1892), an eminent American hydraulic engineer who designed, built and tested the first efficient inward-flow turbine in 1849. All inward-flow turbines are known as Francis turbines, both in the United States and Europe, though they have developed into very different forms since the original.

In the usual Francis turbine water enters the ***scroll case*** (Fig. 16.5) and moves into the runner through a series of ***guide vanes*** (Fig. 16.6) with contracting passages that convert pressure head to velocity head. These vanes, known as ***wicket gates,*** are adjustable so that the quantity and direction of flow can be controlled. They are operated by moving a ***shifting ring*** to which each gate is attached. Constant rotative speed of the runner under varying load is achieved by a governor that actuates a mechanism that regulates the gate openings. A relief valve or a surge tank (Sec. 12.7) is generally necessary to prevent serious water hammer pressures.

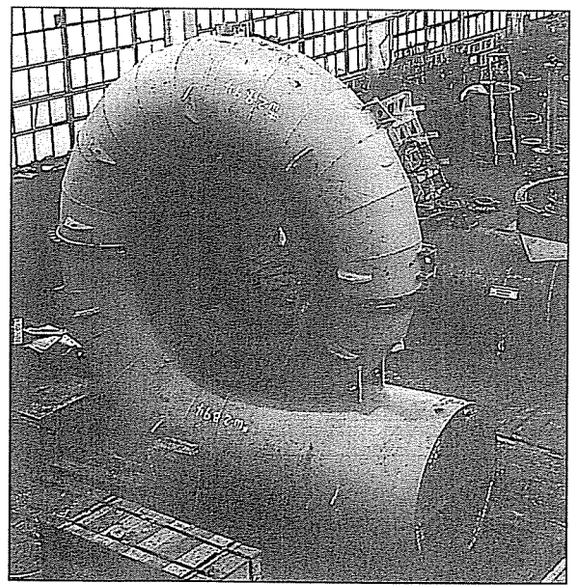


Figure 16.5
Scroll case.

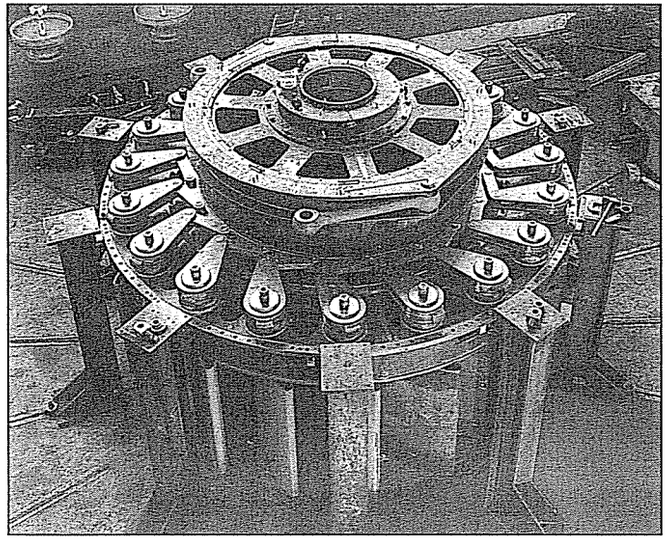


Figure 16.6
Guide-vane assembly showing wicket gates and control mechanism.

Flow through the usual Francis runner (Fig. 16.7) is at first inward in the radial direction, gradually changing to axial. Such turbines are therefore also called ***mixed-flow turbines.*** They are usually mounted on a vertical axis. The scroll case of a Francis turbine is designed to decrease the cross-sectional area in proportion to the decreasing flow rate passing a given section of the casing and thus to maintain constant velocity in the casing so that flow enters the guide vanes uniformly around the periphery of the vanes. Large turbines are sometimes provided with an outside guide-vane assembly known as the ***stay-ring.*** This consists of vanes fixed in position, known as ***stay vanes.*** The vanes of the stay-ring serve as columns to aid in supporting the weight of the generator above, and also they direct the flow smoothly to the inner guide-vane assembly. For heads under about 40 ft (12 m), Francis turbines are often used with an open flume setting (Fig. 16.8), in which case there is no need for a scroll case.

The propeller turbine (Fig. 16.9), an axial-flow machine with its runner confined in a closed conduit, is commonly set on a vertical axis, though it may be set on a horizontal or slightly inclined axis (Fig. 16.10). The usual runner has four to eight blades mounted on a hub, with very little clearance between the blades and the conduit wall. The blades have free outer ends like a marine propeller. Adjustable gates upstream of the runner are used to regulate the flow. These gates are actuated by a governor to maintain constant speed. A ***Kaplan turbine*** is a propeller turbine with movable blades whose pitch can be adjusted to suit existing operating conditions. The adjustment is accomplished by a mechanism in the runner hub, which is actuated hydraulically by the governor in synchronization with guide-vane adjustments. With an axial-flow turbine, the generator may be set outside the water passageway as in Fig. 16.10, or it may be placed in a streamlined watertight steel housing mounted in the center of the

UNIVERSIDAD POLITE... ...LESIANA
BIBLIOTECA

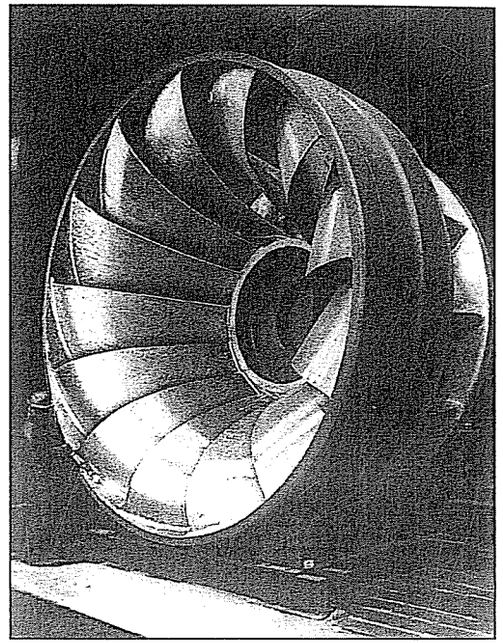


Figure 16.7
One of four vertical-shaft, reaction turbine (Francis) runners for Itauba hydroelectric power station, Brazil. Each unit is designed to generate 172,000 hp (128 MW) at 150 rpm under a head of 287 ft (87.6 m). (Voith Siemens Hydro Generation, Inc., York, PA)

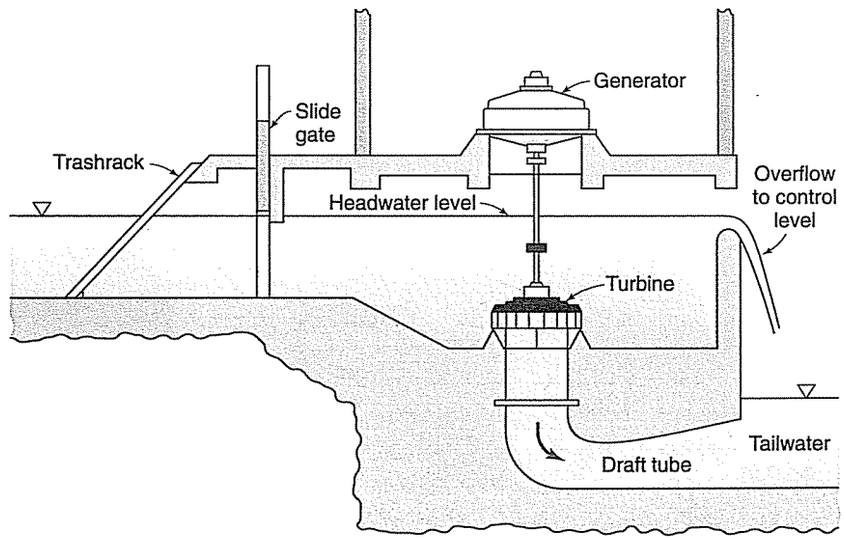


Figure 16.8
Typical open flume setting for a reaction turbine at low heads.

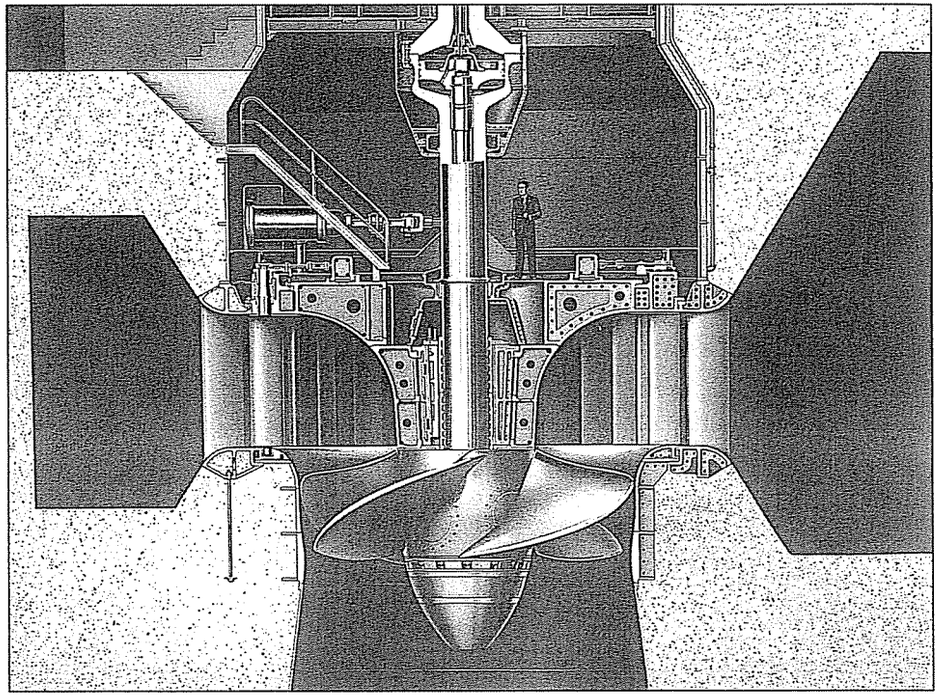


Figure 16.9
Kaplan turbine (a propeller turbine with pitch-adjustable blades) at Watts Bar Dam on the Tennessee R, TN. It develops 42,000 hp (31.3 MW) at 94.7 rpm under a head of 52 ft (15.8 m). Flow is downward.

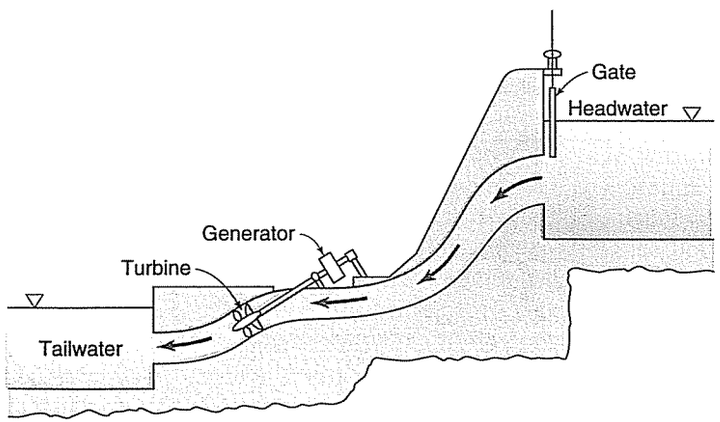


Figure 16.10
Tube turbine (a type of propeller turbine).

passageway. Two other types of propeller turbine include the ***Deriaz,*** an adjustable-blade, diagonal-flow turbine where the flow is directed inwards as it passes through the blades, and the ***tube turbine*** (Fig. 16.10), an inclined-axis type that is particularly well adapted to low-head installations, since the water passages can be formed directly in the concrete structure of a low dam.

16.7 ACTION OF THE REACTION TURBINE[2]

A general understanding of the action of a Francis turbine can be obtained by considering the flow through a radial-flow runner (Fig. 6.10). This is discussed in Sec. 6.11, and the discussion leads to Eq. (6.31), an expression for torque. This represents the torque exerted on the runner by the flowing water. Multiplying Eq. (6.31) by ω, the rotative speed of the runner in radians per second, leads to an expression for power:

$$P = T\omega = \rho Q\omega(r_1V_1\cos\alpha_1 - r_2V_2\cos\alpha_2) \qquad (16.10)$$

where α_1 and α_2 are defined in Fig. 6.10. This represents the power transferred to the shaft of the runner and is referred to as ***shaft power.*** Fluid friction, the fact that all the water passing through does not effectively act on the blades, and friction in the runner bearings slightly reduce the actual power delivered to the runner shaft and generator.

An analysis similar to that of Sec. 6.11 may be used for a mixed-flow runner in which the direction of flow changes from radial to axial. The analysis must, however, be modified to account for the fact that the inflow and outflow vectors do not lie in a plane perpendicular to the axis of the runner. Analysis of a propeller turbine is rather complex, and will not be presented here.

SAMPLE PROBLEM 16.6 A Francis turbine operating at 300 rpm develops a shaft horsepower of 1125 hp when the flow is 120 cfs. Find the torque transmitted from the flowing water to the shaft.

Solution
Shaft power $= T\omega$

$$\frac{T\left(\frac{2\pi n}{60}\right)}{550} = 1125 \text{ hp}, \quad \text{where} \quad n = 330 \text{ rpm}$$

So
$$T = \frac{550 \times 1125}{\left(\frac{2\pi}{60}\right)300} = 19{,}700 \text{ ft·lb} \qquad \textbf{\textit{ANS}}$$

[2] Before reading further, we recommend that you review Sec. 6.11, including Sample Prob. 6.6 in particular.

EXERCISES

16.7.1 A Kaplan turbine operating at 327.3 rpm develops a shaft power of 300 kW when the flow is 5.5 m^3/s. Find the torque transmitted from the flowing water to the shaft.

16.7.2 A torque of 7500 ft·lb is transferred from the flowing water to a reaction turbine whose shaft power is 285.5 hp. How many poles must the generator have to develop 50-cycle electricity?

16.8 DRAFT TUBES AND EFFECTIVE HEAD ON REACTION TURBINES

To operate properly, reaction turbines must have a submerged discharge. The water, after passing through the runner, enters the ***draft tube*** (Figs. 16.8, 16.11, and 16.12), which directs the water to the point of discharge (point 2 in Fig. 16.11). The draft tube is an integral part of a reaction turbine, and its design criteria are usually specified by the turbine manufacturer. The draft tube has two functions. One is to enable the turbine to be set above tailwater level without losing any head thereby. A reduced pressure is produced at the upper end of the draft tube (point 1 in Fig. 16.11), which places a limit on the height z_1 above tailwater at which the turbine runner can be set.

The second function of the draft tube is to reduce the head loss at submerged discharge to thereby increase the net head available to the turbine runner. This is accomplished by using a gradually diverging tube whose cross-sectional

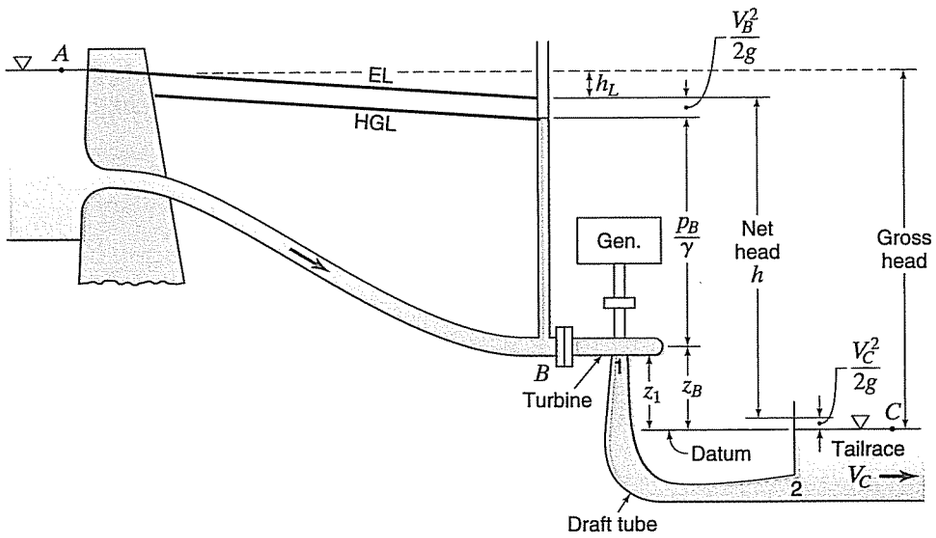


Figure 16.11
Net head h on a reaction turbine. z_B is the draft head.

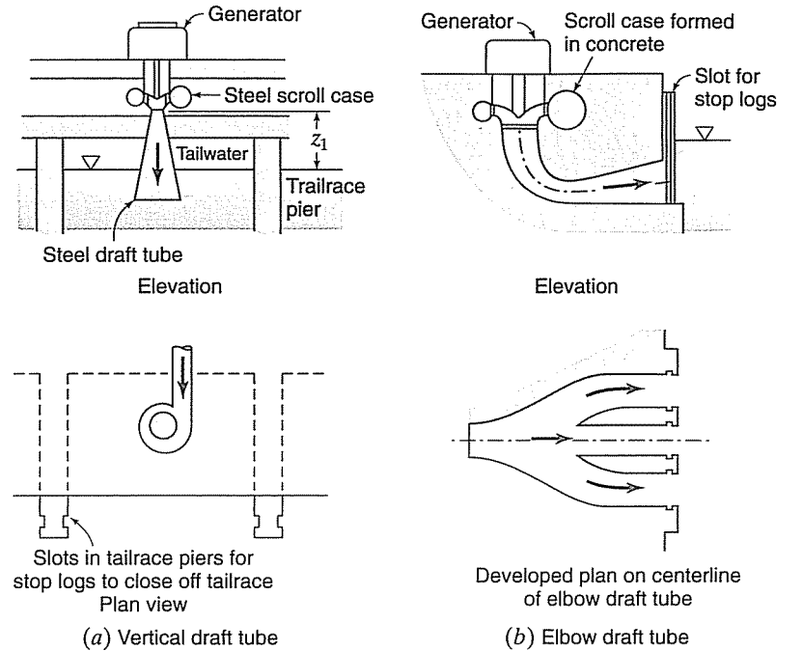


Figure 16.12
Two type of draft tubes for reaction turbines.

area at discharge is considerably larger than the cross-sectional area at entrance to the tube. Applying the energy equation to the draft tube of Fig. 16.11 and letting z_1 signify the elevation of the entrance (or top) of the draft tube above the surface of the water in the tailrace, the absolute pressure head at that section is given by

$$\frac{(p_1)_{abs}}{\gamma} = \frac{p_{atm}}{\gamma} - z_1 - \frac{V_1^2}{2g} + h_L + \frac{V_2^2}{2g} \tag{16.11}$$

where p_{atm} is the atmospheric pressure, h_L is the head loss in the diverging tube, and $V_2^2/2g$ is the kinetic-energy head at exit from the tube. The velocity head at point 2 can be reduced by increasing the cross-sectional area A_2 at exit from the tube. However, there is a practical upper limit to this because of tube length, since the angle of divergence of the tube must be kept reasonably small (usually less than 10°) to prevent or at least minimize separation of the flow from the tube wall. The head loss in the diverging tube can be estimated through application of Eq. (8.78) using the loss coefficients of Fig. 8.20. To prevent cavitation, the vertical distance z_1 (Fig. 16.11) from the tailwater to the draft tube inlet should be limited (Sec. 16.12) so that at no point within the draft tube or turbine will the absolute pressure drop to the vapor pressure of the water.

For a reaction turbine the ***net head*** h is the difference between the energy level just upstream of the turbine and that of the tailrace. Thus in Fig. 16.11 the net head on the turbine $h = H_B - H_C$, or

$$h = \left(z_B + \frac{p_B}{\gamma} + \frac{V_B^2}{2g}\right) - \frac{V_C^2}{2g} \tag{16.12}$$

where z_B is defined as the ***draft head*** and V_C is the velocity in the tailrace. In most instances $V_C^2/2g$ is very small and may be neglected. By comparing Fig. 16.11 with Fig. 16.1, we can see that, for the same setting, the net head on a reaction turbine will be greater than that on a Pelton wheel. The difference is of small importance in a high-head plant, but it is important for a low-head plant.

We consider the draft tube an integral part of a reaction turbine; thus the ***effective head*** h' that is available to act on the runner of a reaction turbine is

$$h' = h - k'\frac{(V_1 - V_2)^2}{2g} - \left[\frac{V_2^2}{2g} - \frac{V_C^2}{2g}\right] \tag{16.13}$$

where h is the net head as defined in the foregoing, and the other two terms refer to the head loss in the draft tube (Eq. 8.78) and the loss at submerged discharge from the tube (Eq. 8.74). The head h'' that is actually extracted from the water by the runner is smaller than h' by an amount equal to the ***shock loss*** at entry to the runner and the hydraulic friction losses in the scroll case, guide vanes, and runner. Shock loss can be quite large if either the magnitude or direction of the velocity changes abruptly when the water leaves the guide vanes and enters the runner. The shock loss and the hydraulic losses in the turbine depend on the conditions under which the turbine is operating.

SAMPLE PROBLEM 16.7 Refer to Fig. 16.11. A draft tube leading from the discharge side of the turbine to the submerged discharge into the tailrace consists of a 40-ft-long pipe ($f = 0.03$) of constant diameter 3 ft. The flow rate is 130 cfs. If this draft tube were replaced by a diverging tube of length 40 ft whose diameter changed uniformly from 3 ft to 8 ft over the 40-ft length, how much additional effective head would be developed by the replacement draft tube? Neglect head loss due to the curvature in the bend and assume the velocity V_C in the tailrace is negligible.

Solution

Original draft tube:

Head loss due to draft tube,

$$h_f = f\frac{L}{D}\frac{V^2}{2g}, \quad \text{where} \quad V = \frac{Q}{A} = \frac{130}{\pi(1.5)^2} = 18.4 \text{ ft/sec}$$

Thus:
$$h_f = 0.03\frac{40}{3}\frac{(18.4)^2}{2(32.2)} = 2.10 \text{ ft}$$

Head loss at discharge,

Eq. (8.74):
$$h_d' = \frac{V_2^2}{2g} - \frac{V_C^2}{2g} = \frac{18.4^2}{2(32.2)} - 0 = 5.26 \text{ ft}$$

Thus total head loss $h_L = h_f + h_d' = 2.10 \text{ ft} + 5.26 \text{ ft} = 7.36 \text{ ft}$

Replacement draft tube:

Head loss due to draft tube,

Eq. (8.78):
$$h' = k'\frac{(V_1 - V_2)^2}{2g}, \quad \text{where} \quad V_1 = 18.4 \text{ ft/sec}$$

and
$$V_2 = \left(\frac{D_1}{D_2}\right)^2 V_1 = \left(\frac{3}{8}\right)^2 18.4 = 2.59 \text{ ft/sec}$$

Obtain k' from Fig. 8.20*a* (we shall assume the solid line applies):

$$\tan\alpha = \frac{(8-3)}{40} = \frac{5}{40} = 0.125, \quad \alpha = 7.1°, \quad k' = 0.13$$

Thus:
$$h' = 0.13\frac{(18.4 - 2.6)^2}{2(32.2)} = 0.50 \text{ ft}$$

Head loss at discharge,

Eq. (8.74):
$$h_d' = \frac{V_2^2}{2g} - \frac{V_C^2}{2g} = \frac{2.59^2}{2(32.2)} - 0 = 0.10 \text{ ft}$$

Thus: Total head loss $h_L = h' + h_d' = 0.50 \text{ ft} + 0.10 \text{ ft} = 0.60 \text{ ft}$

Additional effective head available with the replacement draft tube

$$= 7.36 - 0.60 = 6.76 \text{ ft} \qquad \textit{ANS}$$

Note: $5.26 - 0.10 = 5.16$ ft of the additional effective head is attributable to the decrease in the head loss at discharge. An extra 6.76 ft of effective head is quite significant if we are dealing with a low-head installation.

EXERCISES

16.8.1 Repeat Sample Prob. 16.7 for the case where the flow rate is 90 cfs, with all other data remaining the same.

16.8.2 A Francis turbine operating at 300 rpm develops a shaft power of 1125 hp when the flow is 120 cfs. What is the net head on the runner if the turbine efficiency is 85%?

16.8.3 A Kaplan turbine operating at 327.3 rpm develops a shaft power of 300 kW when the flow is 5.5 m^3/s. What is the net head on the runner if the turbine efficiency is 92%?

16.9 EFFICIENCY OF TURBINES

Equation (16.8) for the efficiency of impulse turbines is equally applicable to reaction turbines. Hence, in general, the efficiency η (eta) of turbines is defined by:

$$\eta = \frac{\text{power delivered to the shaft (brake power)}}{\text{power taken from the water}} = \frac{T\omega}{\gamma Q h} \qquad (16.14)$$

where T is the torque delivered to the shaft by the turbine (Eq. 6.31), ω is the rotative speed in radians per second, Q is the flow rate, and h is the net head on the turbine (Eq. 16.5 for impulse wheels and Eq. 16.12 for reaction turbines).

The efficiencies of various types of turbines change with load (power output) as shown in Fig. 16.13.[3] The impulse turbine maintains high efficiency over a wide range of loads, with significant decrease in efficiency occurring when the load drops below about 30% of normal load.[4] The efficiency of propeller turbines is very sensitive to load, and a significant drop in efficiency is experienced as the load falls below normal load. In contrast, the Kaplan turbine, with its adjustable blades, maintains high efficiency over a wide range of load.

[3] Rated power output refers to conditions of operation at the rated capacity (guaranteed power output) under a unit's normal head, which is usually at a little higher power than normal load.
[4] Normal load refers to the conditions of operation for which efficiency is a maximum.

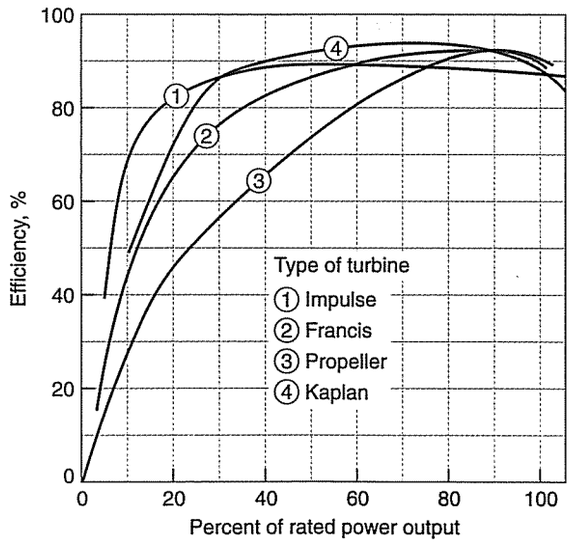


Figure 16.13
Efficiency versus load (power output) for typical turbines.

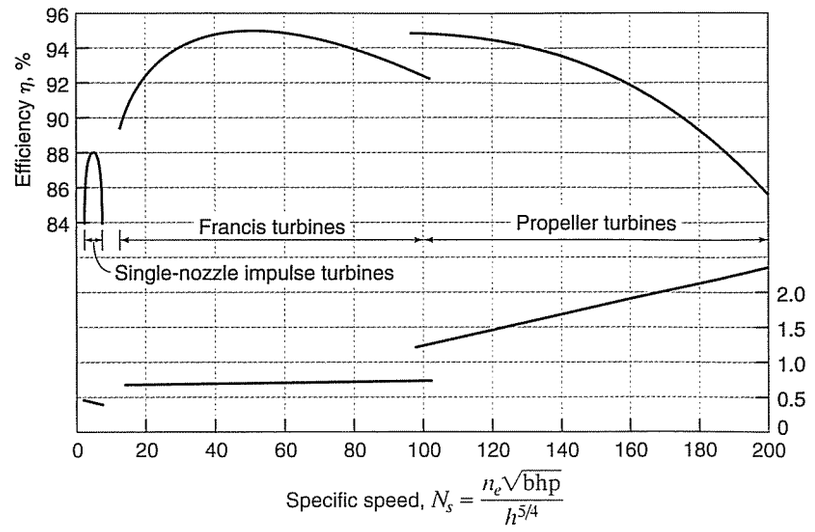


Figure 16.14
Maximum turbine efficiency and typical values of ϕ_e (Sec. 16.11) as functions of specific speed.

Large turbines have somewhat higher efficiencies than small ones, because mechanical friction and windage losses do not increase at the same rate as hydraulic losses. The maximum efficiencies of large turbines as a function of specific speed (Sec. 16.11) are shown in Fig. 16.14.[5] Smaller turbines generally have somewhat smaller maximum efficiencies.

The effect of size (i.e., diameter D) on turbine efficiency is of importance in transferring test results on small models to their prototypes. For both Francis turbines and propeller turbines, this can be accomplished with reasonable accuracy by the ***Moody*[6] *step-up formula,*** which is

$$\frac{1-\eta_m}{1-\eta_p} \approx \left(\frac{D_p}{D_m}\right)^{1/5} \qquad (16.15)$$

where the subscripts m and p refer to the model and prototype, respectively. This equation applies only to homologous machines (with geometrically similar flow patterns, Sec. 15.4), and does not apply to impulse turbines since their efficiency is nearly independent of size. For the case of the Francis turbine the diameter D is the diameter of the runner at flow entrance, while the blade-tip diameter represents D for the axial-flow turbine.

[5] Impulse wheels have specific speeds that range from 2 to 8. Those with $N_s \approx 5$ have maximum efficiency.
[6] Lewis F. Moody (1880–1953), an eminent American engineer and professor, contributed greatly to our understanding of similitude and cavitation as applied to hydraulic machinery.

SAMPLE PROBLEM 16.8 A small Francis runner with a diameter of 2.0 ft is tested and found to have an efficiency of 0.893 when operating under most efficient conditions. Approximately what would be the efficiency of a homologous runner having a diameter of 6.0 ft?

Solution

Eq. (16.15): $$\frac{1-\eta_m}{1-\eta_p} = \left(\frac{D_p}{D_m}\right)^{1/5} = \left(\frac{6}{2}\right)^{1/5} = 3^{0.2} = 1.246$$

$$1-\eta_p = (1-0.893)/1.246$$

From which: $$\eta_p = 0.914 = 91.4\% \qquad \textit{ANS}$$

16.10 SIMILARITY LAWS FOR REACTION TURBINES

In Sec. 15.4 we developed similarity laws for homologous pumps (having geometrically similar machines and flow patterns) and pointed out that these laws can be used to predict performance of a prototype pump from the test of a scaled model. These laws also hold true for reaction turbines; however, they are expressed in different ways than for pumps. For a turbine, we are usually interested in its operation under a certain head, which is fixed by the natural characteristics of the site at which it is to be located. Hence, when dealing with turbines, it is convenient to have h in the similarity expressions.[7] We can accomplish this by rearranging Eq. (15.5) to get

$$n \propto h^{1/2}/D \qquad \text{or} \qquad n = K_n h^{1/2}/D \tag{16.16}$$

Substituting Eq. (16.16) into Eq. (15.4), we get

$$Q \propto nD^3 \propto (h^{1/2}/D)D^3 \propto h^{1/2}D^2$$

Hence, for homologous reaction turbines,

$$Q = K_Q h^{1/2} D^2 \tag{16.17}$$

This expression can also be developed using another approach. Note in Fig. 6.10 that $V_{r_1} \propto V \propto \sqrt{2gh} \propto h^{1/2}$ for a given value of g, and $A_{r_1} = \pi D_1 Z \propto D_1^2$ for geometrically similar runners where V_{r_1} is the radial component of the velocity V at $r = r_1$, A_{r_1} is the effective area through which V_{r_1} is passing, and Z is the width

[7] In applying these similarity laws it would be more appropriate to use the actual head across the runner, h'', rather than the net head h. However, it is more convenient to use h, and in so doing, the results are quite accurate except for the case of very-low-head turbines with heads less than about 10 ft.

of the runner at entrance (i.e., at $r = r_1$). From the above, substituting D for D_1 since they are the same, we get $Q = A_{r_1}V_{r_1} \propto D^2h^{1/2}$, or $Q = K_Q h^{1/2}D^2$.

Since power $P = \gamma Qh$, and from Eq. (16.17) $Q \propto h^{1/2}D^2$, for a given value of γ we get $P \propto h^{1/2}D^2(h) \propto h^{3/2}D^2$. Hence, for homologous reaction turbines with a given value of γ,

$$P = K_p h^{3/2} D^2 \tag{16.18}$$

Equations (16.16)–(16.18) can be used to determine the relationship between homologous reaction turbines.

SAMPLE PROBLEM 16.9 A given turbine runs at a maximum efficiency of 87% when discharging 200 cfs at 150 rpm under a net head of 81 ft. This turbine drives an electric generator that is developing 60-cycle current. (*a*) How many poles are required for this generator? (*b*) What would be the speed, flow rate, and brake horsepower of this same turbine under a net head of 40.5 ft for homologous conditions? (*c*) How many poles would be required for the generator to satisfy the conditions given in (*b*)?

Solution

(*a*) $$N = \frac{7200}{n} = \frac{7200}{150} = 48 \text{ poles},$$

$$\text{bhp} = \frac{\eta\gamma Qh}{550} = \frac{0.87(62.4)(200)81}{550} = 1600 \text{ hp} \qquad \textit{ANS}$$

(*b*) To solve for the speed, we make use of Eq. (16.16):

Eq. (16.16): $n \propto h^{1/2}/D$

So: $$\frac{n_2}{n_1} = \left(\frac{h_2}{h_1}\right)^{1/2}\left(\frac{D_1}{D_2}\right) = \left(\frac{40.5}{81}\right)^{1/2}\left(\frac{D}{D}\right)$$

$$n_2 = n_1\left(\tfrac{1}{2}\right)^{1/2} = 150(0.707) = 106 \text{ rpm} \qquad \textit{ANS}$$

Eq. (16.17): $Q \propto h^{1/2}D^2$

So: $$\frac{Q_2}{Q_1} = \left(\frac{h_2}{h_1}\right)^{1/2}\left(\frac{D_2}{D_1}\right)^2 = \left(\frac{40.5}{81}\right)^{1/2}\left(\frac{D}{D}\right)^2$$

$$Q_2 = Q_1\left(\tfrac{1}{2}\right)^{1/2} = 200(0.707) = 114 \text{ cfs} \qquad \textit{ANS}$$

Eq. (16.18): $P \propto h^{3/2}D^2$

So: $$\frac{P_2}{P_1} = \left(\frac{h_2}{h_1}\right)^{3/2}\left(\frac{D_2}{D_1}\right)^2 = \left(\frac{40.5}{81}\right)^{3/2} \times 1$$

$$P_2 = 1600\left(\tfrac{1}{2}\right)^{3/2} = 566 \text{ hp} \qquad \textit{ANS}$$

(*c*) $N = 7200/n_2 = 7200/106 = 68$ poles **ANS**

Note: If the number of poles does not turn out to be an even number, the rotative speed must be changed somewhat. For example, suppose the head was 66 ft, with all other data unchanged. Then $n_2 = 150(66/81)^{1/2} = 135.4$ rpm. For this case the number of poles $N = 7200/135.4 = 53.2$, which is impossible. So, we shall use 54 poles, in which case $n = 7200/N = 7200/54 = 133.3$ rpm.

SAMPLE PROBLEM 16.10 A turbine homologous to the turbine of Sample Prob. 16.9 has all dimensions one-half as large. For this turbine, find the speed of rotation, flow rate, and brake horsepower when operating under the same net head of 81 ft.

Solution

Eq. (16.16): If h does not change, $n \propto D^{-1}$

Hence: $n_2 = (2/1)n_1 = (2/1)150 = 300$ rpm **ANS**

Eq. (16.17): If h does not change, $Q \propto D^2$

Hence: $Q_2 = (1/2)^2 Q_1 = (1/4)200 = 50$ cfs **ANS**

Eq. (16.18): If h does not change, $P \propto D^2$

Hence: $P_2 = (1/2)^2 P_1 = (1/4)1600 = 400$ hp **ANS**

In these two sample problems we have looked at two different situations. In Sample Prob. 16.9 we took a given turbine and moved it to a different site where the net head was smaller. In so doing, we found a decrease in speed, flow rate, and shaft horsepower. In Sample Prob. 16.10 we compared two turbines that are geometrically similar, but of different size, operating under the same net head. We found that in this situation, to maintain homologous conditions (similar flow patterns), the smaller turbine must rotate at a greater speed. However, the flow rate and shaft horsepower are substantially smaller.

EXERCISES

16.10.1 At maximum efficiency a turbine runs at 150 rpm, discharges 10.0 m^3/s, and develops a shaft power of 2600 kW under a net head of 30 m. (*a*) What is its efficiency? (*b*) What would be the revolutions per minute, the flow rate, and brake power of this same turbine under a net head of 48 m for homologous conditions? (*c*) How many poles would be required for 60-cycle electricity if the net head is 48 m?

16.10.2 At maximum efficiency of 93% a turbine runs at 300 rpm and discharges 310 cfs under a net head of 64 ft. This turbine drives a generator that is developing 50-cycle electricity. (*a*) How many poles are required for this generator? (*b*) What is the brake horsepower?

16.10.3 We wish to operate the turbine of Exer. 16.10.2 under homologous conditions and a net head as close as possible to 48 ft. Find the speed, net head, flow rate, and brake power for the homologous conditions. Be sure to use an even number of poles. *Hint:* It may not be possible to have perfectly homologous conditions.

16.11 PERIPHERAL-VELOCITY FACTOR AND SPECIFIC SPEED OF TURBINES

In Sec. 15.9 we developed Eq. (15.9), an expression for the ***peripheral-velocity factor*** ϕ (phi) applicable to pumps. For turbines, the expression is the same, the only difference being that in the development of the equation we use u_1 rather than u_2. Thus, for turbines,

$$u_1 = \phi\sqrt{2gh} \tag{16.19}$$

where u_1 is the tangential velocity of a point on the periphery of the rotating element. As with pumps, for any given turbine, ϕ varies over a wide range during different conditions of operation. The value of ϕ that is of most importance is that for conditions of maximum efficiency. Hence the ϕ we most commonly use in Eq. (16.19) is that which occurs at maximum efficiency, and we refer to this as ϕ_e (Fig. 16.14).

For convenience, we repeat Eq. (15.10) here:

$$D = \frac{153.3\phi_e\sqrt{h}}{n} \tag{16.20}$$

This equation is equally applicable to reaction turbines, and we can use it to estimate the diameter of the runners for Francis and propeller-type turbines (but not impulse turbines).

Traditionally, in the British Gravitational system we express the ***specific speed*** of a turbine[8] as

$$N_s = \left(\frac{n_e\sqrt{\text{bhp}}}{h^{5/4}}\right)_{\eta_{\max}} \tag{16.21}$$

where the values of n_e (rpm), bhp (shaft or brake horsepower), and h (net head in ft) are those that occur at maximum efficiency. Sometimes we refer to this as

[8] In the metric system the specific speed of turbines has been expressed in various forms, but we will not deal with those here. We will define specific speed of turbines using BG units only (Eq. 16.21).

the "normal" or "nominal" specific speed. Slightly different is the ***rated specific speed,*** which is based on the rated capacity (guaranteed power output) under a unit's normal head; this specific speed is usually at a power a little higher than at the point of maximum efficiency. Two turbines of identical geometric shape, but of different size, will have the same specific speed. So, as with pumps, N_s serves to classify turbines as to type. In fact, we might better call N_s the *type characteristic,* or some similar name, because it indicates the type of turbine (Fig. 16.14). Impulse wheels have low specific speeds, Francis turbines have medium values of N_s, and propeller turbines have high values. Typical values of maximum efficiency and values of ϕ_e for the different types of turbines are shown in Fig. 16.14. Values of ϕ_e vary approximately as follows:

Impulse wheels[9]	0.43–0.48
Francis turbines	0.7–0.8
Propeller turbines	1.4–2.0

SAMPLE PROBLEM 16.11 Assume the Francis turbine of Sample Prob. 16.6 is operating at maximum efficiency. Find the specific speed of the turbine and estimate the diameter of its runner.

Solution

Eq. (16.21): $$N_s = \left(\frac{n_e\sqrt{\text{bhp}}}{h^{5/4}}\right)_{\eta_{max}} = \frac{300\sqrt{1125}}{97.2^{5/4}} = 33.0 \qquad \textit{ANS}$$

From Fig. 16.14: $\phi_e = 0.72$

Eq. (16.20): $$D = \frac{153.3(0.72)\sqrt{97.2}}{300} = 3.63 \text{ ft} \qquad \textit{ANS}$$

EXERCISES

16.11.1 Find the specific speed of an impulse wheel operating at a maximum efficiency of 92.6% under a net head of 2314 ft at 360 rpm if the flow rate from the nozzle is 131.5 cfs. Estimate the diameter of the runner. *Note:* This is the turbine of Sample Prob. 16.4.

16.11.2 Find the specific speed of a turbine that operates at a maximum efficiency of 90% at 150 rpm under a net head of 81 ft with a flow rate of 200 cfs. Estimate the diameter of the runner. What type of turbine is this?

[9] Note that the ϕ_e for impulse wheels is approximately equal to the ratio of u/V_1 of Sec. 16.3.

16.11.3 Find the specific speed of a turbine that operates at a maximum efficiency of 83% at 450 rpm under a net head of 20 ft with a flow rate of 100 cfs. Estimate the diameter of the runner. What type of turbine is this?

16.12 CAVITATION IN TURBINES

Cavitation (Sec. 5.10) is undesirable because it results in pitting (Fig. 16.15), mechanical vibration, and loss of efficiency. The high heads on impulse turbines create extremely high velocities of flow. If the nozzle and buckets are not properly shaped for the particular flow conditions under which the turbine is operating, separation of the flow from the boundaries may cause regions of low pressure and result in cavitation. Low-specific-speed impulse wheels ($N_s \approx 2$) can operate at heads as high as 2000 ft (600 m) without cavitation, while high-specific-speed impulse wheels ($N_s \approx 8$) will experience cavitation at heads in the vicinity of 400 ft (120 m). Figure 16.16 shows recommended limits of safe specific speed of turbines for various heads and settings based on experience at existing power plants.

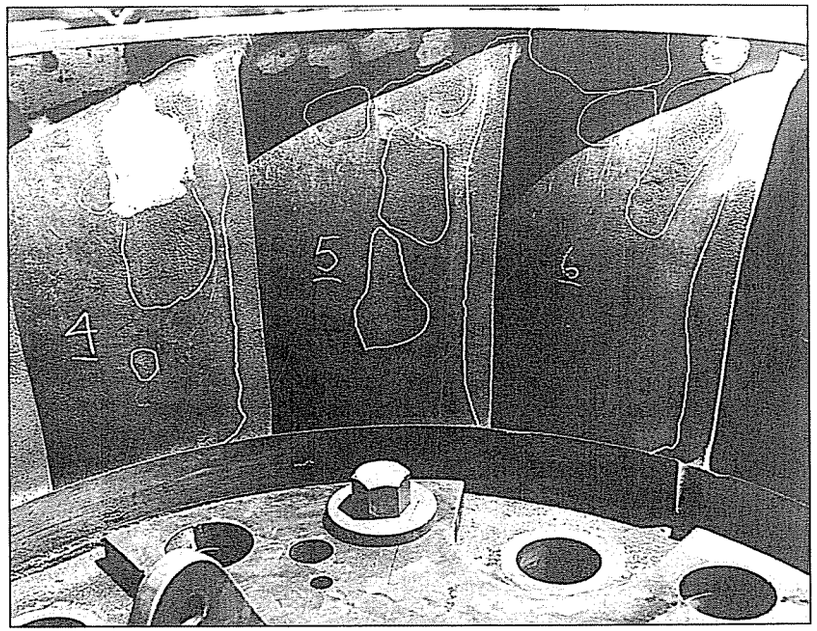


Figure 16.15
Cavitation pitting of Francis wheel and scroll case at Mammoth Pool Powerhouse, California, after $2\frac{1}{2}$ years of operation. Conditions of service were relatively severe. The turbine develops 88,000 hp (65.6 MW) at an effective head of 950 ft (290 m) at 360 rpm. The bright shiny areas are stainless-steel welds that have withstood cavitation pitting for over a year. (Courtesy of Southern California Edison Co.)

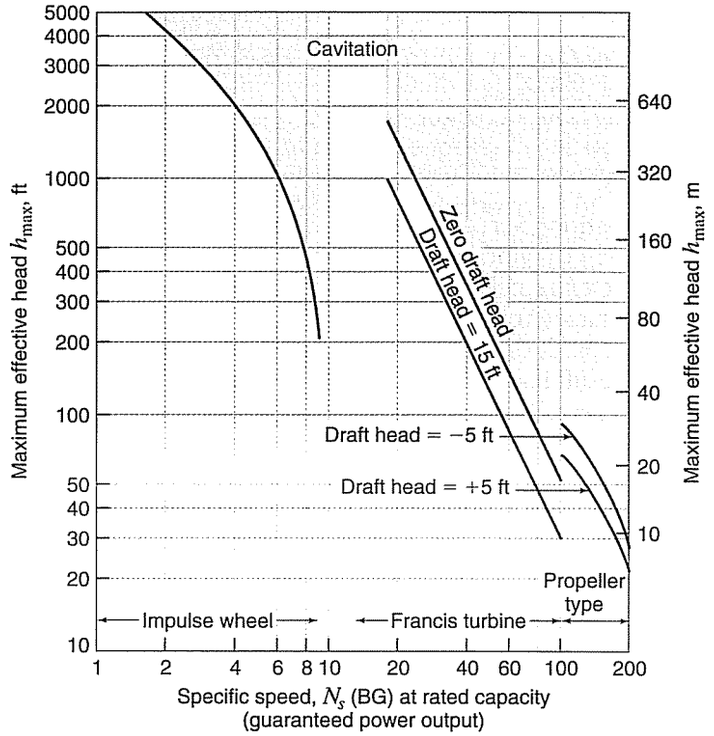


Figure 16.16
Recommended limits of specific speed for turbines under various effective heads at sea level with water temperature at 80°F (27°C). (*After Moody.*)

In reaction turbines the most likely place for the occurrence of cavitation is on the back sides of the runner blades near their trailing edges. Cavitation may be avoided by designing, installing, and operating a turbine in such a manner that at no point will the local absolute pressure drop to the vapor pressure of the water.

In comparing the cavitation characteristic of reaction turbines, it is convenient to define a ***cavitation parameter*** σ (sigma) as

$$\sigma = \frac{p_{atm}/\gamma - p_v/\gamma - z_B}{h} \tag{16.22}$$

where z_B and h are as defined in Fig. 16.11. The term $p_{atm}/\gamma - p_v/\gamma$ represents the height to which water will rise in a water barometer. At sea level with 80°F water, $p_{atm}/\gamma - p_v/\gamma = 32.8$ ft (10.0 m). At higher elevations and at higher water temperatures it is smaller than 32.8 ft (10.0 m).

The critical (or minimum) value σ_c of the cavitation parameter is the value of σ below which cavitation will occur. By inspection of Eq. (16.22), we see that σ tends to get smaller at low values of p_{atm} (high elevations), at high values of p_v (high water temperatures), at large values of draft head z_B (large distances of the turbine setting above tailwater), and at high values of net head h. We can experimentally determine the critical value σ_c for a given turbine by noting the operating conditions under which cavitation first occurs as evidenced by the presence of noise, vibration, or loss of efficiency.

By substituting the critical condition for cavitation ($\sigma = \sigma_c$) into Eq. (16.22), we obtain

$$h_{max} = \frac{p_{atm}/\gamma - p_v/\gamma - z_B}{\sigma_c} \tag{16.23}$$

or

$$(z_B)_{max} = p_{atm}/\gamma - p_v/\gamma - \sigma_c h \tag{16.24}$$

The first of these equations gives the maximum head under which a reaction turbine should operate in order to be safe from cavitation, while the second equation indicates the maximum value of z_B (Fig. 16.11) to assure freedom from cavitation. To be on the safe side, z_B is usually limited to 15 ft (4.5 m). Typical values of σ_c as a function of specific speed are given in Fig. 16.17.

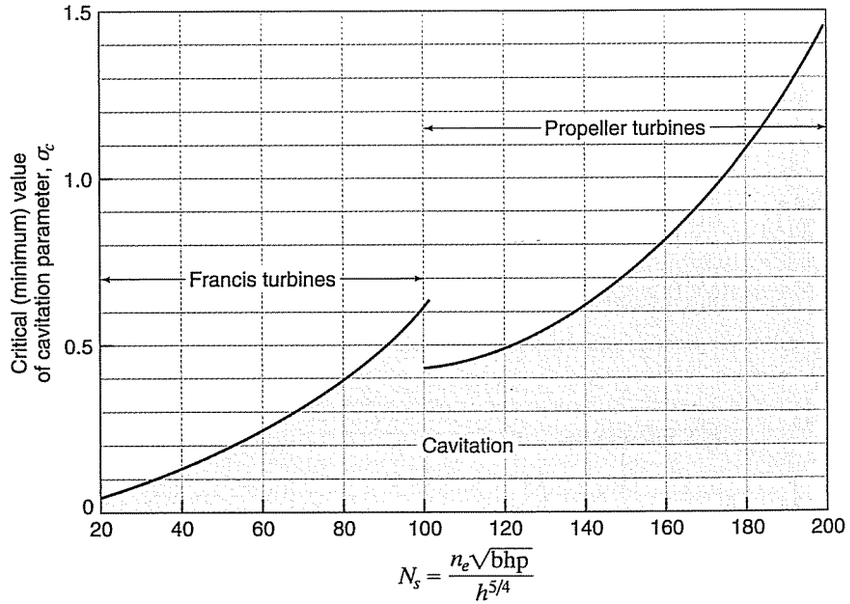


Figure 16.17
Typical values of the critical cavitation parameter σ_c for reaction-type turbines.

SAMPLE PROBLEM 16.12 Find the maximum permissible head under which a Francis turbine ($N_s = 70$) can operate if it is set 10 ft above tailwater at an elevation of 5000 ft with water temperature at 60°F.

Solution

Table A.3 at 5000 ft elevation: $\frac{p_{atm}}{\gamma} = \frac{12.23(144)}{62.4} = 28.22$ ft

Table A.1 for water at 60°F: $\frac{p_v}{\gamma} = 0.59$ ft

Fig. 16.17 for Francis turbine with $N_s = 70$: $\sigma_c = 0.32$

Eq. (16.23): $h_{max} = \frac{28.22 - 0.59 - 10}{0.32} = 55.1$ ft ***ANS***

This is the maximum permissible head to assure against cavitation.

SAMPLE PROBLEM 16.13 Repeat Sample Prob. 16.12, assuming the Francis turbine is at sea level with a water temperature of 80°F. Check the result against Fig. 16.16.

Solution

Table A.3 at sea level: $\frac{p_{atm}}{\gamma} = \frac{14.70(144)}{62.4} = 33.91$ ft

Table A.1 for water at 80°F: $\frac{p_v}{\gamma} = 1.17$ ft

Fig. 16.17 for Francis turbine with $N_s = 70$: $\sigma_c = 0.32$

Eq. (16.23): $h_{max} = \frac{33.91 - 1.17 - 10}{0.32} = 71.1$ ft ***ANS***

This is the maximum permissible head to assure against cavitation. This checks closely with the information given on Fig. 16.16.

EXERCISES

16.12.1 Using Eq. (16.24) and Fig. 16.17 for σ_c calculate the maximum permissible head under which a Francis turbine ($N_s = 40$) can operate if it is set 12 ft above tailwater. The installation is at sea level and the water temperature is 80°F. How does your answer compare to Fig. 16.16?

16.12.2 (*a*) Find the maximum permissible head under which an axial-flow turbine ($N_s = 160$) can operate if it is set 5 ft below tailwater. The installation is at elevation 3150 ft and the water temperature is 65°F. (*b*) What would be the maximum permissible head if the turbine had been set 5 ft above tailwater?

16.12.3 Repeat Exer. 16.12.1 for the case where the turbine is at elevation 10,000 ft rather than at sea level. In this case one cannot compare the result with the curves presented in Fig. 16.16, because of the change in p_{atm}.

16.13 SELECTION OF TURBINES

Inspection of Eq. (16.21) and Fig. 16.14 indicates that at high heads for a given speed and power output, a low-specific-speed machine such as an impulse wheel is required. On the other hand, an axial-flow turbine with a high N_s is indicated for low heads. An impulse turbine may, however, be suitable for a low-head installation if the flow rate (or power requirement) is small, but often under such conditions the efficiency of the impulse wheel is quite low. Impulse wheels have been used for heads as low as 50 ft if the capacity is small, but they are more commonly employed for heads greater than 500 or 1000 ft. The limiting head for Francis turbines is about 1500 ft because of possible cavitation and the difficulty of building casings to withstand such high pressures. By choosing a high speed of operation, and hence a high-specific-speed turbine, the runner size and thus first cost are reduced. However, there is some loss of efficiency at high specific speeds.

In the selection of a turbine or turbines at a given installation, options are available with respect to the number and type (N_s) of turbines. Generally it is considered good practice to have at least two turbines at an installation so that the plant can continue operation while one of the turbines is shut down for repairs or inspection. The head h is determined primarily by topography, and the flow Q by the hydrology of the watershed and characteristics of the reservoir. In selecting a turbine for a given installation, freedom from cavitation must be verified. Some of the factors influencing the choice of turbines are apparent in the following sample problem.

SAMPLE PROBLEM 16.14 Two or more identical turbines are to be selected for an installation where the net head is 350 ft and the total flow is to be 600 cfs. Select turbines for this installation assuming 90% efficiency for all turbines.

Solution

The total available power is

$$\frac{\gamma Q h \eta}{550} = \frac{62.4(600)350(0.90)}{550} = 21{,}400 \text{ hp}$$

Assume two turbines at an operating speed of 75 rpm (96-pole generators for 60-cycle electricity).

Eq. (16.21): $N_s = \frac{75\sqrt{21{,}400/2}}{350^{5/4}} = 5.12$

Thus, if the operating speed is 75 rpm, use two impulse turbines with $N_s = 5.12$, in which case from Fig. 16.14 $\phi_e \approx 0.45$. The required wheel diameter of these

turbines is found from Eq. (16.20):

$$D = \frac{153.3\sqrt{350} \times 0.45}{75} = 17.2 \text{ ft}$$

A wheel diameter of 17.2 ft is quite large; a smaller size is possible by increasing the rotative speed. If $n = 100$ rpm, $N_s = 5.12(100/75) = 6.8$ and $D = 17.2 \times (75/100) = 12.9$ ft. Other combinations of N_s and D could be used with other speeds; however, in accordance with Fig. 16.14, N_s should be less than about 7.0 if impulse wheels are selected. Another possible solution is four identical turbines with $N_s = 5.8$ and $D = 10.7$ ft operating at 120 rpm. ***ANS***

Finally, let us explore the possibility of using Francis turbines. Assume two Francis turbines operating at 600 rpm (12-pole generators for 60-cycle electricity).

Eq. (16.21):
$$N_s = \frac{600\sqrt{21{,}400/2}}{350^{5/4}} = 41$$

According to Fig. 16.16, turbines with $N_s = 41$ and $h = 350$ ft will be safe against cavitation only if they are set one or more feet below tailwater. This may be impractical in some topographic situations. ***ANS***

To provide greater safeguard against cavitation, we might select a lower-specific-speed machine, but then its efficiency may not be so good as indicated by Fig. 16.14. A good choice would be two Francis turbines with $N_s = 30.8$ operating at 450 rpm. The required diameter of these turbines would be about 4.8 ft, assuming $\phi = 0.75$. ***ANS***

Note: There are actually an infinite number of alternatives. The things to watch out for are: (*a*) freedom from cavitation (Fig. 16.16); (*b*) reasonably high efficiency (Fig. 16.14); and (*c*) size not too large (Eq. 16.19). Flexibility of choice is achieved through variation in the number of units (and hence brake horsepower per unit) and in the operating speed. Variation in the draft-head setting also provides some flexibility.

EXERCISES

16.13.1 (*a*) If $Q = 40$ cfs and $h = 70$ ft, would it be possible to use a single impulse wheel with a specific speed less than 7.0 at a site where 60-Hz current is to be generated? Assume a turbine efficiency of 90% and that the generator is to have no more than 96 poles. (*b*) If a single impulse wheel can be used under these conditions, calculate the approximate diameter of the turbine runner for the range of possible speeds of operation.

16.13.2 (*a*) If $Q = 40$ cfs and $h = 70$ ft, would it be possible to use a single Francis turbine with a specific speed of at least 20 at a site where 60-Hz current is to be generated? Assume a turbine efficiency of 90% and assume the generator is to have no less than 12 poles. (*b*) If a single Francis turbine can satisfy these conditions, calculate the approximate diameter of the turbine runner for the range of possible speeds of operation.

16.13.3 Specify the specific speed and the approximate runner diameter at a site where $Q = 2350$ cfs and $h = 480$ ft. Use a 40-pole generator for 60-cycle current. Assume a turbine efficiency of 90%. These are the specifications for some of the turbines at Hoover Dam.

16.13.4 (*a*) What is the maximum head for which a propeller-type turbine can be used if the flow rate is 40 cfs? Assume a turbine efficiency of 90% with specific speeds for propeller-type turbines varying from 100 to 200. Also, assume a 12-pole generator with 50-Hz current. (*b*) Find the maximum head for which a 20-pole generator could have been used with the same turbine efficiency and 50-Hz current. Neglect the possibility of cavitation.

16.14 PUMP TURBINE

More recently the ***pump-turbine hydraulic machine*** was developed. It is very similar in design and construction to the Francis turbine. When water enters the rotor at the periphery and flows inward the machine acts as a turbine. With water entering at the center (or ***eye***) and flowing outward, the machine acts as a pump. The direction of rotation is, of course, opposite in the two cases. The pump turbine is connected to a motor generator, which acts as either a motor or generator depending on the direction of rotation.

The pump turbine is used at pumped-storage hydroelectric plants, which pump water from a lower reservoir to an upper reservoir during off-peak load periods so that water is available to drive the machine as a turbine during the time that peak power generation is needed.

An example of a pump turbine is that at the Kisenyama Pumped Storage Project of the Kansai Electric Company in Japan. There are two identical pump turbines at that installation. Under the normal range of operating conditions each machine has the following characteristics:

As a turbine ($n = 225$ rpm):

Develops 322,000 hp (240 MW) at maximum net head of 722 ft (220 m).
Develops 238,000 hp (177 MW) at minimum net head of 607 ft (185 m).

As a pump ($n = 225$ rpm):

Delivers 3880 cfs (110 m^3/s) at minimum net head of 649 ft (198 m).
Delivers 3040 cfs (86 m^3/s) at maximum net head of 755 ft (230 m).

When operating as a turbine, these machines have a specific speed $N_s = 35$ in the BG system of units. As a pump, in the BG system, these machines have $N_s = 2080$.

At Bad Creek in South Carolina the Duke Power Company has four vertical-shaft pump turbines. When pumping against a head of 1270 ft (378 m) at 300 rpm, the flow rate is 3500 cfs (100 m^3/s). As a turbine each unit develops 480,000 hp (360 MW) under a head of 1204 ft (367 m) at 300 rpm. The corresponding values of specific speed in BG units are 1808 when acting as a pump and 27 when acting as a turbine.

16.15 TURBINE INSTALLATIONS

A few examples of turbine installations are presented here as illustrations of modern practice.

Impulse Turbines

The impulse wheel is especially adapted for use under high heads. For heads above 1500 ft (457 m) it is the only type that can be used. Some interesting installations include the following.

At Dixence, Switzerland, a double-overhung unit operates under a gross head of 5735 ft (1748 m) and a net head of 5330 ft (1625 m), giving a jet velocity of approximately 580 fps (177 m/s). Each single-nozzle wheel delivers 25,000 hp (18.6 MW), or 50,000 hp (37.3 MW) for the unit, and runs at 500 rpm. The jet diameters are 3.71 in (94.2 mm), and the pitch diameter of the wheels in 10.89 ft (3.319 m). Also in Switzerland, the Fully plant in Valais operates under a gross head of 5410 ft (1649 m) and a net head of 4830 ft (1472 m). There are four wheels of 3000 hp (2.24 MW) each running at 500 rpm. The jet diameters are 1.5 in (38 mm) and the wheel diameters are 11.67 ft (3.557 m).

A multiple-nozzle wheel at Pragnières, France, delivers 100,000 hp (75 MW) at 428 rpm under a gross head of 3920 ft (1195 m).

In North America there are a number of plants operating under heads of between 2000 and 2500 ft (610 and 760 m). At the Big Creek-2A plant of the Southern California Edison Co. is a double-overhung single-jet impulse turbine of 137-in (3.48-m) diameter with an 8.4-in (213-mm) jet on each wheel and a similar unit of 135.25-in (3.435-m) diameter with one 8.5-in (216-mm) jet on each wheel. Each unit runs at 300 rpm, and the maximum static head with a full reservoir is 2418 ft (737 m). With both units carrying full load, the net head is 2200 ft (670 m), and the maximum output is 65,100 bhp (48.5 MW) for the first unit and 67,300 bhp (50.2 MW) for the second. At the Balch plant of the Pacific Gas and Electric Co., is a double-overhung unit with 115-in (2.92-m)-diameter wheels and one 7.5-in (191 mm) jet on each wheel. Its runs at 360 rpm and develops 49,000 hp (36.5 MW) under a net head of 2243 ft (684 m). The Colgate plant in California has two vertical-shaft impulse wheels, each with six nozzles operating under a head of 1355 ft (413 m) at 180 rpm and delivering 235,000 hp (175 MW).

The Kitimat plant in British Columbia of the Aluminum Company of Canada has three four-nozzle vertical-shaft units that operate at 327 rpm under a net head of 2500 ft (762 m). Each unit is rated at 140,000 hp (104 MW).

One of the highest-head plants in South America is at Los Molles in Chile, where an impulse wheel develops 11,500 hp (8.58 MW) at 1000 rpm under a head of 3460 ft (1055 m). At Cementos El Cairo in Colombia are two single-overhung impulse turbines operating under a head of 1900 ft (579 m). Each unit develops 3000 hp (2.24 MW) at 900 rpm.

In Peru at the Paucartambo plant of Cerro de Pasco Corp., there is a single-overhung double-jet impulse wheel rated at 28,000 hp (21 MW) under a net head of 1580 ft (482 m) operating at 450 rpm.

Francis Turbines

The Francis turbine is especially adapted for heads ranging from 50 to 1000 ft (15 to 300 m), and has been used on heads as high as 1500 ft (460 m).

Among some interesting installations of Francis turbines are the following. At Fionnay, Switzerland, a Francis turbine operates under a head of 1490 ft (454 m) and delivers 63,200 hp (47.1 MW) at 750 rpm. In Austria a head of 1430 ft (436 m) is used for a 77,700-hp (57.9-MW) turbine at 500 rpm. In Norway a head of 1360 ft (415 m) is used to develop 69,000 hp (51.5 MW) at 500 rpm. In Italy a turbine running at 1000 rpm develops 20,140 hp (15.02 MW) under a head of 1320 ft (402 m). At Oak Grove, Oregon, a Francis turbine under a head of 850 ft (259 m) delivers 35,000 hp (26 MW) at 514 rpm. These are all low-specific-speed reaction turbines.

Examples of higher specific speeds are a Francis turbine in France, which delivers 154,000 hp (115 MW) at 187.5 rpm under a head of 336 ft (102 m) and another one, also in France, which runs under a head of 233 ft (71 m) and develops 135,000 hp (101 MW) at 150 rpm. At Conowingo on the Susquehanna River 54,000-hp (40-MW) units run at 81.8 rpm under a head of 89 ft (27 m). The Conowingo runners are 18 ft (5.5 m) in diameter. Other examples of large reaction turbines are the four vertical-shaft Francis turbines at the Kootenay Canal Power Station in British Columbia, Canada. These are each designed to develop 196,000 hp (146 MW) when operating at 128.6 rpm under a net head of 268 ft (81.7 m). At Grand Coulee Dam on the Columbia River in the State of Washington there are six large Francis turbines ($N_s = 55$). Each unit develops over 800,000 hp (600 MW) under a head of 285 ft (87 m) at 72 rpm. Each runner has a diameter of 32 ft (9.75 m) and weighs 550 tons (500 metric tons). Water is delivered to each unit at a rate as high as 30,000 cfs (850 m^3/s) in a 40-ft- (12.2-m-) diameter pipeline. The turbines operate at heads ranging from 220 to 355 ft (67 to 108 m).

A unique installation is the Itaipu hydroelectric plant on the Parana River in South America. This is a joint project between Brazil and Paraguay. There are 18 turbine units operating under a head of about 400 ft (122 m). Nine of the units develop 50-cycle electricity using 66-pole generators (90.9 rpm), while the other nine units develop 60-cycle electricity using 78-pole generators. Another interesting installation is at the Trinity River Power Plant in California. The turbine is designed to permit the use of either of two interchangeable runners. The high-head runner ($N_s = 30$) is rated at 85,000 hp (63 MW) when operating at 200 rpm under a head of 426 ft (130 m). The low-head runner is rated at 70,000 hp (52 MW) when operating at 200 rpm under a head of 334 ft (102 m).

Propeller Turbines

Propeller turbines, both fixed-blade and adjustable-blade (Kaplan turbine), are best suited for low-head installations, usually for heads less than 100 ft (30 m). A large fixed-blade turbine at the St. Lawrence Power Dam operates under a head of 81 ft (24.7 m) at 94.7 rpm and delivers 79,000 hp (59 MW). At Wheeler Dam in Alabama a fixed-blade propeller unit delivers 45,000 hp (34 MW) at 85.7 rpm

under a head of 48 ft (14.6 m). Kaplan turbines are represented by two units at Camargos in Brazil. Each unit develops 35,700 hp (26.6 MW) under a head of 88.6 ft (27.0 m) at 150 rpm. At Bonneville, on the Columbia River, are units that have 280-in (7.11-m)-diameter runners. They deliver 66,000 hp (49 MW) at 75 rpm under 50 ft (15.2 m) of head. A typical tube turbine would be the one at Ludwigswehr, Germany, where a four-blade runner with movable blades on a horizontal axis develops 1050 hp (0.78 MW) under a head of 15 ft (4.5 m) at 125.5 rpm. Another example of a tube turbine is at Hausen, Germany, where a four-blade runner on an inclined axis develops 1260 hp (0.94 MW) at 200 rpm under a head of 16 ft (5.0 m).

PROBLEMS

16.1 Repeat Sample Prob. 16.1 for rotative speeds of 80, 160, 300, and 360 rpm, with all other data remaining the same. Also find the hydraulic efficiency of the flow through the buckets for each speed. Plot horsepower as ordinate versus rotative speed as abcissa to find the approximate speed at which maximum power will be developed. At maximum power, approximately what is the value of u/V_1?

16.2 Same as Prob. 16.1, except also consider hydraulic friction in the buckets by letting $v_2 = 0.85v_1$.

16.3 Consider an impulse wheel with a pitch diameter of 2.65 m and a bucket angle of 168°. If the jet velocity is 55.0 m/s, the jet diameter is 100 mm, and the rotative speed is 300 rpm, find (*a*) the force on the buckets, (*b*) the torque on the runner, and (*c*) the power transferred to the runner. Assume $v_2 = 0.85v_1$.

16.4 An impulse turbine with a pitch diameter of 4.0 ft and a bucket angle of 160° is rotating at 514.7 rpm. The nozzle produces a jet with a velocity of 240 ft/sec and diameter of 4 in. Find (*a*) the force on the buckets, (*b*) the torque on the runner, (*c*) the horsepower transferred to the runner, and (*d*) the efficiency η of the turbine under these conditions of operation. Assume the head loss through the nozzle can be expressed as $0.05V_j^2/2g$ and assume the velocity relative to the bucket does not change (i.e., assume $v_2 = v_1$). Neglect bearing friction and windage.

16.5 Work Prob. 16.4 where the speed of rotation is 300 rpm, all other data remaining the same.

16.6 (*a*) Solve Sample Prob. 16.1 for blade angles of 160°, 165°, and 170° for the case of an impulse turbine whose pitch diameter is 8 ft rather than 10 ft. All other data are to remain the same. (*b*) Repeat the above for the case where $v_2 = 0.80v_1$.

16.7 Repeat Sample Prob. 16.4 for the case of a 6-in-diameter jet, with all other data remaining the same.

16.8 A torque of 157,600 ft·lb is transmitted to a turbine that is developing 6000 bhp. How many poles does the generator have if it is producing 50-cycle electricity?

16.9 A 1.5-in-diameter water jet with a velocity of 120 ft/sec impinges on a bucket of a stationary impulse wheel with a pitch diameter of 8.0 ft. If the bucket blade angle is 165°, what torque must be applied to the turbine shaft to prevent it from rotating? Consider two cases: (*a*) $v_2 = v_1$, (*b*) $v_2 = 0.9v_1$.

16.10 Water is delivered from a reservoir through a 1200-mm-diameter pipe ($f = 0.022$) 3200 m long to a nozzle that emits a 200-mm-diameter jet (Fig. P16.10). The jet impinges on the buckets of an impulse turbine, pitch diameter 2.5 m, which drives a 16-pole generator that produces 50-Hz electricity. The surface of the reservoir providing the water is at an elevation 800 m higher than the nozzle. Assume that the head loss through the nozzle can be expressed as $0.05V_j^2/2g$ where V_j is the jet velocity. Find the following: velocity of flow in the pipe, jet velocity, flow rate, speed of the buckets, value of u/V_1, net head, force F on the bucket, torque on the wheel, shaft power, power at the base of the nozzle, and overall turbine efficiency. Assume $\beta_2 = 165°$ and $v_2 = v_1$; neglect bearing friction and windage.

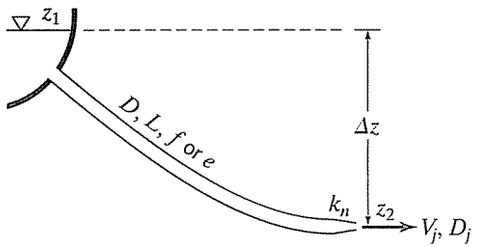


Figure P16.10

16.11 The diameter of a Pelton wheel is 6 ft. The velocity of the 4-in-diameter jet is 400 ft/sec and the bucket angle is 165°. Assuming a k value of 0.21, for bucket speeds of 100, 200, 300, and 400 ft/sec find the torque in ft·lb, the shaft horsepower, and the hydraulic efficiency of the buckets. Neglect bearing friction and windage.

16.12 An impulse turbine is supplied by a 1200-mm-diameter steel penstock ($e = 0.10$ mm) 500 m long (Fig. P16.10). The jet diameter of the single nozzle is 400 mm and its loss coefficient is 0.05. When the water level in the reservoir is 150 m above the nozzle of the turbine, the brake power is measured to be 7900 kW. What is the turbine efficiency?

16.13 Using the data of Prob. 16.11, find values of (*a*) head loss in bucket friction, (*b*) velocity head at discharge, and (*c*) total head loss for bucket speeds of 180, 190, 200, and 210 fps. At what bucket speed is the discharge loss a minimum? At what bucket speed is the total head loss a minimum?

16.14 A 36-in pipeline ($f = 0.020$) 10,000 ft long connects a reservoir whose water-surface elevation is 1800 ft to a nozzle at elevation 1000 ft (Fig. P16.10). The jet from the nozzle is used to drive an impulse wheel. (*a*) If the head loss through the nozzle is expressible as $0.04V_j^2/2g$, determine the horsepower that will be developed by jets having diameters of 8 in, 10 in, and 12 in. (*b*) On the basis of the answers to (*a*), approximately what would be the horsepower and diameter of the jet that gives the maximum horsepower?

16.15 Using the data of Prob. 16.14, express the jet power analytically in terms of D_j and V_j, differentiate, and equate to zero to find the value of D_j for which the jet power is a maximum. Evaluate the value of the maximum jet power. *Hint:* Express power as $P = f(x)$, where $x = D_j^2$.

16.16 In the test of a Francis turbine, a pressure of 140 kPa was measured at the flange at the entrance to a spiral turbine-case where the diameter is 600 mm. Neglecting the small velocity head in the tailrace, find the net head on the turbine if the flow rate was 2.5 m³/s and the flange was 3.0 m above the tailrace.

16.17 A reaction turbine is supplied with water through a 1500-mm-diameter pipe ($e = 0.10$ mm) that is 50 m long. The water surface in the reservoir is 27 m above the draft-tube inlet that is 4.1 m above the water level in the tailrace (Fig. P16.17). If the turbine efficiency is 92% and the discharge is 12 m³/s, what is the power output of the turbine in kilowatts? By how much must the discharge be increased to increase power production by 500 kW? Assume the velocity in the tailrace can be neglected.

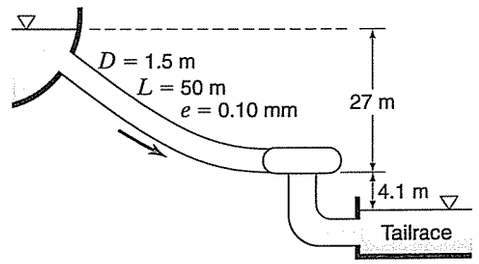


Figure P16.17

16.18 A draft tube leading from the discharge side of a turbine to the submerged discharge in the tailrace as in Fig. 16.11 consists of a pipe ($f = 0.025$) of constant diameter 0.8 m and length 10.0 m. The flow rate is 9.2 m³/s. If this draft tube were to be replaced by a diverging tube 12.0 m long whose diameter changed uniformly from 0.8 m to 2.5 m over the 12.0 m length (Fig. P16.18), how much additional head would be developed by the replacement draft tube? Neglect head loss due to curvature in the bend.

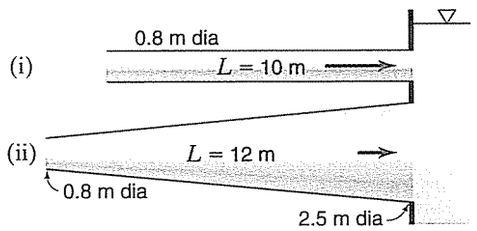


Figure P16.18

16.19 At its upper end a draft tube has a diameter of 24.5 in where it joins the discharge side of the turbine at a point 11.0 ft above the surface of the water in the tailrace. The discharge end of the draft tube has a diameter of 42 in and the velocity in the tailrace is negligible. The total head loss in the draft tube is $0.15V_2^2/2g$ plus the submerged discharge loss of $V_3^2/2g$, where subscripts 2 and 3 refer to the upper and lower ends of the draft tube, respectively. (*a*) When the flow is 38.8 cfs, what is the pressure at the upper end of the draft tube? (*b*) Suppose the draft tube were of uniform diameter and the flow rate does not change; what then would be the pressure at the upper end of the tube? (*c*) How much head is saved by the diverging tube? Assume the draft tube has a length of 18 ft and $f = 0.020$.

16.20 A model turbine (one-twentieth of prototype size) has a maximum hydraulic efficiency of 86.2%. Estimate the efficiency of the prototype utilizing the Moody step-up formula.

16.21 A turbine runs at 150 rpm, discharges 200 cfs, and develops 1600 bhp under a net head of 81 ft. (*a*) What is its efficiency? (*b*) What would be the revolutions per minute, Q, and brake horsepower of the same turbine under a net head of 162 ft for homologous conditions?

16.22 If a turbine homologous to that of Prob. 16.21 has a runner of twice the diameter, what would be the revolutions per minute, Q, and brake horsepower under the same head of 81 ft?

16.23 A 2.6-m-diameter reaction turbine is to be operated at 200 rpm under a net head of 30 m. A 1:10 model of this turbine is built and tested in the laboratory. If the model is operated at 600 rpm, under what net head should it be tested to simulate normal operating conditions in the prototype?

16.24 A small Francis turbine ($N_s = 30$, $D = 2$ ft) is tested and found to have an efficiency of 0.893 when operating under optimum conditions. Approximately what would be the maximum efficiency of a homologous runner ($N_s = 30$) with a diameter of 6 ft?

16.25 A 12-ft-diameter reaction turbine is to be operated at 100 rpm under a net head of 96 ft. A 1:8 model of this turbine is built and tested in the laboratory. If the model is operated at 450 rpm, under what net head should it be tested to simulate normal operating conditions in the prototype?

16.26 A 1:8 model of a 12-ft-diameter turbine is operated at 600 rpm under a net head of 54.0 ft. Under this mode of operation the bhp and Q of the model were observed to be 332 hp and 62 cfs, respectively. From the above data compute (*a*) the specific speed of the model and the value of ϕ_e, (*b*) the efficiency and shaft torque of the model, (*c*) the efficiency of the prototype, and (*d*) the flow rate and horsepower of the prototype if it is operated at 450 rpm under a net head of 200 ft.

16.27 Calculate the specific speed of the turbine whose runner is shown in Fig. 16.2 and estimate the runner diameter.

16.28 Find the specific speed of the turbine whose runner is shown in Fig. 16.7 and estimate the runner diameter.

16.29 Calculate the specific speed of the turbine shown in Fig. 16.9 and estimate the runner diameter.

16.30 Find the specific speed of the propeller turbine mentioned in Exer. 16.7.1. Express the answer in BG units.

16.31 Find the specific speed of a turbine that runs at a maximum efficiency of 90% at 300 rpm under a net head of 81 ft with a flow rate of 50 cfs. Estimate the runner diameter.

16.32 In Sec. 16.15 a double-overhung impulse turbine at Dixence, Switzerland is mentioned. Each wheel of the two-wheel unit develops 25,000 hp at 500 rpm under a head of 5330 ft. (*a*) What is the specific speed of these wheels? (*b*) Estimate their diameter and compare your answer with their actual diameter of 10.89 ft.

16.33 In Sec. 16.15 a Francis turbine on the Susquehanna River is mentioned. This turbine develops 54,000 hp at 81.8 rpm under a head of 89 ft. (*a*) What is the specific speed of this turbine? (*b*) Estimate the runner diameter and compare your answer with the actual diameter of 18 ft.

16.34 A Kaplan turbine at Rock River, Illinois develops 800 hp at 80 rpm under a head of 7 ft. (*a*) What is the specific speed of this turbine? (*b*) Estimate the runner diameter and compare your answer with the actual diameter of 136 in.

16.35 Consider the case of a Francis turbine having a specific speed of 40 that is set 10 ft above tailwater elevation. Perform calculations using Fig. 16.17 and Eq. (16.22) to find the maximum permissible head under which this turbine should operate in order to be safe from cavitation. Check your calculated result against the information shown on Fig. 16.16. Do they agree?

16.36 Consider the case of a propeller turbine having a specific speed of 150 that is set 5 ft below tailwater. Perform calculations using Fig. 16.17 and Eq. (16.22) to find the maximum permissible head under which this turbine should operate in order to be safe from cavitation. Check your calculated result against the information shown in Fig. 16.16. Do they agree?

16.37 At its maximum efficiency of 93% a turbine delivers 3000 hp to the shaft under a head of 72 ft when operating at 300 rpm. For water at 80°F, find the following: (*a*) the flow rate through the turbine; (*b*) the specific speed of the turbine; (*c*) the approximate diameter of the turbine runner; (*d*) how high above tailwater the turbine should be set to be safe against cavitation if it is installed at sea level.

16.38 A propeller turbine operates at a maximum efficiency of 92% under a head of 30 ft at 450 rpm and develops a shaft power of 620 hp. For water at 50°F, find (*a*) the flow rate through the turbine, (*b*) the specific speed of the turbine, (*c*) the approximate diameter of the turbine runner, and (*d*) how far above tailwater elevation the turbine can be set and still be safe from cavitation. Assume the turbine is at an elevation of 100 ft.

16.39 Repeat part (*d*) of Prob. 16.38 for the case where the turbine is installed at elevation 5000 ft.

16.40 A turbine whose specific speed is 80 is to operate under a head of 50 ft at an elevation where the atmospheric pressure is 12.8 psia. The water temperature is 50°F (*a*) If this turbine is set 6 ft above tailwater, will it be safe from cavitation? (*b*) What is the highest permissible elevation of this turbine with respect to tailwater when operating under a head of 60 ft?

16.41 (*a*) A turbine is to be installed at a point where the available head is 175 ft and the available flow will average 1000 cfs. What type of turbine would you recommend? Specify the operating speed and the number of generator poles for 60-cycle electricity if a turbine is selected with the highest specific speed that is safe against cavitation. Assume a draft head of 10 ft and 90% turbine efficiency. Approximately what size of turbine runner is required? (*b*) For the same conditions, select a set of two identical turbines to be operated in parallel. Specify the speed and size of the units.

16.42 What is the least number of identical turbines that can be used at a powerhouse where the available head is 1200 ft and $Q = 1650$ cfs? Assume turbine efficiency is 90% and speed of operation is 138.5 rpm. Specify the size and specific speed of the units.

16.43 A six-jet impulse turbine operating at 300 rpm develops 60,000 hp under a net head of 1060 ft. The runner has a diameter of 6.0 ft. (*a*) How large a homologous runner would be needed for a single-jet machine operating under the same head and developing the same horsepower? (*b*) Check to confirm that this turbine will be free of cavitation.

16.44 An impulse turbine ($N_s \approx 4$) develops 100,000 hp under a head of 2000 ft. (*a*) For 60-cycle electricity calculate the turbine speed (rpm), wheel diameter (ft), and number of poles in the generator. (*b*) Solve the problem for a six-nozzle unit using the same N_s, bhp, and head. In both instances assume $\phi_e = 0.45$. Neglect cavitation.

16.45 A multinozzle impulse turbine is to be designed to develop 60,000 hp at 300 rpm under a head of 1200 ft. How many nozzles should this turbine have? Specify the approximate wheel diameter for this design. How large would a single-jet machine have to be to satisfy these requirements?

16.46 It is desired to develop 15,000 bhp under a head of 1000 ft. Make any necessary assumptions and estimate the diameter of the wheel required and the rotative speed.

16.47 A single hydraulic turbine is to be selected for a power site with a net head of 100 ft. The turbine is to produce 25,000 hp at maximum efficiency. What speed (rpm) and diameter should this turbine have if (*a*) a Francis turbine is selected; (*b*) a propeller turbine is selected? What are the highest "settings" (above or below tailwater) that should be recommended for each of these machines for them to run cavitation-free at their points of maximum efficiency?

16.48 For 50-cycle electricity how many poles would you recommend for a generator that is connected to a turbine operating under a design head of 3000 ft with a flow of 80 cfs? Assume turbine efficiencies as given in Fig. 16.14 and be sure the turbine is free of cavitation.

16.49 (*a*) Specify the type, speed, and size of a single turbine to be installed at a site with an effective head of 48 m, a maximum draft head of 2 m, and a flow rate of 5 m^3/s. (*b*) How would your recommendation change if the available flow was 50 m^3/s?

16.50 It is desired to develop 300,000 hp under a head of 49 ft and to operate at 60 rpm. (*a*) If turbines with a specific speed of approximately 150 are to be used, how many units would be required? (*b*) If Francis turbines with a specific speed of 80 were to be used, how many units would be required?

16.51 Select two, four, and six identical turbines for an installation where $h = 400$ ft and total $Q = 300$ cfs. Develop 60-cycle electricity using either 36- or 72-pole generators. Be sure your selection is free of cavitation. Assume the turbine efficiency is 90%

16.52 A turbine is to be installed at a point where the net available head is 35 m and the available flow will average 23 m^3/s. What type of turbine would you recommend? Specify the operating speed and number of generator poles for 60-cycle electricity if a turbine is selected with the highest specific speed that is safe against cavitation. Assume a draft head of 3 m and 90% turbine efficiency. Approximately what size of turbine runner is required?

16.53 A turbine is to be installed where the net available head is 185 ft, and the available flow will average 900 cfs. What type of turbine would you recommend? Specify the operating speed and number of generator poles for 60-cycle electricity if a turbine is selected with the highest specific speed that is safe against cavitation. Assume the turbine is set 5 ft above tailwater, and its efficiency is 90%. Approximately what size of runner is required?

Appendix A
Fluid and Geometric Properties

Figure ***Contents***

A.1 Absolute viscosity μ of fluids

A.2 Kinematic viscosity ν of fluids

Table ***Contents***

A.1 Physical properties of water at standard sea-level atmospheric pressure

A.2 Physical properties of air at standard sea-level atmospheric pressure

A.3 The ICAO standard atmosphere

A.4 Physical properties of common liquids at standard sea-level atmospheric pressure

A.5 Physical properties of common gases at standard sea-level atmospheric pressure

A.6 Areas of circles

A.7 Properties of areas

A.8 Properties of solid bodies

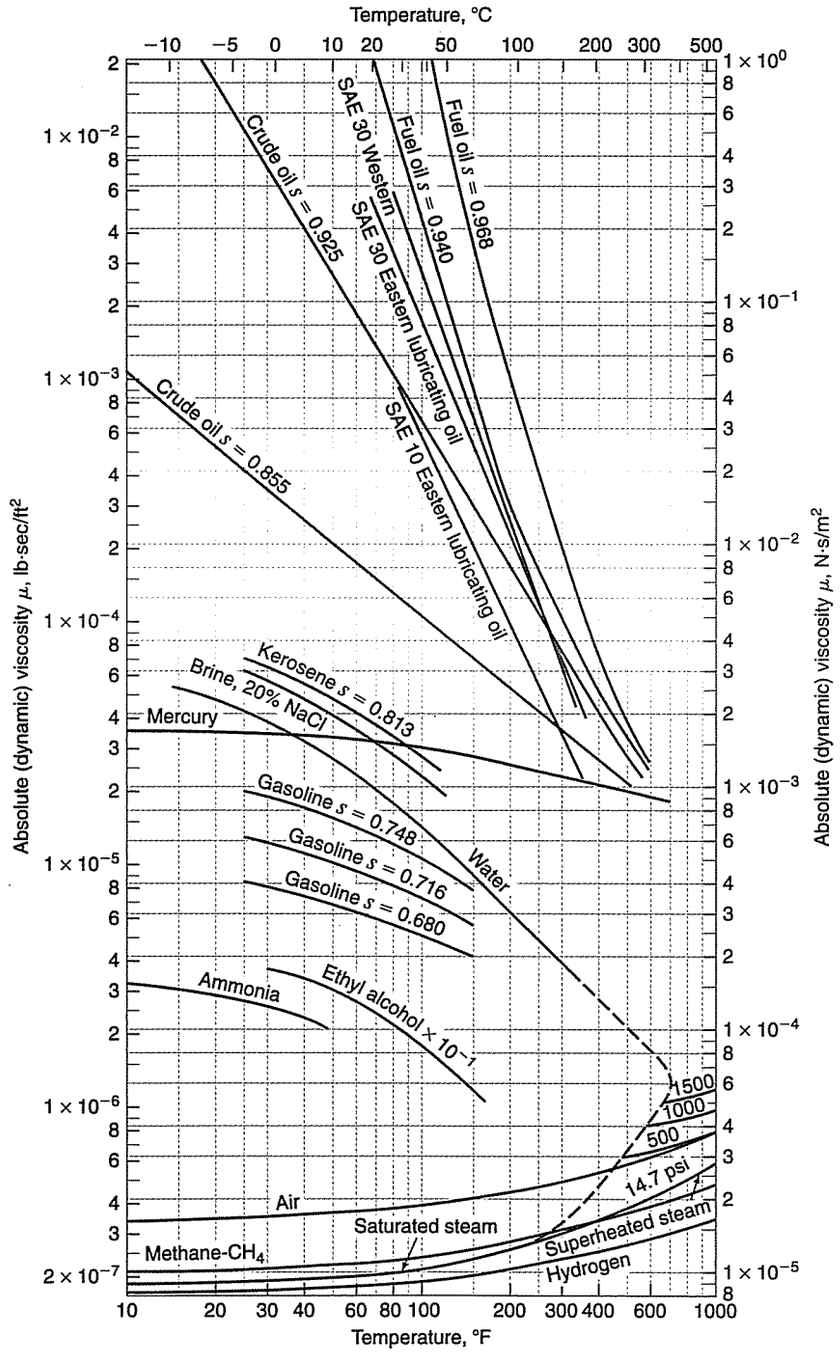


Figure A.1
Absolute viscosity μ of fluids. (s = specific gravity at 60°F relative to water at 60°F. For water and air see also Tables A.1 and A.2, respectively.)

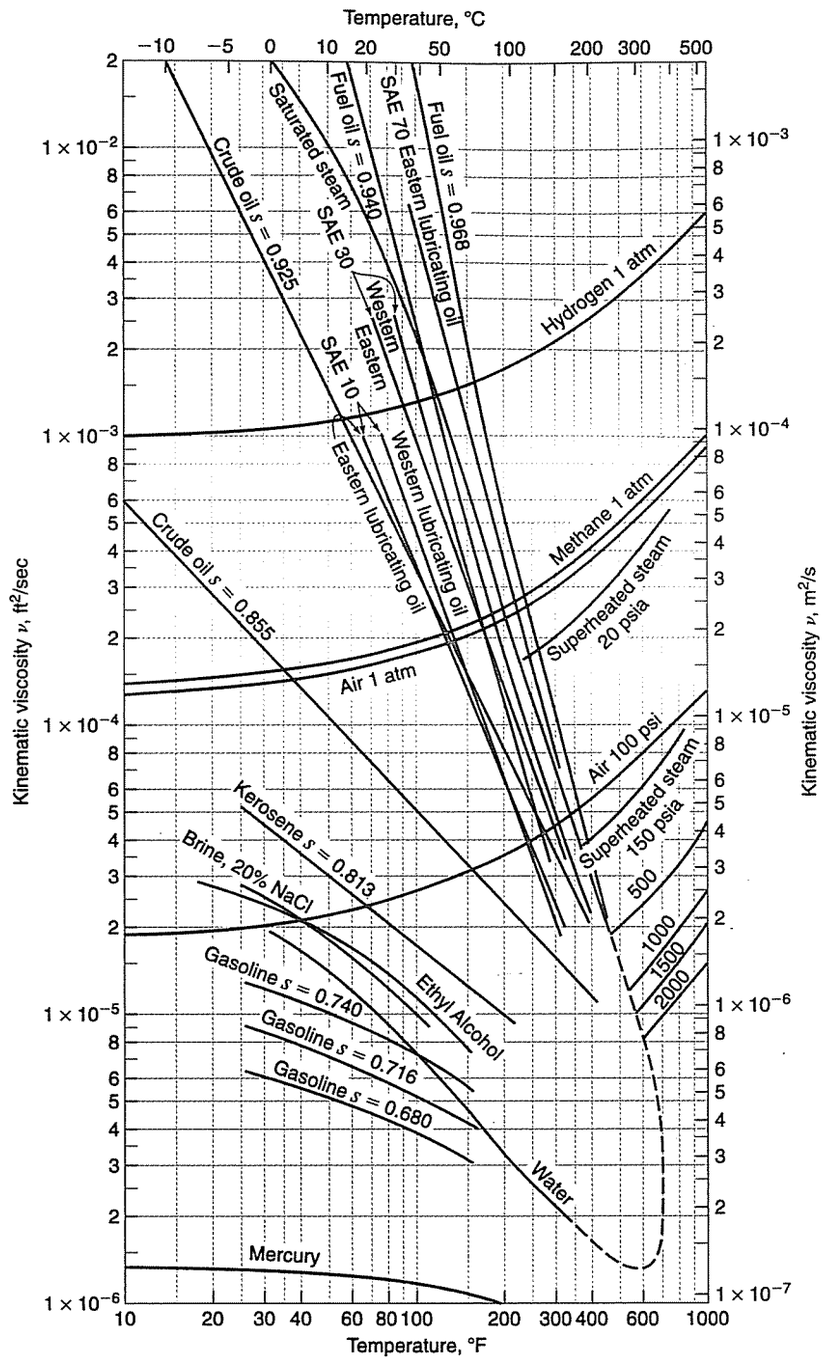


Figure A.2
Kinematic viscosity ν of fluids. (s = specific gravity at 60°F relative to water at 60°F. For water and air see also Tables A.1 and A.2, respectively.)

TABLE A.1 **Physical properties of water at standard sea-level atmospheric pressure**[a]

Temperature, T	Specific weight, γ	Density, ρ	Absolute viscosity,[b] μ	Kinematic viscosity,[b] ν	Surface tension, σ	Saturation vapor pressure, p_v	Satur'n vapor pressure head, p_v/γ	Bulk modulus of elasticity, E_v
°F	lb/ft³	slugs/ft³	10^{-6} lb·sec/ft²	10^{-6} ft²/sec	lb/ft	psia	ft abs	psi
32°F	62.42	1.940	37.46	19.31	0.00518	0.0885	0.204	293,000
40°F	62.43	1.940	32.29	16.64	0.00514	0.122	0.281	294,000
50°F	62.41	1.940	27.35	14.10	0.00509	0.178	0.411	305,000
60°F	62.37	1.938	23.59	12.17	0.00504	0.256	0.592	311,000
70°F	62.30	1.936	20.50	10.59	0.00498	0.363	0.839	320,000
80°F	62.22	1.934	17.99	9.30	0.00492	0.507	1.173	322,000
90°F	62.11	1.931	15.95	8.26	0.00486	0.698	1.618	323,000
100°F	62.00	1.927	14.24	7.39	0.00480	0.949	2.20	327,000
110°F	61.86	1.923	12.84	6.67	0.00473	1.275	2.97	331,000
120°F	61.71	1.918	11.68	6.09	0.00467	1.692	3.95	333,000
130°F	61.55	1.913	10.69	5.58	0.00460	2.22	5.19	334,000
140°F	61.38	1.908	9.81	5.14	0.00454	2.89	6.78	330,000
150°F	61.20	1.902	9.05	4.76	0.00447	3.72	8.75	328,000
160°F	61.00	1.896	8.38	4.42	0.00441	4.74	11.18	326,000
170°F	60.80	1.890	7.80	4.13	0.00434	5.99	14.19	322,000
180°F	60.58	1.883	7.26	3.85	0.00427	7.51	17.84	318,000
190°F	60.36	1.876	6.78	3.62	0.00420	9.34	22.28	313,000
200°F	60.12	1.868	6.37	3.41	0.00413	11.52	27.59	308,000
212°F	59.83	1.860	5.93	3.19	0.00404	14.69	35.36	300,000
°C	kN/m³	kg/m³	N·s/m²	10^{-6} m²/s	N/m	kN/m² abs	m abs	10^6 kN/m²
0°C	9.805	999.8	0.001781	1.785	0.0756	0.611	0.0623	2.02
5°C	9.807	1000.0	0.001518	1.519	0.0749	0.872	0.0889	2.06
10°C	9.804	999.7	0.001307	1.306	0.0742	1.230	0.1255	2.10
15°C	9.798	999.1	0.001139	1.139	0.0735	1.710	0.1745	2.14
20°C	9.789	998.2	0.001002	1.003	0.0728	2.34	0.239	2.18
25°C	9.777	997.0	0.000890	0.893	0.0720	3.17	0.324	2.22
30°C	9.765	995.7	0.000798	0.800	0.0712	4.24	0.434	2.25
40°C	9.731	992.2	0.000653	0.658	0.0696	7.38	0.758	2.28
50°C	9.690	988.0	0.000547	0.553	0.0679	12.33	1.272	2.29
60°C	9.642	983.2	0.000466	0.474	0.0662	19.92	2.07	2.28
70°C	9.589	977.8	0.000404	0.413	0.0644	31.16	3.25	2.25
80°C	9.530	971.8	0.000354	0.364	0.0626	47.34	4.97	2.20
90°C	9.467	965.3	0.000315	0.326	0.0608	70.10	7.40	2.14
100°C	9.399	958.4	0.000282	0.294	0.0589	101.33	10.78	2.07

[a] In these tables, if (for example, at 32°F) μ is given as 37.46 and the units are 10^{-6} lb·sec/ft² then $\mu = 37.46 \times 10^{-6}$ lb·sec/ft².
[b] For viscosity, see also Figs. A.1 and A.2.

TABLE A.2 **Physical properties of air at standard sea-level atmospheric pressure**[a]

Temperature, T	Density, ρ	Specific weight, γ	Absolute viscosity,[b] μ	Kinematic viscosity,[b] ν
°F	slug/ft³	lb/ft³	10^{-6} lb·sec/ft²	10^{-3} ft²/sec
−40°F	0.002940	0.09460	0.312	0.106
−20°F	0.002807	0.09030	0.325	0.116
0°F	0.002684	0.08637	0.338	0.126
10°F	0.002627	0.08453	0.345	0.131
20°F	0.002572	0.08277	0.350	0.136
30°F	0.002520	0.08108	0.358	0.142
40°F	0.002470	0.07945	0.362	0.146
50°F	0.002421	0.07790	0.368	0.152
60°F	0.002374	0.07640	0.374	0.158
70°F	0.002330	0.07495	0.382	0.164
80°F	0.002286	0.07357	0.385	0.169
90°F	0.002245	0.07223	0.390	0.174
100°F	0.002205	0.07094	0.396	0.180
120°F	0.002129	0.06849	0.407	0.189
140°F	0.002058	0.06620	0.414	0.201
160°F	0.001991	0.06407	0.422	0.212
180°F	0.001929	0.06206	0.434	0.225
200°F	0.001871	0.06018	0.449	0.240
250°F	0.001739	0.05594	0.487	0.280
°C	kg/m³	N/m³	10^{-6} N·s/m²	10^{-6} m²/s
−40°C	1.515	14.86	14.9	9.8
−20°C	1.395	13.68	16.1	11.5
0°C	1.293	12.68	17.1	13.2
10°C	1.248	12.24	17.6	14.1
20°C	1.205	11.82	18.1	15.0
30°C	1.165	11.43	18.6	16.0
40°C	1.128	11.06	19.0	16.8
60°C	1.060	10.40	20.0	18.7
80°C	1.000	9.81	20.9	20.9
100°C	0.946	9.28	21.8	23.1
200°C	0.747	7.33	25.8	34.5

[a] In these tables, if (for example, at −40°F) μ is given as 0.312 and the units are 10^{-6} lb·sec/ft² then $\mu = 0.312 \times 10^{-6}$ lb·sec/ft².
[b] For viscosity, see also Figs. A.1 and A.2. Absolute viscosity μ is virtually independent of pressure, whereas kinematic viscosity ν varies with pressure (density) (Sec. 2.11).

TABLE A.3 **The ICAO[a] standard atmosphere[b]**

Elevation above sea level	Temperature, T	Absolute pressure, p	Specific weight, γ	Density, ρ	Absolute viscosity, μ	Kinematic viscosity, ν	Speed of sound, c	Gravitational acceleration, g
ft	°F	psia	lb/ft³	slug/ft³	10^{-6} lb·sec/ft²	10^{-3} ft²/sec	ft/sec	ft/sec²
0	59.000	14.6959	0.076472	0.0023768	0.37372	0.15724	1116.45	32.1740
5,000	41.173	12.2283	0.065864	0.0020481	0.36366	0.17756	1097.08	32.158
10,000	23.355	10.1083	0.056424	0.0017555	0.35343	0.20133	1077.40	32.142
15,000	5.545	8.2970	0.048068	0.0014961	0.34302	0.22928	1057.35	32.129
20,000	−12.255	6.7588	0.040694	0.0012672	0.33244	0.26234	1036.94	32.113
25,000	−30.048	5.4607	0.034224	0.0010663	0.32166	0.30167	1016.11	32.097
30,000	−47.832	4.3726	0.028573	0.00089065	0.31069	0.34884	994.85	32.081
35,000	−65.607	3.4676	0.023672	0.00073819	0.29952	0.40575	973.13	32.068
40,000	−69.700	2.7300	0.018823	0.00058726	0.29691	0.50559	968.08	32.052
45,000	−69.700	2.1489	0.014809	0.00046227	0.29691	0.64230	968.08	32.036
50,000	−69.700	1.6917	0.011652	0.00036391	0.29691	0.81589	968.08	32.020
60,000	−69.700	1.0488	0.0072175	0.00022561	0.29691	1.3160	968.08	31.991
70,000	−67.425	0.65087	0.0044485	0.00013920	0.29836	2.1434	970.90	31.958
80,000	−61.976	0.40632	0.0027366	0.000085707	0.30182	3.5215	977.62	31.930
90,000	−56.535	0.25540	0.0016952	0.000053145	0.30525	5.7436	984.28	31.897
100,000	−51.099	0.16160	0.0010575	0.000033182	0.30865	9.3018	990.91	31.868
km	°C	kPa abs	N/m³	kg/m³	10^{-6} N·s/m²	10^{-6} m²/s	m/s	m/s²
0	15.000	101.325	12.0131	1.2250	17.894	14.607	340.294	9.80665
1	8.501	89.876	10.8987	1.1117	17.579	15.813	336.43	9.8036
2	2.004	79.501	9.8652	1.0066	17.260	17.147	332.53	9.8005
3	−4.500	70.121	8.9083	0.90925	16.938	18.628	328.58	9.7974
4	−10.984	61.660	8.0250	0.81935	16.612	20.275	324.59	9.7943
5	−17.474	54.048	7.2105	0.73643	16.282	22.110	320.55	9.7912
6	−23.963	47.217	6.4613	0.66011	15.949	24.161	316.45	9.7882
8	−36.935	35.651	5.1433	0.52579	15.271	29.044	308.11	9.7820
10	−49.898	26.499	4.0424	0.41351	14.577	35.251	299.53	9.7759
12	−56.500	19.399	3.0476	0.31194	14.216	45.574	295.07	9.7697
14	−56.500	14.170	2.2247	0.22786	14.216	62.391	295.07	9.7636
16	−56.500	10.352	1.6243	0.16647	14.216	85.397	295.07	9.7575
18	−56.500	7.565	1.1862	0.12165	14.216	116.86	295.07	9.7513
20	−56.500	5.529	0.8664	0.08891	14.216	159.89	295.07	9.7452
25	−51.598	2.549	0.3900	0.04008	14.484	361.35	298.39	9.7300
30	−46.641	1.197	0.1788	0.01841	14.753	801.34	301.71	9.7147

[a] International Civil Aviation Organization; see Sec. 2.9.
[b] In these tables, if (for example, at 0 ft) μ is given as 0.373 72 and the units are 10^{-6} lb·sec/ft² then $\mu = 0.373\,72 \times 10^{-6}$ lb·sec/ft².

TABLE A.4 **Physical properties of common liquids at standard sea-level atmospheric pressure[a]**

Liquid	Temperature, T	Density, ρ	Specific gravity,[b] s	Absolute viscosity,[c] μ	Surface tension, σ	Vapor pressure, p_v	Bulk modulus of elasticity, E_v	Specific heat, c
	°F	slug/ft³	—	10^{-6} lb·sec/ft²	lb/ft	psia	psi	ft·lb/(slug·°R) = ft²/(sec²·°R)
Benzene	68°F	1.70	0.88	14.37	0.0020	1.45	150,000	10,290
Carbon tetrachloride	68°F	3.08	1.594	20.35	0.0018	1.90	160,000	5,035
Crude oil	68°F	1.66	0.86	150	0.002	—	—	—
Gasoline	68°F	1.32	0.68	6.1	—	8.0	—	12,500
Glycerin	68°F	2.44	1.26	31,200	0.0043	0.000002	630,000	14,270
Hydrogen	−430°F	0.143	0.074	0.435	0.0002	3.1	—	—
Kerosene	68°F	1.57	0.81	40	0.0017	0.46	—	12,000
Mercury	68°F	26.3	13.56	33	0.032	0.000025	3,800,000	834
Oxygen	−320°F	2.34	1.21	5.8	0.001	3.1	—	~5,760
SAE 10 oil	68°F	1.78	0.92	1,700	0.0025	—	—	—
SAE 30 oil	68°F	1.78	0.92	9,200	0.0024	—	—	—
Fresh water	68°F	1.936	0.999	21.0	0.0050	0.34	318,000	25,000
Seawater	68°F	1.985	1.024	22.5	0.0050	0.34	336,000	23,500
	°C	kg/m³	—	10^{-3} N·s/m²	N/m	kN/m² abs	10^6 N/m²	N·m/(kg·K) = m²/(s²·K)
Benzene	20°C	876	0.88	0.65	0.029	10.0	1030	1720
Carbon tetrachloride	20°C	1588	1.594	0.97	0.026	13.1	1100	842
Crude oil	20°C	856	0.86	7.2	0.03	—	—	—
Gasoline	20°C	680	0.68	0.29	—	55.2	—	2100
Glycerin	20°C	1258	1.26	1494	0.063	0.000014	4344	2386
Hydrogen	−257°C	73.7	0.074	0.021	0.0029	21.4	—	—
Kerosene	20°C	808	0.81	1.92	0.025	3.20	—	2000
Mercury	20°C	13550	13.56	1.56	0.51	0.00017	26200	139.4
Oxygen	−195°C	1206	1.21	0.278	0.015	21.4	—	~964
SAE 10 oil	20°C	918	0.92	82	0.037	—	—	—
SAE 30 oil	20°C	918	0.92	440	0.036	—	—	—
Fresh water	20°C	998	0.999	1.00	0.073	2.34	2171	4187
Seawater	20°C	1023	1.024	1.07	0.073	2.34	2300	3933

[a] In these tables, if (for example, for benzene at 68°F) μ is given as 1.437 and the units are 10^{-6} lb·sec/ft² then $\mu = 1.437 \times 10^{-6}$ lb·sec/ft².
[b] Relative to pure water at 60°F.
[c] For viscosity, see also Figs. A.1 and A.2.

TABLE A.5 **Physical properties of common gases at standard sea-level atmospheric pressure**[a]

Gas	Chemical formula	Molar mass, M	Density, ρ	Absolute viscosity,[b] μ	Gas constant, R	Specific heat, c_p	Specific heat, c_v	Specific heat ratio, $k = c_p/c_v$
at 68°F	—	**slug/slug-mol**	**slug/ft³**	**10^{-6} lb·sec/ft²**	**ft·lb/(slug·°R) = ft²/(sec²·°R)**	**ft·lb/(slug·°R) = ft²/(sec²·°R)**		—
Air		28.96	0.00231	0.376	1,715	6,000	4,285	1.40
Carbon dioxide	CO_2	44.01	0.00354	0.310	1,123	5,132	4,009	1.28
Carbon monoxide	CO	28.01	0.00226	0.380	1,778	6,218	4,440	1.40
Helium	He	4.003	0.000323	0.411	12,420	31,230	18,810	1.66
Hydrogen	H_2	2.016	0.000162	0.189	24,680	86,390	61,710	1.40
Methane	CH_4	16.04	0.00129	0.280	3,100	13,400	10,300	1.30
Nitrogen	N_2	28.02	0.00226	0.368	1,773	6,210	4,437	1.40
Oxygen	O_2	32.00	0.00258	0.418	1,554	5,437	3,883	1.40
Water vapor	H_2O	18.02	0.00145	0.212	2,760	11,110	8,350	1.33
at 20°C	—	**kg/kg-mol**	**kg/m³**	**10^{-6} N·s/m²**	**N·m/(kg·K) = m²/(s²·K)**	**N·m/(kg·K) = m²/(s²·K)**		—
Air		28.96	1.205	18.0	287	1003	716	1.40
Carbon dioxide	CO_2	44.01	1.84	14.8	188	858	670	1.28
Carbon monoxide	CO	28.01	1.16	18.2	297	1040	743	1.40
Helium	He	4.003	0.166	19.7	2077	5220	3143	1.66
Hydrogen	H_2	2.016	0.0839	9.0	4120	14450	10330	1.40
Methane	CH_4	16.04	0.668	13.4	520	2250	1730	1.30
Nitrogen	N_2	28.02	1.16	17.6	297	1040	743	1.40
Oxygen	O_2	32.00	1.33	20.0	260	909	649	1.40
Water vapor	H_2O	18.02	0.747	10.1	462	1862	1400	1.33

[a] In these tables, if (for example, for air at 68°F) μ is given as 0.376 and the units are 10^{-6} lb·sec/ft² then $\mu = 0.376 \times 10^{-6}$ lb·sec/ft².

[b] For viscosity, see also Figs. A.1 and A.2. Absolute viscosity μ is virtually independent of pressure, whereas kinematic viscosity ν varies with pressure (density) (Sec. 2.11).

TABLE A.6 **Areas of circles**

Diameter	Area	
in	**in²**	**ft²**
0.25	0.049087	0.00034088
0.5	0.19635	0.0013635
1.0	0.78540	0.0054542
2.0	3.1416	0.021817
3.0	7.0686	0.049087
4.0	12.566	0.087266
6.0	28.274	0.19635
8.0	50.265	0.34907
9.0	63.617	0.44179
10.0	78.540	0.54542
12.0	113.097	0.78540
m	**m²**	
0.05	0.0019635	
0.10	0.0078540	
0.15	0.017671	
0.20	0.031416	
0.25	0.049087	
0.30	0.070686	
0.50	0.19635	
1.00	0.78540	
1.50	1.76715	
2.00	3.14159	

TABLE A.7 **Properties of areas**

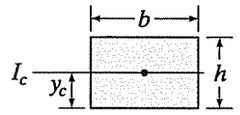
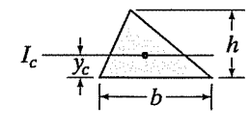
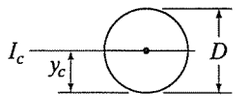
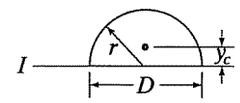
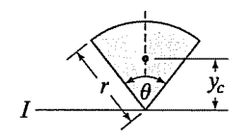
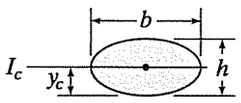
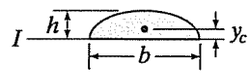
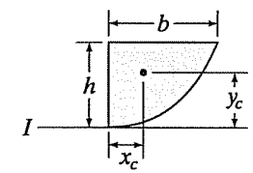
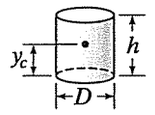
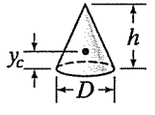
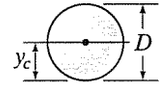
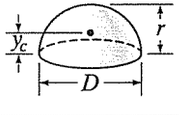
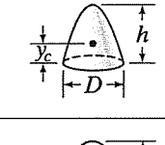
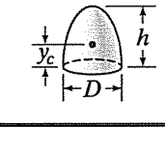
Shape	Sketch	Area	Location of centroid	I_c or $I = I_c + Ay_c^2$
Rectangle		bh	$y_c = \frac{h}{2}$	$I_c = \frac{bh^3}{12}$
Triangle		$\frac{bh}{2}$	$y_c = \frac{h}{3}$	$I_c = \frac{bh^3}{36}$
Circle		$\frac{\pi D^2}{4}$	$y_c = \frac{D}{2}$	$I_c = \frac{\pi D^4}{64}$
Semicircle		$\frac{\pi D^2}{8}$	$y_c = \frac{4r}{3\pi}$	$I = \frac{\pi D^4}{128}$
Circular sector		$\frac{\theta r^2}{2}$	$y_c = \frac{4r}{3\theta}\sin\frac{\theta}{2}$	$I = \frac{r^4}{8}(\theta + \sin\theta)$
Ellipse		$\frac{\pi bh}{4}$	$y_c = \frac{h}{2}$	$I_c = \frac{\pi bh^3}{64}$
Semiellipse		$\frac{\pi bh}{4}$	$y_c = \frac{4h}{3\pi}$	$I = \frac{\pi bh^3}{16}$
Parabola		$\frac{2bh}{3}$	$x_c = \frac{3b}{8}$ $y_c = \frac{3h}{5}$	$I = \frac{2bh^3}{7}$

TABLE A.8 **Properties of solid bodies**

Body	Sketch	Volume	Surface area	Location of centroid
Cylinder		$\frac{\pi D^2 h}{4}$	$\pi Dh + \frac{\pi D^2}{2}$	$y_c = \frac{h}{2}$
Cone		$\frac{1}{3}\left(\frac{\pi D^2 h}{4}\right)$	$\frac{\pi Dh}{2}\sqrt{1+\frac{D^2}{4h^2}} + \frac{\pi D^2}{4}$	$y_c = \frac{h}{4}$
Sphere		$\frac{\pi D^3}{6}$	πD^2	$y_c = \frac{D}{2}$
Hemisphere		$\frac{\pi D^3}{12}$	$\frac{\pi D^2}{2} + \frac{\pi D^2}{4} = \frac{3\pi D^2}{4}$	$y_c = \frac{3r}{8}$
Paraboloid		$\frac{1}{2}\left(\frac{\pi D^2 h}{4}\right)$		$y_c = \frac{h}{3}$
Half ellipsoid		$\frac{2}{3}\left(\frac{\pi D^2 h}{4}\right)$		$y_c = \frac{3h}{8}$

Appendix B
Equations in Fluid Mechanics[1]

Equations are a vital part of the language of fluid mechanics. Therefore, below we summarize the characteristics of the principal types and forms of equations used in fluid mechanics, and we present some of their useful properties.

All equations are either *identities* or *conditional equations.*

Identities are equations that are true for *all* values of the quantities involved. For example

$$\sin^2\theta + \cos^2\theta = 1 \qquad \text{and} \qquad x^2 - y^2 = (x + y)(x - y)$$

are identities. $y = f(x)$, where a function is being *defined* by an equation, is another type of identity. The equality sign is sometimes written $\equiv$ for identities.

Conditional equations are true only for certain values of the unknown quantities involved. For example,

$$x^2 - 1 = 0$$

is true for only $x = \pm 1$. More generally, $f(x) = 0$ is conditional.

Theoretical (or analytical) equations result from analysis. They are usually dimensionally homogeneous (Sec. 7.7). All equations are either theoretical (the majority in this book), empirical, or a combination of the two.

Empirical equations are based on experimental data. We discussed these in Sec. 1.1 and presented some classical examples from fluid mechanics in Sec. 8.18. It is interesting that some of these, of fundamental importance to the subject, are dimensionally nonhomogeneous (their dimensions do not "agree"; see Sec. 7.7).

Explicit equations. When the variable of interest is isolated on one side of an equation, as in $y = f(x, z)$, mathematicians say this equation has ***explicit form.*** Any equation that we can arrange into this form we shall call an explicit equation in y. Thus we can compute explicitly the value of the unknown. For example,

$$pv^n = \text{constant} \qquad (n = \text{constant}) \tag{2.6}$$

[1] The authors gratefully acknowledge the suggestions for this appendix provided by Dr. Peter Ross.

is explicit in v, and

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{e/D}{3.7} + \frac{2.51}{\mathbf{R}\sqrt{f}}\right) \tag{8.51}$$

is explicit in e.

Implicit equations. If an equation does not have a variable isolated on one side, in mathematics we say the equation has ***implicit form.*** Some equations that have implicit form, such as Eq. (2.6), we can rearrange into explicit form, and others we can not. Any equation that we can *not* rearrange into explicit form to isolate the variable of interest we shall call an implicit equation in that variable. Such equations only "imply" the value of the unknown variable of interest, and so we usually find there is more challenge in solving for it. A famous example in fluid mechanics is Eq. (8.51), which is implicit in f. Equation (8.81) is implicit in Q, L, D, h_L, and $\sum k$. An equation is either explicit or implicit in a particular variable; but it may be explicit in some variables and implicit in others, as in Eq. (8.51).

Algebraic equations contain only variables raised to various constant powers, not necessarily integers. They do not include transcendental or differential equations, etc., discussed here later. An example of an algebraic equation is Eq. (2.6) given earlier here, and

$$\frac{T_2}{T_1} = \frac{[2k\mathbf{M}_1^2 - (k-1)][2 + (k-1)\mathbf{M}_1^2]}{(k+1)^2\mathbf{M}_1^2} \tag{13.50}$$

which is a more complex example in the variable $\mathbf{M}_1$.

Transcendental equations are those in which the unknown variable or variables occur within a nonalgebraic function such as a trigonometric, logarithmic, or exponential function. Equations (8.46), (8.51), (12.23), (13.64), and (14.18) are examples. It follows that transcendental equations can not be algebraic or polynomial.

Polynomial equations are algebraic equations containing only terms with variables raised to positive integral powers and with constant coefficients. The equation

$$ax^n + bx^{n-1} + \cdots + dx^2 + ex + f = 0 \tag{B.1}$$

is a polynomial equation in x. All terms need not occur, i.e., some coefficients may be zero. If $a \neq 0$ we say that Eq. (B.1) is of ***degree*** n, from its largest exponent, and it has n ***roots,*** i.e., n real or complex values of x that cause y to be zero. Some hand calculators, such as the HP 48G, TI-82, and TI-92, will display the values of all the roots when given the values of all the coefficients.

Linear equations are first-degree polynomial equations, e.g.,

$$y = mx + c \tag{B.2}$$

Quadratic equations are second-degree polynomial equations, e.g.,

$$y = ax^2 + bx + c \tag{B.3}$$

Their name includes "quad" because we can use second-order equations to compute areas of (four-sided) rectangles. Equations (5.44) and (10.45) are examples of quadratic equations.

The two roots of the equation

$$ax^2 + bx + c = 0$$

we know well are

$$x = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} \tag{B.4}$$

Cubic equations are third-degree polynomial equations, e.g.,

$$ax^3 + bx^2 + cx + d = 0 \tag{B.5}$$

They occur quite frequently in fluid mechanics, as in Sample Probs. 3.5, 8.13, 10.6, and 10.8. Formulas to find the three roots from a, b, c, and d are quite involved and require a series of substitutions. As we noted earlier, some handheld calculators will display all the roots; others will solve for the root closest to an estimate provided by the user.

Sometimes we know one of the nonzero roots, say $x = r$, as in Sample Prob. 10.6; if not, we can find one by trials (see Sample Prob. 3.5). Then, dividing Eq. (B.5) by $(x - r)$, we obtain the quadratic equation

$$ax^2 + (b + ar)x - \frac{d}{r} = 0 \tag{B.6}$$

from which we can easily find the other two roots; they are

$$x = -\left(\frac{b}{2a} + \frac{r}{2}\right) \pm \sqrt{\left(\frac{b}{2a} + \frac{r}{2}\right)^2 + \frac{d}{ar}} \tag{B.7}$$

A particular form of equation, which occurs frequently in Chap. 10, is the specific energy Eq. (10.17), which we can write

$$E = y + \frac{G}{y^2}$$

where E and G are known positive constants. We can rearrange this into the cubic equation

$$y^3 - Ey^2 + G = 0$$

If we know one nonzero root, $y = r$, then dividing through by $(y - r)$ yields the quadratic equation

$$y^2 - \frac{G}{r^2}y - \frac{G}{r} = 0 \tag{B.8}$$

from which the other two roots are

$$y = \frac{G}{2r^2} \pm \sqrt{\left(\frac{G}{2r^2}\right)^2 + \frac{G}{r}} \tag{B.9}$$

Quartic equations are fourth-degree polynomial equations.

Infinite series equations are those including an infinite number of terms, such as Eqs. (5.18) and (13.31). Such equations may "converge" (to a specific total value) or "diverge" (the sum of the terms is increasing without bound).

Nonlinear equations are all equations other than linear (first-order polynomial) equations. They may be higher degree polynomial equations, transcendental equations, equations with nonintegral exponents such as

$$pv^n = \text{constant} \qquad (\text{for } n \neq 1) \tag{2.6}$$

and those with complex quotients, e.g.,

$$\frac{V_2}{V_1} = \frac{(k-1)\mathbf{M}_1^2 + 2}{(k+1)\mathbf{M}_1^2} \tag{13.51}$$

Differential equations, both ordinary and partial, have their own special terminology.

Ordinary differential equations (ODEs) contain ordinary derivatives, like dy/dx, i.e., derivatives of one or more unknown functions (dependent variables, y) of a *single* (independent) variable (x). Examples are

$$\tau = \mu\frac{du}{dy} \tag{2.9}$$

and

$$\frac{dp}{\gamma} + dz + d\left(\frac{V^2}{2g}\right) = 0 \tag{5.6}$$

The highest derivative in a differential equation determines its ***order;*** if the highest derivative is d^ny/dx^n we say the equation is of the nth order.

Partial differential equations (PDEs) contain partial derivatives of an unknown function (or functions) involving *more* than one independent variable; an example is

$$(a_x)_{st} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \tag{4.23a}$$

A celebrated *second-order* PDE is Laplace's equation,

$$\frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} = 0 \tag{14.25}$$

This equation is *linear* in ϕ, i.e., if ϕ_1 and ϕ_2 are each solutions, then so is $\phi_3 = a_1\phi_1 + a_2\phi_2$.

An example of a *vector* PDE is

$$\mathbf{a}_{st} = u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z} \tag{4.22}$$

where the boldface symbols are vectors.

Some equations governing fluid phenomena occur in the form of *systems* of simultaneous differential equations, such as the Navier-Stokes equations (6.8).

Difference equations include differences between two quantities, usually indicated by Δ. An example is

$$\frac{\Delta v}{v} \approx -\frac{\Delta p}{E_v} \tag{2.3a}$$

Equations with integrals occur in fluid mechanics, such as

$$\alpha = \frac{1}{V^2}\frac{\int u^3 dA}{\int u dA} \tag{5.4}$$

Some of these include *definite* integrals (with limits), like

$$\overline{\tau}_0 = \frac{1}{P}\int_0^P \tau_0 dP \tag{8.6}$$

Others include *line* integrals around a closed path, such as

$$\Gamma = \oint_L V\cos\beta\, dL \tag{14.9}$$

Note that these are not the same as ***integral equations,*** in which the unknown function $y(x)$ appears within an integral sign.

Simultaneous equations or ***systems of equations*** occur when more than one equation with the same unknown variables holds at the same time; all the equations are satisfied by the same values of the variables. Generally, the number of equations must equal the number of unknowns; otherwise the system may be underdetermined or overdetermined. Examples include the system of four equations governing flow in a single pipe, as summarized in Sec. 8.14. Larger such systems govern all multipipe flow problems (Secs. 8.29–8.32).

Vector equations are those involving vector quantities; an example is

$$\sum \mathbf{F} = \dot{m}_2\mathbf{V}_2 - \dot{m}_1\mathbf{V}_1 \tag{6.5}$$

where the boldface symbols are vectors.

Appendix C
Programming and Computer Applications

Many problems in fluid mechanics and hydraulics require repetitive calculations to solve them, so we can save considerable time and tedious labor by using programmed procedures. Although quite a few of these problems require progressive, step-by-step calculations, our most common need for repetitive calculations is when solving challenging equations. These equations are often nonlinear, they may involve transcendental functions, and they are often implicit (see Appendix B). Quite a lot of our problems are governed by a simultaneous set of n such equations in n unknowns.

This appendix introduces students to the various programming procedures available, and gives some idea of the characteristics and capabilities of each. Because all of these procedures make use of advanced technology, they continue to evolve quite rapidly, and students should remain conscious of this.

The most common types of programmable aids we use to help solve problems in fluid mechanics and hydraulics are:

- Programmable scientific calculators (and equation solvers)
- Spreadsheets
- Generic mathematics software
- Other generic software
- Applications software
- Programming languages

We shall discuss each of these types here.

Before you decide to adopt any of these aids, it is important that you know how much start up time you will need for learning how to prepare and/or use such programs. This can vary from a few hours to a number of weeks or more, and can require considerable effort in some cases. As a result, for most introductory problems (including most of those in Chaps. 1–7) it would not be time-effective to use such aids unless you are already familiar with them. Furthermore, it is not wise to use preprogrammed procedures unless you properly understand the systems simulated, how the programs work, and the nature of the assumptions involved. Most of these procedures are more appropriate with more advanced problems, as in Chap. 8 and thereafter. The authors *strongly* recommend that students *first* concentrate on mastering the principles involved by solving a larger number of simpler problems using manual procedures, with

pencil, paper, and a basic scientific calculator,[1] and by making a few manual repetitions where necessary.

C.1 PROGRAMMABLE SCIENTIFIC CALCULATORS

The programmable scientific calculator is attractive for programming because it is an extension of a very familiar tool, the basic (not programmable) scientific calculator, because its small size makes it easily portable to classes, to examinations, and into the field, and because its response times are usually very rapid. Although it is obviously less powerful than a personal computer (PC), the capabilities of some programmable scientific calculators are very extensive, and these capabilities continue to increase rapidly.

Earlier programmable calculators had small memories, and so could execute only short programs and store perhaps only one equation. Also, they could solve only explicit equations, and we had to convert these into unfamiliar codes before entry. Nevertheless, engineers devised ways to facilitate the solution of many previously tedious problems in fluid mechanics and hydraulics.[2]

More recent programmable scientific calculators, such as the HP 48G and 48GX, TI-82 and -92, have built-in preprogrammed polynomial solvers and equation solvers.

Polynomial solvers will automatically solve for all the roots of a polynomial equation (Appendix B) when we give them the values of all the coefficients. Some equations that we meet in fluid mechanics we can rearrange into polynomials (as in Sample Probs. 3.5, 5.8, 8.13, 10.1*d*, 10.6*a*), and then conveniently solve them this way.

Equation solvers on calculators will solve a single equation for one unknown, and the equation may be linear or nonlinear, transcendental or not, and explicit or implicit (Appendix B); the solver uses an iterative optimization procedure to accurately find the value of the unknown variable. If the equation has multiple roots, it will find the root closest to the guessed value that the user must provide. Thus the repetitive calculations are automated, and so greatly facilitated.

Although the calculator stores the equation in the form of codes, we can conveniently enter it with the help of a so-called ***equation writer,*** which presents the equation on the screen in the same form as we write it on paper. We can store equations in a directory in the calculator's memory, and recall them at any time for editing or problem solving. Each equation may include many different variables, and when solving them the user assigns the known values and designates the unknown to be solved. This is of great convenience because it enables the same equation to be used to solve a variety of problems.

[1] A basic scientific calculator is here defined to be one that is not programmable and does not have automatic equation solving capabilities.

[2] T. E. Croley, *Hydrologic and Hydraulic Computations on Small Programmable Calculators,* Iowa Institute of Hydraulic Research, University of Iowa, Iowa City, 1977.

In an alternative method we can plot the function on screen and then identify the root or roots from the graph.

When a simultaneous set of n nonlinear equations in n unknowns (Appendix B) governs the problem, we cannot solve them using present (2001) equation solvers on scientific calculators unless the equations can be combined into one. We can sometimes achieve this by substitution, as for the case of the four equations governing single-pipe flow (Sec. 8.14), although the result may be rather cumbersome. In such cases, we can also "link" the equations so that once we have solved the combined equation, the solver can then conveniently retrieve the other equations in the set and use the same variable values to solve them, one by one, for the remaining (eliminated) unknowns. Sample Probs. 8.9*a*, 8.11*b*, and 10.1*d* each involves an application to four equations in four unknowns which may be combined into one.

Appendix D further discusses the use of equation solvers and provides some example applications.

C.2 SPREADSHEETS

Nowadays most engineering students are familiar with spreadsheets like Excel, Lotus 1-2-3, or Quattro Pro. These continue to become more powerful and convenient for scientific applications, and they are programmable and now able to handle some quite complex functions. Because of their tabular organization, they are particularly suitable for solving problems that are traditionally solved in a stepwise and/or iterative fashion that is best tabulated; we have noted many appropriate applications throughout this book, a good example being to the solution of gradually varied flow profiles in open channels (Sample Prob. 10.9).

We can, of course, also set up spreadsheets to solve trial-and-error problems. For example, we can input an estimate of the unknown variable, and in subsequent columns have the spreadsheet evaluate the left and right sides of the governing equation. We can then repetitively try various values for the unknown until the two sides become equal. This is particularly useful where part of the input must come from reading a chart that is not available in equation form (see Sample Probs. 11.4 and 11.5).

Some spreadsheets include an equation solver (Appendix D), sometimes as an option, that automates the repetitive trials, and that therefore can save much labor and time. We have found such solvers, added on to spreadsheets, generally more difficult to understand and/or less convenient to program and use than software that is specifically designed for solving such mathematics problems (Sec. C.3). However, you should balance this against the time it will take you to become familiar with other software. In Appendix D.2 we provide example solutions using the equation solver in Excel.

Most spreadsheets can generate graphs and other visual representations of tabulated data, and it is usually quite easy to export data from spreadsheets to specialized graphing software.

C.3 GENERIC MATHEMATICS SOFTWARE

We can very usefully apply software for solving mathematics problems to solving problems in fluid mechanics. Mathematica is probably the largest such program, with extensive capabilities, often very involved. Other such packages, less involved to varying degrees, include Derive, GAMS, Maple, Mathcad, MATLAB, Theorist, and TK Solver. Specialized packages are also available for symbolic manipulations, statistics, numerical methods, and optimization.

Most of these software packages are available in a variety of versions or forms, some of which include inexpensive but limited-capability student editions. Check carefully whether the capabilities you need (such as to solve simultaneous nonlinear equations as described in this book) are included in the version you are planning to use. Some brands have the needed capabilities in optional supplemental packages.

Compared with programmable calculators, generic mathematics software has all the potential advantages that come with a PC, such as greater computing power and the ability to produce fancy reports. Certain types of problems mentioned earlier, involving a set of simultaneous nonlinear equations that we cannot reduce to one equation by substitution (see, e.g., Sample Prob. 8.14), we can only solve with an equation solver on PC-based software or by manual iteration. Generic software has the disadvantages of a PC also, which include slower bootup times than a calculator and less convenient accessibility. Accessibility is improving, however, with increasingly portable and affordable laptop computers.

In Appendix D.3 we include example applications of an equation solver and a polynomial solver in generic mathematics software using Mathcad. We chose Mathcad because it is a more basic package particularly geared to student use. In addition to solving equations, Mathcad readily plots graphs, it has a convenient report-writing capability that includes equations, it has a spreadsheet equivalent, and it has optional modules for special applications.

C.4 OTHER GENERIC SOFTWARE

A variety of generic software packages have been developed for the solution of conditions that vary throughout a region, such as temperature, pressure, or stress. In particular, these packages can be used to solve flow fields, such as flow through passages of various shapes, flow around immersed bodies, and flow through porous media such as aquifers and earth dams. These packages use relatively advanced procedures, such as the ***finite difference method*** (FDM), the ***finite element method*** (FEM), or the ***boundary element method*** (BEM), to solve the differential equations governing the flow field by discretizing the flow region and/or its boundaries. Each procedure has its advantages and disadvantages such as: simplicity, flexibility, and convenience of formulating the problem to be solved. ***Computational fluid mechanics*** and ***computational fluid dynamics*** (CFD) are general terms used for any program developed to specifically solve flow fields.

Some of these packages can solve flow fields with a free surface, known as ***free-surface flows.*** A free surface may be part of a problem because the liquid surface forms one of the boundaries, such as for waves or the groundwater table, or because an internal surface forms, as around cavities produced by cavitation (Sec. 5.10). The fact that the position of the free surface usually depends on the solution of the flow field, and vice versa, adds to the complexity of the problem.

C.5 APPLICATIONS SOFTWARE

Much software for particular applications in fluid mechanics and hydraulic engineering has been developed by individual authors, private companies, research institutions, and government agencies. A textbook by Olfe (see Ref. 66 in Appendix E) has accompanying software devoted to fluid mechanics problems.

Applications software packages prepared commercially or by government agencies, primarily for fluid mechanics rather than thermodynamics, include the following.

Pipeline Models

These may be for the steady or unsteady flow (including surges and water hammer) of gases or liquids in single pipes or branching pipes, which may incorporate pumps, turbines and valves. They may model laminar and/or turbulent flow, various viscosities and friction factors, flow patterns, and multiphase flow. They include: ALGEBRA, GENEQUA, IMPREM, LIQT, PipeCalc, Pipe Flow 2, PIPE-PRO, PIPESIZE, SINGLE, SUR, SURGES, WH, WHAMO, and WinPipeD.

Pipe System Models

These model systems of pipelines in series and/or parallel that have a single inflow and a single outflow. They may include components such as fittings, valves, and pumps. They may simulate laminar and/or turbulent flow of liquids, gases (compressible flow), and slurries. They include: AFT Arrow, DESIGNHELP Slurry Software, DFS Branch, HYDROFLO, Integrated Plant Package (IPP), PIPE-FLO Express, and SF Pressure Drop.

Pipe Network Models

These can model networks that have loops and multiple inflows and/or outflows. They often can handle pressure zones, valves, fittings, pumps, storage, and graphical presentations. Some can model both pressure and gravity flow, pressure transients, cavitation, heat transfer, and the flow of many fluids including liquids, compressible gases, and slurries. They include: AFT Fathom, AFT Impulse, Autowater, Aquanet, CYBERNET, Click & Go, DFS DesigNet, EPAnet, Faast, FlowMaster, Helix delta-Q, Hydroflo II, Hydronet, KYPIPE, Micro

Hardy Cross, PIPENET, Stoner, Tdhnet, Wadiso, Water, WaterCAD, Watermax, Waterworks, Watnet, Watsys, Wgraph, and ZRPNet.

System Component Models

These model such components as accumulators, pumps, fans, compressors, pressure and flow control valves, orifices, chokes, steam traps, surge tanks, and hydraulic cylinders. They include: AFT Arrow, AFT Fathom, AFT Impulse, Helix delta-Q, HydraCyl, MAYA ESC, Orifices, SizeFE, and TrapBase.

Measuring Device Models

These model such fluid measurement devices as orifices, flow meters, and weirs. They include: FE-Sizer, Flow Flex, Orifices, and WEIRS.

Flow Field (CFD) Models

These simulate the flow of fluids in complex geometries that may be two- or three-dimensional. They can model laminar and/or turbulent flows, compressible and/or incompressible flows, steady and/or unsteady flows, fluids of various densities, viscosities, temperatures, pressures, with various solute concentrations, and particle trajectories. Some can handle sources and sinks, rarefied gases, convection, vortices and wakes, large eddies, boundary layer separation, moving boundaries, hydrodynamic forces, combustion, explosions, shocks, high-Mach-number flows, heat transfer, jets, transport, chemical mixing and reactions. Some have been tailored for applications in aerodynamics, electronics cooling, HVAC, flows around underwater vehicles, cavity filling (casting), and environmental problems. They include: CFD2000, COMPACT, DSMC, DSWT, FIDAP, FLACS, Fluent/UNS, MAYA ESC, Rampant, SUBFLO_2, TASCflow, TMRC, and WRAFTS.

Shape Analysis Models

These analyze hydrodynamic forces and velocity distributions around particular shapes such as airfoils, pump impellers, and ship hulls. Many include analysis of specialized characteristics such as pressure coefficients, pitch damping, and vortex effects. They include: FDLIB, Michlet, PumpBase, Pump Design Code (PDC), VisualFoil, and VORSTAB-PC.

Flow Process Models

These non-CFD codes may simulate such flow processes as turbulence, sonic choking, flow-stream mixing, chemical reactions, and heat transfer. They often calculate pressure- and temperature-dependent properties and parameters. They include: AFT Arrow, FDLIB, FlowComp, FLUIDTOOLS, Fluidyn, IR-Vision, and VISCOS.

Open-Channel Flow Models

These often can simulate both free surface and pressure flows, adverse flows, and junction losses, and are usually compatible with AutoCAD or similar computer-aided drafting (CAD) packages. Some are models of storm drains and natural drainage channels. They include: Boss, CDS, CIA, CivilTools, CulvertMaster, DAMBRK (N.O.A.A.), Dodson, DWOPER (N.O.A.A.), Eagle Point, Engenious, FEQ, FLDWAV (N.O.A.A.), FLDWY (U.S. S.C.S.), FlowMaster, FLOWPROF, FLOW31, GWN-STORM, HEC-2 and HEC-RAS (U.S. Army Corps of Engineers), HY-8, Hydra, Hydraflow, Hydrain-HYCLV, Intelisolve, MOUSE, Networx, PIPECAP, Pizer, Quickpipe, SECTION, SoftDesk, StormCAD, Streamline, SWMM (U.S. E.P.A.), THYSYS, WSPRO (U.S.G.S.), WSP2 (U.S. S.C.S.), and XP Software.

Sanitary Sewer System Models

These models usually simulate open-channel flow, as above, but can also simulate pressure flow (as in pipelines, above) if the sewers become full. They include: CAPS, Eagle Point, Hydra, MODS, Pizer, and SewerCAD.

Other Applications

Other related areas for which applications software packages are available include: potential flow; complex flow systems of interconnected components; water hammer and surges; jet and spillway flows; air entrainment; shallow-water hydrodynamics; reservoir hydrodynamics; coastal hydraulics; natural and urban hydrology; seepage; groundwater hydraulics and transport; water quality of rivers, impoundments, and estuaries; and stream sedimentation.

C.6 PROGRAMMING LANGUAGES

Instead of using commercial software of the various types described in the preceding sections, an engineer can learn a programming language and then prepare "custom-made" programs to solve specific problems.

Assuming you know a programming language, which takes quite a while to learn, it will still take you a considerable amount of time to write programs. Because there are quite a lot of problems that are best solved with computing aids, it would be very inefficient to require students to write programs to solve them. Therefore we have not included such assignments or sample codes in this book, and instead we encourage students to master programmable calculators, spreadsheets, and generic mathematics software (Secs. C.1–C.3). While an ability to write computer programs is important, we believe that most students would better defer this until graduate studies or for research or professional projects. Students interested to pursue programming for fluid mechanics may see example programs in other texts (Refs. 24, 26, 73, 88, and 89).

The two major programming languages in current use by engineers in this field are FORTRAN and C. Each has its strengths and weaknesses.

FORTRAN stands for FORmula TRANslation. It was released in 1957, to provide a high-level language for scientific computing. It uses a notation that simplifies the programming of mathematical formulas, as its name suggests. Long the preferred language of civil and mechanical engineers, it is the most convenient of the languages used for programming numerical methods. In the area of large scientific programs, FORTRAN is also the preferred language for parallel processing.

The programming language C, released around 1970, uses a low level of programming abstraction (is more detailed), which gives it a more flexible and therefore more powerful programming capability than FORTRAN. As a result, it has broader abilities, being suitable for graphics and for data bases and transaction processing in addition to scientific computation. However, its extra power and breadth comes at the cost of requiring more convoluted programming; hence C is harder to learn than FORTRAN and a casual programmer will find it harder to follow the logic of C.

As a result, there exists a wealth of scientific and engineering software written in FORTRAN, and there are many graphics packages written in C that are attractive to scientists and engineers. Although the latest revision, FORTRAN 95, can reportedly do the same graphics as C, scientists would naturally prefer to use what is readily available, and ways have been developed to do this. Now one can execute a software package that includes both languages. In a modular mode, a main FORTRAN program, for example, can call a subroutine written in C.

It seems we regularly hear mention that FORTRAN is a dying language, even though people have now said this for many decades. It will not die, because it is the most convenient language for advanced numerical operations, because there is such a vast inventory of FORTRAN-based software, and because all high-powered, high-level computational software is written in FORTRAN. Many of these software packages like ANSYS, SPICE, and FEM and CFD programs contain *millions* of lines of FORTRAN code that will not translate into other languages. I-DEAS, for example, contains over ten million lines.

The language BASIC, which stands for Beginner's All-purpose Symbolic Instruction Code, is somewhat like FORTRAN. BASIC was developed in the mid-1960s, after FORTRAN, to provide an alternative that is easier to learn and to use (but less efficient to execute). Much BASIC code looks like a simplified subset of FORTRAN, but it resembles the English language more closely. To facilitate "debugging" (correcting errors), in early versions of BASIC the compiler converts statements into machine language and executes them on a line-by-line basis, rather than compiling the entire program before execution.

Another important tool for program development is the more recently developed Visual Basic. Not related to the early BASIC just discussed, Visual Basic is a visual language based on an integrated graphical user interface (GUI) and a compiled language more like a simplified C. It works in a Windows environment, where it is a powerful tool for developing user-friendly applications.

C.7 SUMMARY

Of the discussed programmable aids, probably the most important for students of fluid mechanics to learn to use, particularly in a second course, are: spreadsheets, equation solvers on programmable calculators, and a simple and flexible mathematics software package.

Note that the mathematics software is the only one of these three aids that can perform the functions of all of them. Although advanced scientific calculators can solve a system of linear equations, and can solve a single nonlinear (and implicit) equation, they cannot (in 2001) automatically solve a system of nonlinear equations such as we frequently meet in applied fluid mechanics. While computer software can do this, computers usually are not so accessible as a calculator, and by using the proper procedures calculators can solve the majority of problems in this book. It is clear that students need to be familiar with the abilities and advantages of each.

In addition, students should take every opportunity to see demonstrations of, and learn more about, the capabilities of the various types of generic and applications software described in Secs. C.3–C.5.

Students interested in graduate studies or research are also advised to learn a programming language or two, probably FORTRAN and/or C.

Appendix D

Examples of Using Solvers

We have already discussed automated ***equation solvers*** a little in Appendix C. They are available in programmable scientific calculators (Sec. C.1) and in certain software packages (Secs. C.2 and C.3). An equation solver automatically uses a process of iteration to find roots, i.e., values of unknowns that satisfy equations. If there is more than one solution, it will find the values closest to the estimated values that were provided initially. Because implicit equations and nonlinear simultaneous equations (Appendix B) occur quite often in fluid mechanics, you could find it very helpful to have some familiarity with equation solvers. Problems of these kinds occur in Sample Probs. 3.5, 5.8, 8.9, 8.11–8.19, 10.1, 10.6, 10.8, 10.12–10.13, 12.6, and 13.7.

Some of these equations may be reducible to polynomial equations (Appendix B), which we can also conveniently solve with ***polynomial solvers.*** Appendix C.1 discusses polynomial solvers provided on programmable scientific calculators, and of course they are also available in computer software.

Here, we have chosen to illustrate solutions using the HP 48G calculator, the Excel spreadsheet, and the mathematics software package Mathcad, because of their popularity and convenience for student use. In Secs. D.2 and D.3 you can easily compare the appearance and methods of solution of the equation solvers in Excel and Mathcad when solving the same problems; Sample Prob. 3.5 involves eight simultaneous equations and Sample Prob. 8.14 entails fourteen. Because Sample Prob. 3.5 can be reduced to a third-degree polynomial (Appendix B), we have also included solutions using polynomial solvers. Your instructor can provide solver solutions to additional sample problems.

D.1 Solvers in the HP 48G and 48GX Programmable Scientific Calculators

The following should help students learn the required procedures and keystrokes.

Using the HP 48G Equation Solver

A number of available procedures may help you solve equations. These include procedures which:

- Key in and save a single equation
- Display, edit, or "rearrange" saved equations
- Purge a saved equation from memory
- Solve with a single, presaved equation
- Solve using a set of (not linked) saved equations
- Solve using a set of linked, saved equations

Your instructor can provide sample keystrokes for each of these.

Once installed, you can, for example, use either of the last two procedures mentioned to conveniently solve any turbulent flow problem in a single pipe. Such problems are governed by four simultaneous equations that can and must be first combined into a single equation (Secs. 8.13–8.14), because programmable scientific calculators can only solve one equation in one unknown at a time (Appendix C.1).

Sample Problem 3.5 (by HP 48G Polynomial Solver)

Use the following keystrokes, for the purposes indicated:

Access the solvers: ↱, SOLVE
Move down two levels: ▼, ▼
Select the Polynomial Solver: OK (= Function Key F)
Open square brackets: ↰, [ ]
Enter coefficients AN…A0 in order with spaces between them (zero coefficients must be entered as zeros; follow coefficients by +/– to insert a minus sign): 27.7333333333, SPC, 0, SPC, 998.4, +/–, SPC, 600
Indicate that all the coefficients are entered: OK (= Function Key F)
Instruct the calculator to find the roots: SOLVE (= Function Key F)
Make the results readable: ENTER, ↰, EDIT
Move the cursor across the results to read them: Repeat ▶ or ◀ (*The results are:* 0.607179511351 5.67332414539 –6.2050365674)
Prepare for another problem: ON, ↰, CLEAR

Note: AN is the coefficient of x^n, entered first, and *A0* is the constant, entered last.

D.2 Solver in the Excel Spreadsheet

First ensure that your equation solver in Excel is active. It may be in a so-called "add-on," which has not been activated (installed) to save memory.

To use this solver, you will find it convenient to place the known variable values in one area, the estimates of the unknown variable values in a second area, and the equations (formulas) in a third area. You must enter equations in

the form of codes beginning with =, and use symbols for mathematical operations as indicated below in the examples. You may display (and so print) equations by toggle-pressing CTRL+` (accent key at top left). Be sure to set up the same number of equations and unknowns (guessed values), with equations and targets referring only to knowns and unknowns (not to each other). You can preadjust your guessed unknowns to be reasonable and similar to the computed values.

For convenience and to simplify equations, we suggest you assign "names" to the cells, individually or in blocks (select the cells, then in the **Insert** menu select **Name** and **Define**). We have assigned the cell names to be the same as the block titles, and have used the prefix *g* to indicate guessed quantities and *c* to indicate quantities computed by equations.

Then, in **Solver** (in the **Tools** menu), select a target cell and set it equal to a value. You may set additional "targets" by using Constraints in a similar way. Specify, by the cell range (e.g., B9:F11) which (guessed) values are to be changed. And in Constraints, require the guessed values to be equal to the computed values (e.g., $gQ = cQ$, etc.). You may also use Constraints to prevent Excel from exploring inappropriate variable ranges (in Sample Prob. 8.14 we required all cQ, gV, gh, gR, $gQQ1$, and $gQQ3$ to be ≥ 0). Note that as you adjust the Constraints, etc., the name of the target cell may change, so be sure to reset this last, just before executing the solver.

When you press Solve, Excel automatically iterates to find a solution closest to your guessed values. For larger numbers of simultaneous equations (and unknowns), note that the success of the procedure tends to become more sensitive to the estimated values; so if at first you are not successful, try some different estimated values.

Immediately after executing Solve, you may request (and so print) an Answer Report by selecting "Answer" Report in the Solver Results box.

Sample Problem 3.5 (by Excel Equation Solver)

Here we have solved this problem in its initial form, with eight equations in eight unknowns (it could be simplified). For our target we have used a formula that represents the sum of the moments about hinge H when $R_E = 0$ (the gate is about to open), which must be zero.
Initial setup:

A	B	C	D	E	F	G	H	I	J
2	Knowns:								
3		W	dw	gam	Av	dv			
4		500	1.2	62.4	8	2			
5	Unknowns:								
6		gFs	gAs	gds	ghcs	gOH	gFv	ghcv	gx
7		30	1	0.3	0.3	1	300	0.6	1
8	Equations:								
9		cFs	cAs	cds	chcs	cOH	cFv	chcv	
10		=gam*ghcs*gAs	=gOH*2	=gOH/3	=gx/2	=gx/(COS(PI()/6))	=gam*ghcv*Av	=gx	
11						Target			
12						=W*dw+gFs*gds-gFv*dv			

A	B	C	D	E	F	G	H	I	J
2	Knowns:								
3		W	dw	gam	Av	dv			
4		500	1.2	62.4	8	2			
5	Unknowns:								
6		gFs	gAs	gds	ghcs	gOH	gFv	ghcv	gx
7		30	1	0.3	0.3	1	300	0.6	1
8	Equations:								
9		cFs	cAs	cds	chcs	cOH	cFv	chcv	
10		18.72	2	0.3333	0.5	1.1547005	299.52	1	
11						Target			
12						9			

After solving:

A	B	C	D	E	F	G	H	I	J
2	Knowns:								
3		W	dw	gam	Av	dv			
4		500	1.2	62.4	8	2			
5	Unknowns:								
6		gFs	gAs	gds	ghcs	gOH	gFv	ghcv	gx
7		26.564	1.4022	0.2337	0.3036	0.7011105	303.1	0.6072	0.6072
8	Equations:								
9		cFs	cAs	cds	chcs	cOH	cFv	chcv	
10		26.564	1.4022	0.2337	0.3036	0.7011105	303.1	0.6072	
11						Target			
12						-2.533E-07			

Microsoft Excel 8.0a Answer Report
Worksheet: [D3-5E2lt.XLS]Sheet1
Report Created: 10/11/00 4:01:50 PM

Target Cell (Value Of)

Cell	Name	Original Value	Final Value
$G$12	Target	5.2	-2.53298E-07

Adjustable Cells

Cell	Name	Original Value	Final Value
$C$7	gFs	26	26.56367604
$D$7	gAs	1	1.402221018
$E$7	gds	0.2	0.233703503
$F$7	ghcs	0.3	0.303589756
$G$7	gOH	0.7	0.701110509
$H$7	gFv	300	303.1040122
$I$7	ghcv	0.6	0.607179512
$J$7	gx	0.6	0.607179512

Constraints

Cell	Name	Cell Value	Formula	Status	Slack
$C$7	gFs	26.56367604	$C$7=$C$10	Binding	0
$D$7	gAs	1.402221018	$D$7=$D$10	Not Binding	0
$E$7	gds	0.233703503	$E$7=$E$10	Binding	0
$F$7	ghcs	0.303589756	$F$7=$F$10	Binding	0
$G$7	gOH	0.701110509	$G$7=$G$10	Binding	0
$H$7	gFv	303.1040122	$H$7=$H$10	Binding	0
$I$7	ghcv	0.607179512	$I$7=$I$10	Not Binding	0

Note: With the guessed values shown, Excel made eight trial solutions before finding the above solution. The final value for x is 0.607 ft.

Sample Problem 8.14 (by Excel Equation Solver)

This problem has 14 unknowns and 14 governing equations (two under "Targets"); the equation for the answer is not one of them as it does not satisfy any constraints.

In this example we have placed variables and equations in named "blocks" (three members per block; see the Answer Report). To add the implied subscripts

to such equations, you must select the block, enter the equation, and then press Ctrl+Shift+Enter; brackets { } will appear around the equation.
Initial setup:

A	B	C	D	E	F	G
2	Knowns:					
3			g	nu	ElevA	ElevC
4			32.2	0.00001217	250	160
5		Pipe	Q	D	L	e
6		1		=15/12	6000	0.0004
7		2	3.3	=10/12	1500	0.0004
8		3		=8/12	4500	0.0004
9	Unknowns:					
10		Pipe	gQ	gV	gh	
11		1	5	5	40	
12		2		5	20	
13		3	2	5	50	
14		Pipe	gR		gf	
15		1	400000		0.02	
16		2	400000		0.02	
17		3	400000		0.02	
18	Equations:					
19		Pipe	cQ		ch	
20		1	=gV*PI()*D*D/4		=gf*L*gV^2/(D*2*g)	
21		2	=gV*PI()*D*D/4		=gf*L*gV^2/(D*2*g)	
22		3	=gV*PI()*D*D/4		=gf*L*gV^2/(D*2*g)	
23		Pipe	cR		cf	
24		1	=D*gV/nu		=(1/(-2*LOG(e/(3.7*D)+2.51/(gR*SQRT(gf)))))^2	
25		2	=D*gV/nu		=(1/(-2*LOG(e/(3.7*D)+2.51/(gR*SQRT(gf)))))^2	
26		3	=D*gV/nu		=(1/(-2*LOG(e/(3.7*D)+2.51/(gR*SQRT(gf)))))^2	
27				Targets		Answer
28				=ghh1+ghh3-ElevA+ElevC		ElevB
29				=gQQ1-QQ2-gQQ3		=ElevA-ghh1-ghh2

A	B	C	D	E	F	G
2	Knowns:					
3			g	nu	ElevA	ElevC
4			32.2	1.217E-05	250	160
5		Pipe	Q	D	L	e
6		1		1.25	6000	0.00040
7		2	3.3	0.833333333	1500	0.00040
8		3		0.666666667	4500	0.00040
9	Unknowns:					
10		Pipe	gQ	gV	gh	
11		1	5	5	40	
12		2		5	20	
13		3	2	5	50	
14		Pipe	gR		gf	
15		1	400000.00		0.02	
16		2	400000.00		0.02	
17		3	400000.00		0.02	
18	Equations:					
19		Pipe	cQ		ch	
20		1	6.135923152		37.26708075	
21		2	2.727076956		13.97515528	
22		3	1.745329252		52.4068323	
23		Pipe	cR		cf	
24		1	513557.93		0.016579069	
25		2	342371.95		0.017690873	
26		3	273897.56		0.018410373	
27				Targets		Answer
28				0		ElevB
29				-0.3		190

After solving:

A	B	C	D	E	F	G
2	Knowns:					
3			g	nu	ElevA	ElevC
4			32.2	1.217E-05	250	160
5		Pipe	Q	D	L	e
6		1		1.25	6000	0.00040
7		2	3.3	0.833333333	1500	0.00040
8		3		0.666666667	4500	0.00040
9	Unknowns:					
10		Pipe	gQ	gV	gh	
11		1	5.336757706	4.348781409	23.34145522	
12		2		6.050434317	18.1286951	
13		3	2.036757706	5.834881022	66.65854478	
14		Pipe	gR		gf	
15		1	446670.24		0.016559126	
16		2	414299.80		0.017717678	
17		3	319631.94		0.018679873	
18	Equations:					
19		Pipe	cQ		ch	
20		1	5.336757706		23.34145522	
21		2	3.3		18.12869507	
22		3	2.036757706		66.65854478	
23		Pipe	cR		cf	
24		1	446670.24		0.016559126	
25		2	414299.80		0.017717678	
26		3	319631.94		0.018679873	
27				Targets		Answer
28				0.00000E+00		ElevB
29				-2.22045E-14		208.52985

Microsoft Excel 8.0a Answer Report
Worksheet: [D8-12E3Dt.XLS]Sheet1
Report Created: 10/11/00 4:20:33 PM

Target Cell (Value Of)

Cell	Name	Original Value	Final Value
$E$28	Target	0	0

Adjustable Cells

Cell	Name	Original Value	Final Value
$D$11	gQQ1	5	5.336757706
$D$13	gQQ3	2	2.036757706
$E$11	gVV1	5	4.348781409
$E$12	gV	5	6.050434317
$E$13	gVV3	5	5.834881022
$F$11	ghh1	40	23.34145522
$F$12	ghh2	20	18.1286951
$F$13	ghh3	50	66.65854478
$D$15	gR	400000.00	446670.24
$D$16	gR	400000.00	414299.80
$D$17	gR	400000.00	319631.94
$F$15	gf	0.02	0.016559126
$F$16	gf	0.02	0.017717678
$F$17	gf	0.02	0.018679873

Constraints

Cell	Name	Cell Value	Formula	Status	Slack
$E$29	Targets	-2.22045E-14	$E$29=0	Not Binding	0
$D$21	cQQ2	3.3	$D$21=$D$7	Not Binding	0
$D$20	cQQ1	5.336757706	$D$20>=0	Not Binding	5.336757706
$D$21	cQQ2	3.3	$D$21>=0	Not Binding	3.3
$D$22	cQQ3	2.036757706	$D$22>=0	Not Binding	2.036757706
$D$11	gQQ1	5.336757706	$D$11=$D$20	Not Binding	0
$D$13	gQQ3	2.036757706	$D$13=$D$22	Not Binding	0
$F$11	ghh1	23.34145522	$F$11=$F$20	Not Binding	0
$F$12	ghh2	18.1286951	$F$12=$F$21	Not Binding	0
$F$13	ghh3	66.65854478	$F$13=$F$22	Not Binding	0
$F$15	gf	0.016559126	$F$15=$F$24	Not Binding	0
$F$16	gf	0.017717678	$F$16=$F$25	Not Binding	0
$F$17	gf	0.018679873	$F$17=$F$26	Not Binding	0
$D$15	gR	446670.24	$D$15=$D$24	Not Binding	0
$D$16	gR	414299.80	$D$16=$D$25	Not Binding	0
$D$17	gR	319631.94	$D$17=$D$26	Not Binding	0

Note: With the guessed values shown, Excel made 13 trial solutions before finding this last solution.

D.3 SOLVERS IN THE MATHCAD GENERIC MATHEMATICS SOFTWARE

When using the equation solver in Mathcad, we enter equations and see them displayed on the screen in the same form that we normally write them on paper. This is a great advantage. (With many other solvers we must enter them in the form of less familiar "codes," which makes it harder to relate to the equations and to detect errors.) We simply list the known variable values and estimates of the unknown variable values on a "worksheet," below these we list the equations in a designated "solve block" beginning with the word "Given," and below the equations, beginning with a "Find" statement, we call for the solutions for the unknown variables. Mathcad automatically iterates to find the solution closest to our estimates.

Learning time is relatively short. Your instructor can provide you with a copy of our "Tips on Mathcad" which will help you get started, and with Mathcad solutions to more sample problems.

For larger numbers of simultaneous equations (and unknowns), note that the success of the procedure tends to become more sensitive to the estimated values; so if at first you are not successful, try some different estimated values.

The second solution below to Sample Prob. 3.5 uses a polynomial solver called *polyroots,* which does not require guessed values and which returns all the roots at once.

Sample Problem 3.5 (by Mathcad Equation Solver)

Only the text and symbols in Swiss (sans serif) font are active.

We have let dw, dv, and ds be the distances from H to the forces W, F_v, and F_s, respectively. The moment equation represents moments about hinge H when $R_E = 0$ (as the gate is about to open).

Known values: $W := 500$ $dw := 1.2$ $\gamma := 62.4$

$Av := 4 \cdot 2$ $dv := \frac{4}{2}$

Guessed values: $Fs := 30$ $OH := 1$ $As := 1$ $ds := 0.3$ $hcs := 0.3$

$Fv := 300$ $x := 1$ $hcv := 0.6$

Given

Clockwise moments: $W \cdot dw + Fs \cdot ds - Fv \cdot dv = 0$

where $Fs = \gamma \cdot hcs \cdot As$ $hcs = \frac{x}{2}$ $ds = \frac{OH}{3}$ $OH = \frac{x}{\cos(30 \cdot deg)}$ $As = OH \cdot 2$

$Fv = \gamma \cdot hcv \cdot Av$ $hcv = x$

with constraints $x > 0$ $x < 1$

$$\begin{bmatrix} Fs \\ Fv \\ As \\ OH \\ ds \\ hcs \\ hcv \\ x \end{bmatrix} := Find(Fs, Fv, As, OH, ds, hcs, hcv, x)$$

Results: $OH = 0.7011$ $Fs = 26.5637$ $hcs = 0.3036$ $As = 1.4022$ $ds = 0.2337$

$x = 0.6072$ $Fv = 303.1040$ $hcv = 0.6072$

Note: When the above calculations are repeated for $x > 1$, the following results are obtained:

$OH = 6.551$ $Fs = 2319.2$ $hcs = 2.8367$ $As = 13.102$ $ds = 2.1837$

$x = 5.6733$ $Fv = 2832.1$ $hcv = 5.6733$

Answer: From these two sets of results we find the gate will remain closed for $0.607 < x < 5.67$ ft.

Notes: Mathcad displays on screen the colon equals (or "definition symbol") as := but you must type it in as a colon. Use the bold (or Boolean) equals to specify that two sides of an equation must have the same numerical value; type it in as [Ctrl]=. Use the "regular" equals to call for the display of results, e.g., $x =$.

Choose **Matrix** in the **Insert** menu to create the column vectors. We may use constraints to prevent Mathcad from exploring inappropriate variable ranges. Because in this case all unknowns must be > 0 when $x > 0$, no additional constraints are needed.

Sample Problem 3.5 (by Mathcad Polynomial Solver)

Known values: $W := 500$ $dw := 1.2$ $\gamma := 62.4$ $Av := 4 \cdot 2$ $dv := \frac{4}{2}$

Equations to solve:

Clockwise moments: $W \cdot dw + Fs \cdot ds - Fv \cdot dv = 0$

where $Fs = \gamma \cdot hcs \cdot As$ $hcs = \frac{x}{2}$ $ds = \frac{OH}{3}$ $OH = \frac{x}{\cos(30 \cdot deg)}$ $As = OH \cdot 2$

$Fv = \gamma \cdot hcv \cdot Av$ $hcv = x$

Eliminating unknowns, these are equivalent to: $a \cdot x^3 + b \cdot x^2 + c \cdot x + d = 0$

where the coefficients are: $a := 1$ $b := 0$ $c := -2.25 \cdot Av \cdot dv$ $d := \frac{2.25}{\gamma} \cdot W \cdot dw$

Solutions:

$$r := \begin{bmatrix} d \\ c \\ b \\ a \end{bmatrix} \qquad \text{polyroots}(r) = \begin{bmatrix} -6.2805 \\ 0.6072 \\ 5.6733 \end{bmatrix}$$

Notes: The small black block (or square) following some equations indicates that we have made them inactive (see **Properties** in the **Format** menu) because they are not necessary to the solution. This worksheet may be simplified by omitting all entries above "Solutions" and by directly entering the numerical values of the coefficients a, b, c, and d (not forgetting any zeros) into the first vector.

The negative root (depth) is meaningless. From the moment equation we see the gate will remain closed when $0.607 < x < 5.67$ ft.

Sample Problem 8.14 (by Mathcad Equation Solver)

Again we have 14 governing equations with 14 unknowns.

Known values: $L1 := 6000$ $D1 := \frac{15}{12}$ $e1 := 0.0004$ $ElevA := 250$ $g := 32.2$

$L2 := 1500$ $D2 := \frac{10}{12}$ $e2 := 0.0004$ $Q2 := 3.3$

$L3 := 4500$ $D3 := \frac{8}{12}$ $e3 := 0.0004$ $ElevC := 160$ $\nu := 1.217 \cdot 10^{-5}$

Guessed values: $f1 := 0.03$ $V1 := 10$ $R1 := 800000$ $Q1 := 10$ $h1 := 10$

$f2 := 0.03$ $V2 := 10$ $R2 := 800000$ $h2 := 10$

$f3 := 0.03$ $V3 := 10$ $R3 := 800000$ $Q3 := 10$ $h3 := 10$

Given

$$\frac{1}{\sqrt{f1}} = -2 \cdot \log\left(\frac{e1}{3.7 \cdot D1} + \frac{2.51}{R1 \cdot \sqrt{f1}}\right) \qquad R1 = \frac{D1 \cdot V1}{\nu} \qquad h1 = \frac{f1 \cdot L1 \cdot (V1)^2}{D1 \cdot 2 \cdot g} \qquad Q1 = \frac{V1 \cdot \pi \cdot (D1)^2}{4}$$

$$\frac{1}{\sqrt{f2}} = -2 \cdot \log\left(\frac{e2}{3.7 \cdot D2} + \frac{2.51}{R2 \cdot \sqrt{f2}}\right) \qquad R2 = \frac{D2 \cdot V2}{\nu} \qquad h2 = \frac{f2 \cdot L2 \cdot (V2)^2}{D2 \cdot 2 \cdot g} \qquad Q2 = \frac{V2 \cdot \pi \cdot (D2)^2}{4}$$

$$\frac{1}{\sqrt{f3}} = -2 \cdot \log\left(\frac{e3}{3.7 \cdot D3} + \frac{2.51}{R3 \cdot \sqrt{f3}}\right) \qquad R3 = \frac{D3 \cdot V3}{\nu} \qquad h3 = \frac{f3 \cdot L3 \cdot (V3)^2}{D3 \cdot 2 \cdot g} \qquad Q3 = \frac{V3 \cdot \pi \cdot (D3)^2}{4}$$

$h1 + h3 = ElevA - ElevC$ $\qquad Q1 = Q2 + Q3$

with constraints: $V1 > 0$ $V2 > 0$ $V3 > 0$ $f1 > 0$ $f2 > 0$ $f3 > 0$

$$\begin{bmatrix} Q1 \\ Q3 \\ V1 \\ V2 \\ V3 \\ R1 \\ R2 \\ R3 \\ f1 \\ f2 \\ f3 \\ h1 \\ h2 \\ h3 \end{bmatrix} := \text{Find}(Q1, Q3, V1, V2, V3, R1, R2, R3, f1, f2, f3, h1, h2, h3)$$

Results: $Q1 = 5.3368$ $V1 = 4.3488$ $R1 = 4.4667 \cdot 10^5$ $f1 = 0.0166$ $h1 = 23.3415$

$V2 = 6.0504$ $R2 = 4.1430 \cdot 10^5$ $f2 = 0.0177$ $h2 = 18.1287$

$Q3 = 2.0368$ $V3 = 5.8349$ $R3 = 3.1963 \cdot 10^5$ $f3 = 0.0187$ $h3 = 66.6585$

$ElevB := ElevA - h1 - h2$ Answer: $ElevB = 208.5298$

Note: With V and f constrained to be > 0, $\mathbb{R}$, h, and Q cannot be < 0.

Appendix E
References

There is a great volume of literature available on the various aspects of fluid mechanics and hydraulics. The results of original research are usually reported in papers published in technical journals. We list here selected journals that have published numerous articles dealing with fluids, their properties and applications; some of them have changed their names over time:

Experiments in Fluids, 1983–
Geophysical and Astrophysical Fluid Dynamics, 1977–
Geophysical Fluid Dynamics, 1970–1976
International Journal for Numerical Methods in Fluids, 1981–
International Journal of Numerical Methods for Heat & Fluid Flow, 1991–
Recent Advances in Numerical Methods in Fluids, 1980–
Journal of Computational Physics, 1966–
Journal of Fluid Mechanics, 1956–
Journal of Fluids and Structures, 1987–
Transactions of the ASME, 1880–
Journal of Applied Mechanics, ASME, 1935–
Journal of Fluids Engineering, ASME, 1973–
Journal of Turbomachinery, ASME, 1986–
Journal of Geophysical Research, Papers in Hydrology, 1961–
Journal of Geophysical Research, Oceans and Atmospheres, 1969–1983
Journal of Geophysical Research, Atmospheres, 1984–
Journal of Geophysical Research, Oceans, 1984–
Journal of Hydraulic Research, IAHR/AIHR, 1964–
Transactions of the ASCE, 1867–
Proceedings of the ASCE, 1873–1955
Journal of the Hydraulic Division, Proceedings of the ASCE, 1956–1982
Journal of Hydraulic Engineering, ASCE, 1983–
Journal of Non-Newtonian Fluid Mechanics, 1976–
Physics of Fluids, 1958–1988, 1994–
Physics of Fluids, A: Fluid Dynamics, 1989–1993
Transactions of the Institution of Civil Engineers (GB), 1836–1842
Journal of the Institution of Civil Engineers (GB), 1935–1951
Proceedings of the Institution of Civil Engineers (GB), 1952–1972
Proceedings of the Institution of Civil Engineers (GB), Part 1: Design and Construction, 1972–1991
Proceedings of the Institution of Civil Engineers (GB), Part 2: Research and Theory, 1972–1991
Proceedings of the Institution of Civil Engineers (GB), 1992–
Journal & Proceedings, Institution of Mechanical Engineers (London), 1847–1982
Proceedings, Institution of Mechanical Engineers (London), Part C: Mechanical Engineering Science, 1983–

For the convenience of students we next include a list of books covering many different topics of fluid mechanics and its engineering applications. This list by no means includes all the important books that have been written; we intend merely to provide a representative list. We encourage students to "probe deeper" and to widen their horizons by further reading.

1. Ackers, P., W. R. White, A. J. Harrison, and J. A. Perkins. *Weirs and Flumes for Flow Measurement.* Wiley, New York, 1978.
2. Anderson, J. D. *Computational Fluid Dynamics: The Basics with Applications.* McGraw-Hill, New York, 1995.
3. Anderson, J. D. *Fundamentals of Aerodynamics,* 2d ed. McGraw-Hill, New York, 1991.
4. Anderson, J. D. *Modern Compressible Flow: With Historical Perspective,* 2d ed. McGraw-Hill, New York, 1989 & 1990.
5. Barenblatt, G. I. *Dimensional Analysis,* translated from Russian by P. Makinen. Gordon and Breach, New York, 1987.
6. Batchelor, G. K. *An Introduction to Fluid Dynamics.* Cambridge University Press, Cambridge, England, 2000.
7. Bear, J. *Dynamics of Fluids in Porous Media.* Dover, New York, 1988.
8. Bertin, J. J., and M. L. Smith. *Aerodynamics for Engineers,* 3d ed. Prentice Hall, Upper Saddle River, NJ, 1998.
9. Böhme, G. *Non-Newtonian Fluid Mechanics.* Elsevier, New York, 1987.
10. Bos, M. G., J. A. Replogle, and A. J. Clemmens. *Flow Measuring Flumes for Open Channel Systems.* Wiley, New York, 1984.
11. Brater, E. F., H. W. King, J. E. Lindell, and C. Y. Wei. *Handbook of Hydraulics,* 7th ed. McGraw-Hill, New York, 1996.
12. Cengel, Y. A., and M. A. Boles. *Thermodynamics: An Engineering Approach,* 3d ed. WCB/McGraw-Hill, New York, 1998.
13. Chadwick, A. J. *Hydraulics in Civil and Environmental Engineering,* 3d ed. Spon, New York, 1998.
14. Chaudhry, M. H. *Applied Hydraulic Transients,* 2d ed. Van Nostrand Reinhold, New York, 1987.
15. Chaudhry, M. H. *Open Channel Flow.* Prentice-Hall, Englewood Cliffs, NJ, 1993.
16. Cheremisinoff, N. P. (ed.). *Encyclopedia of Fluid Mechanics.* Gulf Publishing, Houston, TX, 1986–1990.
17. Cheremisinoff, N. P. *Fundamentals of Wind Energy.* Ann Arbor Science, Ann Arbor, MI, 1978.
18. Chorin, A. J., and J. E. Marsden. *A Mathematical Introduction to Fluid Mechanics,* 3d ed. Springer-Verlag, New York, 1994.
19. Chow, C.-Y. *An Introduction to Computational Fluid Mechanics.* Seminole, Boulder, CO, 1983.
20. Chow, V. T. *Open-Channel Hydraulics.* McGraw-Hill, New York, 1959.
21. Churchill, S. W., and H. Brenner. *Viscous Flows: The Practical Use of Theory.* Butterworth-Heinemann, Boston, 1988.
22. Clift, R., J. R. Grace, and M. E. Weber. *Bubbles, Drops, and Particles.* Academic Press, New York, 1978.
23. Colt Industries, Inc. *Hydraulic Handbook: Fundamental Hydraulics and Data Useful in the Solution of Pump Application Problems,* 11th ed. Fairbanks Morse Pump Division, Kansas City, KS, 1979.
24. Crowe, C. T., D. F. Elger, and J. A. Roberson. *Engineering Fluid Mechanics,* 7th ed. Wiley, New York, 2000.
25. Eggleston, D. M., and F. S. Stoddard. *Wind Turbine Engineering Design.* Van Nostrand Reinhold, New York, 1987.
26. Evett, J. B., and C. Liu. *Fundamentals of Fluid Mechanics.* McGraw-Hill, New York, 1987.
27. Fischer, H. B., E. J. List, R. C. Y. Koh, J. Imberger, and N. H. Brooks. *Mixing in Inland and Coastal Waters.* Academic Press, New York, 1979.
28. Fox, R. W., and A. T. McDonald. *Introduction to Fluid Mechanics,* 5th ed. Wiley, New York, 1998.
29. Freeze, R. A., and J. A. Cherry. *Groundwater.* Prentice-Hall, Englewood Cliffs, NJ, 1979.
30. French, R. H. *Open-Channel Hydraulics.* McGraw-Hill, New York, 1985.
31. *Friction Factors for Large Conduits Flowing Full,* Engineering Monograph No. 7. U.S. Department of the Interior, Bureau of Reclamation, Denver, CO, 1977.
32. Garay, P. N. *Pump Application Desk Book.* Fairmont Press, Liburn, GA, 1990.
33. Goldstein, R. J. (ed.). *Fluid Mechanics Measurements,* 2d ed. Taylor & Francis, Washington, DC, 1996.
34. Graf, W. H. *Hydraulics of Sediment Transport.* McGraw-Hill, New York, 1971,

reprinted by Water Resource Publications, Littleton, CO, 1984.
5. Granger, R. A. *Experiments in Fluid Mechanics.* Holt, Rinehart and Winston, New York, 1988.
6. Gulliver, J. S., and R. E. A. Arndt (eds.). *Hydropower Engineering Handbook.* McGraw-Hill, New York, 1991.
7. Hamrock, B. J. *Fundamentals of Fluid Film Lubrication.* McGraw-Hill, New York, 1994.
8. Henderson, F. M. *Open Channel Flow.* Macmillan, New York, 1966.
9. Herring, J. R., and J. C. McWilliams. *Lecture Notes on Turbulence.* World Scientific, Singapore, 1989.
10. Hinze, J. O. *Turbulence,* 2d ed. McGraw-Hill, New York, 1975.
11. Holton, J. R. *An Introduction to Dynamic Meteorology.* Academic Press, San Diego, CA, 1992.
12. Horváth, I. *Hydraulics in Water and Waste-Water Treatment Technology.* Wiley, New York, 1994.
13. Hucho, W.-H. (ed.). *Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering,* 4th ed. Society of Automotive Engineers, Warrendale, PA, 1998.
14. Hydraulic Institute. *Engineering Data Book,* 2d ed. Hydraulic Institute, Cleveland, OH, 1990.
15. Hydraulic Institute. *Hydraulic Institute Standards for Centrifugal, Rotary, & Reciprocating Pumps,* 15th ed. Hydraulic Institute, Parsippany, NJ, 1994.
16. Hydraulic Institute. *Pipe Friction Manual,* 3d ed. Hydraulic Institute, Cleveland, OH, 1975.
17. Hydraulic Institute. *Standards of Hydraulic Institute,* 14th ed. Hydraulic Institute, New York, 1983.
18. Idel'chik, I. E. *Handbook of Hydraulic Resistance,* 3d ed. CRC Press, Boca Raton, FL, 1994.
19. Johnson, R. W. (ed.). *The Handbook of Fluid Dynamics.* CRC Press, Boca Raton, FL, 1998.
20. Kirchhoff, R. H. *Potential Flows: Computer Graphic Solutions.* M. Dekker, New York, 1985.
21. Kline, S. J. *Similitude and Approximation Theory.* Springer-Verlag, New York, 1986.
52. Knapp, R. T., J. W. Daily, and F. G. Hammitt. *Cavitation.* McGraw-Hill, New York, 1970, reprinted by Institute of Hydraulic Research, University of Iowa, Iowa City, IA, 1979.
53. Kuethe, A. M., and C.-Y. Chow. *Foundations of Aerodynamics: Bases of Aerodynamic Design,* 5th ed. Wiley, New York, 1998.
54. Lamb, H. *Hydrodynamics,* 6th ed. Cambridge University Press, Cambridge, England, 1932, reprinted 1993.
55. Langhaar, H. L. *Dimensional Analysis and the Theory of Models.* Wiley, New York 1951, reprinted by Krieger, Huntington, NY, 1980.
56. Liggett, J. A. *Fluid Mechanics.* McGraw-Hill, New York, 1994.
57. Lighthill, M. J. *Waves in Fluids.* Cambridge University Press, London, 1978.
58. Linsley, R. K., J. B. Franzini, D. L. Freyberg, and G. Tchobanoglous. *Water Resources Engineering,* 4th ed. McGraw-Hill, New York, 1992.
59. Lobanoff, V. S., and R. R. Ross. *Centrifugal Pumps: Design and Application,* 2d ed. Gulf Publishing, Houston, TX, 1992.
60. Merzkirch, W. *Flow Visualization,* 2d ed. Academic Press, Orlando, FL, 1987.
61. Miller, R. W. *Flow Measurement Engineering Handbook,* 3d ed. McGraw-Hill, New York, 1996.
62. Moran, M. J., and H. N. Shapiro. *Fundamentals of Engineering Thermodynamics,* 4th ed. Wiley, New York, 2000.
63. Munson, B. R., D. F. Young, and T. H. Okiishi. *Fundamentals of Fluid Mechanics,* 3d ed. Wiley, New York, 1998.
64. National Committee for Fluid Mechanics Films. *Illustrated Experiments in Fluid Mechanics.* MIT Press, Cambridge, MA, 1972.
65. Novak, P., and J. Cábelka. *Models in Hydraulic Engineering: Physical Principles and Design Applications.* Pitman, Boston, 1981.
66. Olfe, D. B. *Fluid Mechanics Programs for the IBM PC.* McGraw-Hill, New York, 1987.
67. Pai, S.-I., and S. Luo. *Theoretical and Computational Dynamics of a Compressible Flow.* Van Nostrand Reinhold, New York, 1991.

68. Parker, S. P. (ed.). *Fluid Mechanics Source Book.* McGraw-Hill, New York, 1988.
69. Peyret, R., and T. D. Taylor. *Computational Methods for Fluid Flow.* Springer-Verlag, New York, 1990.
70. Potter, M. C., and D. C. Wiggert. *Mechanics of Fluids,* 2d ed. Prentice Hall, Upper Saddle River, NJ, 1997.
71. Rahman, M. *Water Waves: Relating Modern Theory to Advanced Engineering Applications.* Oxford University Press, Oxford, 1995.
72. Raudkivi, A. J. *Loose Boundary Hydraulics,* 4th ed. Balkema, Rotterdam, 1998.
73. Roberson, J. A., J. J. Cassidy, and M. H. Chaudhry. *Hydraulic Engineering,* 2d ed. Wiley, New York, 1998.
74. Rouse, H., and S. Ince. *History of Hydraulics.* Dover, New York, 1963.
75. Saad, M. A. *Compressible Fluid Flow,* 2d ed. Prentice Hall, Englewood Cliffs, NJ, 1993.
76. Saad, M. A. *Thermodynamics Principles and Practices.* Prentice Hall, Upper Saddle River, NJ, 1997.
77. Sabersky, R. H., A. J. Acosta, E. G. Hauptmann, and E. M. Gates. *Fluid Flow: A First Course in Fluid Mechanics,* 4th ed. Prentice Hall, Upper Saddle River, NJ, 1999.
78. Schetz, J. A. *Boundary Layer Analysis.* Prentice-Hall, Englewood Cliffs, NJ, 1993.
79. Schlichting, H. *Boundary Layer Theory,* 8th ed. Springer, New York, 2000.
80. Shames, I. H. *Mechanics of Fluids,* 3d ed. McGraw-Hill, New York, 1992.
81. Sharpe, G. J. *Solving Problems in Fluid Dynamics.* Longman Scientific and Technical, Harlow, Essex, England, 1994.
82. Sherman, F. S. *Viscous Flow.* McGraw-Hill, New York, 1990.
83. Sonntag, R. E., C. Borgnake, and G. J. Van Wylen. *Fundamentals of Thermodynamics,* 5th ed. Wiley, New York, 1998.
84. Soo, S. L. *Particulates and Continuum: Multiphase Fluid Dynamics.* Hemisphere, New York, 1989.
85. Stepanoff, A. J. *Centrifugal and Axial Flow Pumps: Theory, Design, and Application,* 2d ed. Wiley, New York, 1957, reprinted by Krieger, Malabar, FL, 1993.
86. Stewart, H. L. *Pumps,* 5th ed. Macmillan, New York, 1991.
87. Stoker, J. J. *Water Waves: The Mathematical Theory with Applications.* Wiley, New York, 1992.
88. Street, R. L., G. Z. Watters, and J. K. Vennard. *Elementary Fluid Mechanics,* 7th ed. Wiley, New York, 1996.
89. Streeter, V. L., E. B. Wylie, and K. W. Bedford. *Fluid Mechanics,* 9th ed. WCB/McGraw-Hill, New York, 1998.
90. Sullivan, J. A. *Fluid Power: Theory and Applications,* 3d ed. Prentice-Hall, Englewood Cliffs, NJ, 1989.
91. Sutton, G. P. *Rocket Propulsion Elements,* 7th ed. Wiley, New York, 2001.
92. Tannehill, J. C., D. A. Anderson, and R. H. Pletcher. *Computational Fluid Mechanics and Heat Transfer,* 2d ed. Taylor & Francis, Washington, DC, 1997.
93. Task Committee on Hydraulic Modeling (R. Ettema, chmn.). *Hydraulic Modeling: Concepts and Practice.* ASCE Manuals and Reports on Engineering Practice No. 97, American Society of Civil Engineers, Reston, VA, 2000.
94. Tokaty, G. A. *A History and Philosophy of Fluid Mechanics.* Foulis, Henley-on-Thames, England, 1971.
95. Tullis, J. P. *Hydraulics of Pipelines: Pumps, Valves, Cavitation, Transients.* Wiley, New York, 1989.
96. Turton, R. K. *Rotodynamic Pump Design.* Cambridge University Press, Cambridge, England, 1994.
97. Upp, E. L. *Fluid Flow Measurement: A Practical Guide to Accurate Flow Measurement.* Gulf Publishing, Houston, 1993.
98. U. S. Committee on Extension to the Standard Atmosphere. *U.S. Standard Atmosphere, 1976.* Government Printing Office, Washington, DC, 1976. Also produced by the National Oceanic and Atmospheric Administration.
99. Vallentine, H. R. *Applied Hydrodynamics,* 2d ed. Butterworth, London, and Plenum Press, New York, 1967.
100. Walski, T. M. *Analysis of Water Distribution Sytems.* Krieger, Malabar, FL, 1992.
101. Warnick, C. C. *Hydropower Engineering.* Prentice-Hall, Englewood Cliffs, NJ, 1984.

102. Watters, G. Z. *Analysis and Control of Unsteady Flow in Pipelines,* 2d ed. Butterworths, Boston, 1984.
103. White, F. M. *Fluid Mechanics,* 4th ed. WCB/McGraw-Hill, New York, 1999.
104. White, F. M. *Viscous Fluid Flow,* 2d ed. McGraw-Hill, New York, 1991.
105. Wylie, E. B., and V. L. Streeter. *Fluid Transients in Systems.* Prentice-Hall, Englewood Cliffs, NJ, 1993.
106. Yalin, M. S. *River Mechanics.* Pergamon, Oxford, 1992.
107. Yang, W.-J. (ed.). *Handbook of Flow Visualization.* Hemisphere, New York, 1989.
108. Yaws, C. L. *Handbook of Viscosity,* 3 vols. Gulf Publishing, Houston, TX, 1995–1997.
109. Zipparro, V. J., and H. Hasen (eds.). *Davis' Handbook of Applied Hydraulics,* 4th ed. McGraw-Hill, New York, 1993.
110. Záruba, J. *Water Hammer in Pipe-Line Systems.* Elsevier, Amsterdam, 1993.

Appendix F
Answers to Exercises

Chapter 1

1.5.1 All are F or MLT^{-2}.
1.5.2 All are L^3/T.
1.5.3 All are L^2/T.
1.5.4 All are L.
1.5.5 All are M/T.
1.5.6 (*a*) 8.33 lb; (*b*) 37.0 N; (*c*) 3.70×10^6 dyne.
1.5.7 (*a*) 2.20 lb; (*b*) 9.81 N; (*c*) 981×10^3 dyne.
1.5.8 (*a*) 36.9 cfs; (*b*) 1.905 m^3/s.
1.5.9 (*a*) 27.8 m/s; (*b*) 91.1 ft/sec.

Chapter 2

2.3.1 1.615 slugs/ft^3.
2.3.2 826 kg/m^3.
2.3.3 0.0859 lb/ft^3.
2.3.4 14.01 N/m^3.
2.3.5 1.631 kg/m^3, 0.613 m^3/kg, 1.333.
2.3.6 2.44 slugs/ft^3, 1.260, 12.36 kN/m^3.
2.3.7 1.335 slugs/ft^3, 0.749 ft^3/slug, 0.689.
2.5.1 2500 MN/m^2.
2.5.2 6400 psi.
2.5.3 1.34%.
2.5.4 66.2 MPa.
2.5.5 497 mm.
2.6.1 (*a*) About 61.4 pcf; (*b*) about 61.8 pcf.
2.6.2 9.59 kN/m^3, 9.65 kN/m^3.
2.6.3 0.338% increase, 1.050 lb.
2.6.4 4.07 m^3.
2.7.1 298 $m^2/(s^2 \cdot K)$, carbon monoxide or nitrogen.
2.7.2 6600 ft^3.
2.7.3 0.233 pcf, 0.00724 slug/ft^3, 138.2 ft^3/slug.
2.7.4 36.0 N/m^3, 3.67 kg/m^3, 0.273 m^3/kg.
2.7.5 0.0452 lb/ft^3, 713 ft^3/slug, 2858 ft·lb/(slug·°R).
2.7.6 O_2: 3.09 psia, 0.01842 pcf; N_2: 11.47 psia, 0.0600 pcf.
2.7.7 Q.E.D.
2.8.1 Methane 22.0 psi, N_2: 22.4 psi, methane is more compressible.
2.8.2 Methane 140.0 kPa, N_2: 140.0 kPa, equal compressibilities.
2.8.3 (*a*) 50.0 kPa abs; (*b*) 34.7 kPa abs, −63.0°C.
2.8.4 362 psia.
2.11.1 4.8×10^{-6} ft^2/sec, 4.5×10^{-7} m^2/s.
2.11.2 375°F.
2.11.3 1:53.7, 15.5:1.
2.11.4 240 N.
2.11.5 276 lb.
2.11.6 15.32 mN·s/m^2, 17.07×10^{-6} m^2/s.
2.11.7 (*a*) 3.21; (*b*) 290; (*c*) 1.782.
2.11.8 (*a*) 4.50 N; (*b*) 5.24 N.
2.11.9 0.00655 kN·m/s, 6.55 J/s, 22.4 Btu/hr, 4.83 ft·lb/sec, 0.00878 hp.
2.11.10 $\pi\mu\omega d^4/(32\Delta h)$.
2.11.11 At $y = 0$ in, $du/dy = 20$ sec^{-1}, $\tau = 0.251$ lb/ft^2.
At $y = 3$ in, $du/dy = 15$ sec^{-1}, $\tau = 0.1880$ lb/ft^2.
At $y = 6$ in, $du/dy = 10$ sec^{-1}, $\tau = 0.1253$ lb/ft^2.
At $y = 9$ in, $du/dy = 5$ sec^{-1}, $\tau = 0.0627$ lb/ft^2.
At $y = 12$ in, $du/dy = 0$ sec^{-1}, $\tau = 0$ lb/ft^2.
2.11.12 (*a*) Because ν varies strongly with pressure; (*b*) 46.3×10^{-6} ft^2/sec.
2.12.1 About 4.44 in.
2.12.2 About 13.89 mm.
2.12.3 0.334 in.
2.12.4 37.8 mm.
2.12.5 0.164 in, cf. 0.130 in from Fig. 2.7.
2.13.1 311.6 mb abs.
2.13.2 198.6°F.

Chapter 3

3.2.1 6890 psi.
3.2.2 46700 kPa.
3.2.3 55.2 lb/ft^3, 1.714 slugs/ft^3, 0.885.
3.2.4 57.9 psi.
3.3.1 16 kPa (=kN/m^2), 65.1 kPa.
3.3.2 0.839 psi, 3.87 psi.
3.3.3 8.44 km.

3.4.1 54.3 psi.
3.4.2 391 kPa.
3.4.3 34.1 kPa abs.
3.4.4 8.06 psia.
3.4.5 70.2 kPa abs.
3.4.6 33.14 ft, 30.35 ft.
3.5.1 36.64 ft.
3.5.2 7.07 m.
3.5.3 2148 mb abs, 1.615 mHg abs.
3.5.4 34.7 in.
3.5.5 34.8 psi.
3.5.6 280 kPa.
3.5.7 (*a*) 25.9 psi; (*b*) 26.1 psi.
3.5.8 43.8 ft, 5.31 ft.
3.5.9 (*a*) About 0.65 psi ($p_A > p_B$); (*b*) 54.8 inHg, 76.5 ft kerosene.
3.5.10 (*a*) 9.19 ft of water, 3.98 psi; (*b*) 85.2 in.
3.5.11 (*a*) 2.76 m, 27.0 kPa; (*b*) 2.13 m.
3.7.1 $5d/8$.
3.7.2 $0.5d$.
3.7.3 $0.75d$.
3.7.4 $(6a^2 + 8ad + 3d^2)/(6a + 4d)$.
3.7.5 $3b/8$.
3.7.6 61.6 kN, 2.03 m.
3.7.7 5280 lb, 6.84 ft.
3.7.8 (*a*) 1248 lb, 1.333 ft; (*b*) 2500 lb, 2.17 ft.
3.7.9 (*a*) 1.800 m; (*b*) $B_x = 77.4$ kN/m, $B_y = 0$, $N_x = 17.59$ kN/m.
3.7.10 4240 ft·lb.
3.7.11 5.31 ft; yes.
3.7.12 (*a*) 392 kN, 2.67 m; (*b*) 589 kN, 3.44 m; (*c*) 20 000 kN, 102.0 m.
3.7.13 2.95 ft, $x_p = 0.850$ ft.
3.7.14 1.123 m, $x_p = 0.269$ m.
3.7.15 1177 kN for any θ; 0.750 m at 90°, 0.650 m at 60°, 0.375 m at 30°, 0 m at 0°.
3.7.16 392 lb, 147.0 lb, 245 lb; (*a*) 21.0 psi; (*b*) 7.88 psi.
3.8.1 2360 psi.
3.8.2 11,520 lb/ft at 5 ft below top, 6910 lb/ft at 3.0 ft to left of AB.
3.8.3 175 kN/m at 1.75 m below top, 100 kN/m at 1.0 m to left of AB.
3.8.4 1313 kN/m, 65 600 kPa, 9520 psi.
3.8.5 740 lb.
3.8.6 (*a*) 3920 lb, 1046 lb; (*b*) 4120 lb, 1354 lb; (*c*) 10,860 lb, 1.257 lb; (*d*) 3950 lb, 523 lb.
3.8.7 3120 lb/ft at 6.67 ft depth, 1980 lb/ft at 3.40 ft to left of AB.
3.8.8 60.1 kN/m at 2.33 m depth, 37.9 kN/m at 1.123 m to left of AB.
3.8.9 (*a*) 706 kN at 3.44 m depth; (*b*) 609 kN at 1.107 m to right of N; (*c*) 933 kN through intersection of F_x and F_y, at 40.8° to horizontal.
3.8.10 (*a*) 43,800 lb at 7.54 ft depth; (*b*) 35,900 lb at 2.44 ft to right of N; (*c*) 56,700 lb through intersection of F_x and F_y, at 39.4° to horizontal.
3.9.1 340 lb.
3.9.2 6.06 kN.
3.9.3 0.875, 10.26%.
3.9.4 0.0849 m³, 12 160 N/m³.
3.9.5 2.53 in.
3.9.6 (*a*) 0.962 in deep; (*b*) 4.95 in deep.
3.9.7 (*a*) 6.66 kN, 15.41%; (*b*) 18.00 kN.
3.9.8 161.3 mm, 99.4 mm, 58.2 mm; no, $dy/ds = -7000/(28.3\ s^2)$.
3.9.9 (*a*) 185.5 mm; (*b*) 19.63 L.
3.9.10 (*a*) 1.307 ft; (*b*) 4.47 lb.
3.9.11 (*a*) 31.5 lb/ft³; (*b*) yes; (*c*) 242 lb·ft ↶.
3.9.12 (*a*) 5240 N/m³; (*b*) yes; (*c*) 298 N·m ↶.
3.10.1 0.364 psi.
3.10.2 3040 Pa.
3.10.3 2.70 psi.
3.10.4 33.5 kPa.
3.10.5 135 ft³.
3.10.6 220 m³.
3.10.7 225 psf; top left 225 psf, bottom right 200 psf, bottom left 425 psf.
3.10.8 36.0 kPa; bottom right 32.0 kPa, top left 36.0 kPa, bottom left 68.0 kPa.

Chapter 4

4.3.1 (*a*) Unsteady, nonuniform; (*b*) unsteady (but steady with respect to rotating frame), nonuniform; (*c*) steady, uniform; (*d*) almost steady, nonuniform; (*e*) unsteady, nonuniform; (*f*) unsteady, uniform.
4.5.1 Q.E.D.
4.5.2 7.41 fps, 4.63 fps.
4.5.3 14.14 L/s, 0.014 14 m³/s, 19.79 kg/s, 0.1942 kN/s.
4.5.4 0.0511, 0.012 77, and 0.003 19 ft/sec.
4.5.5 0.015 92, 0.003 98, and 0.000 995 m/s.
4.7.1 350 cfs.
4.7.2 10.30 m³/s.
4.7.3 −0.000 1981 lb/ft³ per sec.
4.10.1 0.597 m/s at D, 4.08 m/s at B.
4.10.2 About 0.16 m/s.
4.10.3 (*a*) About 16.6 ft/sec; (*b*) About 12.4 ft/sec.
4.10.4 (*a*) About 5.5 m/s; (*b*) zero at stagnation point.
4.12.1 5.39 L/T.
4.12.2 1.204 L/T^2; converging.
4.12.3 $a_x = 2x$, $a_y = 2y$, 3.61 L/T, 6.32 L/T^2.
4.12.4 $a_x = 4x$, $a_y = y$; 6.32 L/T, 12.17 L/T^2.
4.12.5 $a_x = 2xy$, $a_y = 2y^2 + x^2y$; 8.49 L/T, 32.3 L/T^2.
4.12.6 1.210 and 0 ft/sec².
4.12.7 0.375 and 0 m/s².
4.12.8 0.1273 ft/sec, 0.012 97 ft/sec² at 2.5 ft; 0.0318 ft/sec, 0.000 405 ft/sec² at 5 ft.
4.13.1 16.16 L/T, 5.39 L/T^2.

4.13.2 68.8 L/T, 1988 L/T^2.
4.13.3 1.920 and 1.200 ft/sec².
4.13.4 0.836 and 0.600 m/s².

Chapter 5

5.1.1 17/16 = 1.063.
5.1.2 10/9 = 1.111.
5.1.3 54/35 = 1.543.
5.2.1 9.33 fps, 21.0 fps, 20.5 ft.
5.2.2 1.365 cfs, −15.0 ft.
5.2.3 0.1565 m³/s, −5.0 m.
5.2.4 318 kPa.
5.2.5 0.978 cfs.
5.3.1 (*a*) 79.9 ft; (*b*) 85.5 ft.
5.3.2 (*a*) 25.2 m; (*b*) 27.4 m.
5.3.3 (*a*) 58.9 ft; (*b*) 74.1 ft.
5.3.4 (*a*) 13.14 ft, from B to A; (*b*) 12.35 ft, from A to B.
5.3.5 2.09 m, from B to A.
5.3.6 15.07 ft.
5.3.7 20.9 N/m².
5.4.1 33.9 psi.
5.4.2 219 kPa.
5.4.3 6.72 psi.
5.6.1 (*a*) 2330 ft·lb/lb; (*b*) 3.00 Btu/lb.
5.6.2 (*a*) 854 J/N; (*b*) 204 cal/N.
5.6.3 0.226 Btu/lb, 0.226°R = 0.226°F.
5.6.4 52 J/N, 0.1218 K = 0.1218°C.
5.7.1 0, 0.0344 Btu/lb gain.
5.7.2 0, 8.15 J/N loss.
5.7.3 1.161 Btu/lb, 1990 ft·lb/slug loss.
5.7.4 269 J/N, 232 N·m/kg loss.
5.9.1 8880 hp.
5.9.2 7740 kW.
5.9.3 19.38 hp.
5.9.4 6.83 kW.
5.9.5 217 hp, 4780 hp.
5.9.6 1383 kW, 10 350 kW.
5.10.1 41.3 cfs.
5.10.2 0.279 m³/s.
5.10.3 1.179 ft.
5.10.4 0.371 m.
5.10.5 1.749 m³/s.
5.13.1 0.1238 m³/s, −4.10 m.
5.13.2 4.02 cfs, −12.00 ft.
5.13.3 8.70 m.
5.13.4 29.3 ft.
5.13.5 15,760 hp.
5.13.6 13 730 kW.
5.13.7 −26.2 ft, 33.8 ft, 71.8 ft.
5.13.8 0.0926 m, 15.09 m, 29.8 m.
5.14.1 55.7 cfs per ft.
5.14.2 2.75 m³/s per m.
5.14.3 9.51 ft.
5.14.4 2.41 m.
5.15.1 2.21 m/s, 3.13 m/s, 3.84 m/s, 4.43 m/s.
5.15.2 39.5° or 78.0°.
5.15.3 6.03 ft/sec.
5.16.1 10.87 psi.
5.16.2 111.7 kN/m².
5.17.1 245 ft (any fluid); (*a*) 0.1294 psi; (*b*) 106.1 psi; (*c*) 78.3 psi.
5.17.2 267 kPa.
5.17.3 171.4 rpm for both.
5.18.1 0.1372 m³/s, 12.38 m/s at 16°, 85.4 kPa.
5.18.2 5.41 cfs, 35.7 ft/sec at 16°, 8.24 psi.

Chapter 6

6.3.1 4/3 = 1.333.
6.3.2 49/48 = 1.021.
6.4.1 1283 lb to the right.
6.4.2 11.00 kN/m to the right.
6.4.3 742 lb/ft to the right.
6.4.4 273 lb/ft to the right.
6.4.5 42.9 kN to the right.
6.5.1 3910 lb to the right; 53.4 ft.
6.5.2 3.98 kN to the right; 28.5 m.
6.5.3 10.83°.
6.5.4 7450 lb at 45° toward the SE.
6.5.5 26.0 kN at 45° toward the SE.
6.5.6 5.27 lb to the right.
6.6.1 95.5 lb.
6.6.2 785 N.
6.6.3 (*a*) 60.7 psi; (*b*) 5.53 psi.
6.6.4 (*a*) 313 kN/m²; (*b*) 25.0 kN/m².
6.6.5 $(2\gamma A/g)V^2 \sin(\theta/2)$.
6.6.6 (*a*) 135.9 lb →; (*b*) 86.5 lb ↑; (*c*) 161.1 lb at 32.5° ↗.
6.6.7 (*a*) 1117 N →; (*b*) 712 N ↑; (*c*) 1325 N at 32.5° ↗.
6.8.1 Q.E.D.
6.8.2 (*a*) 942 lb; (*b*) 5890 lb.
6.8.3 (*a*) 6.13 kN; (*b*) 28.1 kN.
6.8.4 26.8°, 29.0°, 35.7°, 51.3°, 99.4 fps.
6.8.5 25.7°, 27.8°, 34.2°, 49.1°, 33.3 m/s.
6.8.6 $F_x = 996$ lb, $F_y = 252$ lb.
6.9.1 69.1 lb.
6.9.2 557 N.
6.10.1 1724 lb.
6.10.2 3.50 kN.
6.11.1 36.2 ft·lb; 65.8 hp.
6.11.2 47.7 N·m; 71.6 kW.
6.13.1 Q.E.D.
6.14.1 928 rpm.
6.14.2 0.0756 cfs; 15.70 ft·lb; zero power.
6.14.3 0.002 23 m³/s; 21.8 N·m; zero power.
6.14.4 1414 rpm.
6.14.5 1770 rpm.
6.15.1 (*a*) 1.498 lb; (*b*) 0.848 psf; (*c*) 0.0365 hp.
6.15.2 (*a*) 16.69 N; (*b*) 6.56 N/m²; (*c*) 27.3 W.
6.15.3 (*a*) 150.0 cfs; (*b*) 45.4 lb.

Chapter 7

7.4.1 293,000.
7.4.2 1240.
7.4.3 35.8 ft/sec.
7.4.4 7240 ft/sec > sonic velocity, so model is unsuitable.
7.4.5 4.15 ft/sec, 0.231.
7.4.6 30 atm.
7.4.7 2.72 m/s, 5150 N.
7.4.8 743 m/s.
7.7.1 (*a*) $\tau/(\rho V^2)$; (*b*) $\sigma/(\rho L V^2)$.
7.7.2 (*a*) $(\Delta p)g/(\gamma V^2)$; (*b*) $F/(\rho L^2 V^2)$.
7.7.3 $P = CT\omega$.
7.7.4 $\Pi_1 = \mathbf{R}$, $\Pi_2 = \tau/(\rho V^2)$, so $\tau = \rho V^2 \phi(\mathbf{R})$.
7.7.5 $\Pi_1 = F_D/(\rho D^2 V^2)$, $\Pi_2 = \mathbf{R}$, $\Pi_3 = \mathbf{F}$, so $F_D = \rho D^2 V^2 \phi(\mathbf{F}, \mathbf{R})$.

Chapter 8

8.2.1 1.040 ft/sec.
8.2.2 0.247 m/s.
8.2.3 Laminar.
8.3.1 3.43 in.
8.3.2 Zero.
8.5.1 0.1877 psf.
8.5.2 9.13 N/m^2.
8.5.3 0.00594 ft/ft.
8.5.4 0.01010 m/m.
8.6.1 0.0848 ft/ft, 0.01276 hp/ft.
8.6.2 0.0813 m/m, 28.0 W/m.
8.7.1 0.746 cfs, 2660 ft.
8.7.2 $0.707r_0$.
8.7.3 $V = \frac{2}{3}V_c$.
8.8.1 27.6 ft.
8.8.2 0.1244 m.
8.9.1 (*a*) 0.0001429 psf, 0.388 psf; (*b*) 0.771 in.
8.9.2 (*a*) 11.12 N/m^2; (*b*) 2.08 N·s/m^2; (*c*) 0.00534 N/m^2; (*d*) 11.11 N/m^2.
8.10.1 0.0387 in.
8.10.2 (*a*) 0.000957 in; (*b*) 0.000558 in.
8.10.3 (*a*) 0.0459 mm; (*b*) 0.0267 mm.
8.10.4 0.01340 in.
8.10.5 0.643 mm.
8.10.6 (*a*) 8.51 μm; (*b*) yes.
8.11.1 0.767 m^3/s.
8.11.2 0.776.
8.11.3 8.81 ft/ft; 42.2 psf; 70.1 ft/sec.
8.12.1 (*a*) 0.0399, 0.0404, 0.0397; (*b*) 0.0259, 0.0257, 0.0266; (*c*) 0.0180, 0.0178, 0.0178; (*d*) For $\mathbf{R} = 20000$ (3.2%).
8.12.2 Left side = 8.74, right side = 8.68.
8.12.3 Left side = 8.74, right side = 8.72.
8.13.1 0.01204 ft/ft.
8.13.2 0.0232 m/m.
8.13.3 (*a*) 0.0241 mm; (*b*) 0.01967 mm.
8.13.4 (*a*) 3790 lb; (*b*) 485 lb.
8.15.1 (*a*) 0.0357 ft, 8.15 ft; (*b*) 0.582 ft, 133 ft; (*c*) 2.01 ft, 459 ft.
8.15.2 (*a*) 7.55 psi; (*b*) 205 ft.
8.15.3 0.1394 cfs.
8.15.4 35.2 L/s.
8.16.1 0.1393 cfs.
8.16.2 35.2 L/s.
8.17.1 (*a*) and (*b*) 0.414 m^3/s.
8.17.2 (*a*), (*b*), and (*c*) 0.1393 cfs.
8.17.3 (*a*), (*b*), and (*c*) 35.2 L/s.
8.18.1 103.2.
8.18.2 103.3.
8.18.3 0.011153.
8.22.1 (*a*) 18.75; (*b*) 0.938; (*c*) 0.0469.
8.22.2 (*a*) 19.85; (*b*) 0.794; (*c*) 0.0397.
8.22.3 (*a*) 5.28 ft; (*b*) 97.9%.
8.22.4 1.474 m.
8.22.5 11.31 ft, 4.51 psi.
8.24.1 (*a*) 0.778 m; (*b*) 1.326 m; (*c*) 0.826 m, 0.232 m.
8.24.2 (*a*) 1.186 cfs; (*b*) 1.294 cfs.
8.26.1 6.15 ft.
8.26.2 2.64 m.
8.27.1 5.12 cfs, 75.3 psi.
8.27.2 94.8 L/s, 586 kN/m^2.
8.27.3 (*a*) 2.30 psi; (*b*) −1.882 psi.
8.27.4 (*a*) 20.6 kN/m^2; (*b*) −15.39 kN/m^2.
8.27.5 (*a*) 20.6 cfs; (*b*) 12.33 cfs.
8.27.6 $Q = 19.81$ cfs, $f = 0.0304$.
8.28.1 72,900 hp.
8.28.2 216 hp.
8.28.3 143.8 kW.
8.28.4 4.58 cfs.
8.28.5 (*a*) 112.4 ft; (*b*) 76.5 hp.
8.28.6 (*a*) 34.1 m; (*b*) 56.4 kW.
8.28.7 30.1 hp.
8.28.8 20.3 kW.
8.28.9 10.53 hp.
8.28.10 8.25 kW.

Chapter 9

9.3.1 3470; fair agreement with 2000.
9.3.2 0.0615 and 0.0435 lb/ft^2.
9.3.3 (*a*) 0.552 N/m^2, 3.03 mm; (*b*) 0.391 N/m^2, 4.28 mm; (*c*) 1.405 N.
9.4.1 (*a*) 17.53 ft/sec; (*b*) 19.35 ft/sec.
9.4.2 0.00293, 0.00300 (within 2.6%).
9.4.3 (*a*) 0.0337 lb/ft^2; (*b*) 0.0236 lb/ft^2; (*c*) 0.0209 lb/ft^2.
9.4.4 (*a*) 1.999 N/m^2, 9.52 mm; (*b*) 1.740 N/m^2, 16.57 mm; (*c*) 3.92 N; 2.8–4.5 times greater.
9.4.5 65.9 hp.
9.5.1 0.1124 lb.

9.5.2 (*a*) 1.728 lb; (*b*) 1.084 in.
9.5.3 (*a*) 0.214 mm; (*b*) 415 N.
9.7.1 0.374 lb·sec/ft^2.
9.7.2 8.41 N·s/m^2.
9.7.3 0, 1.250, 5.28, 12.50 N.
9.7.4 3.35 hp, 0.669 hp.
9.7.5 94.0 lb, about one-third of the friction drag.
9.7.6 17.68 ft.
9.7.7 5.15 m.
9.7.8 6630 N, 33.1 m/s.
9.8.1 8.47 ft·lb.
9.8.2 (*a*) 2455 Hz; (*b*) 2452 Hz.
9.9.1 86.6 lb/ft.
9.10.1 At −23.4°; 9690 lb.
9.10.2 At −21.3°; 83.9 kN.
9.12.1 (*a*) 0.279; (*b*) 3.70 m/s; (*c*) 3200 N.
9.13.1 2.58°.
9.13.2 2.58°.
9.13.3 (*a*) 116.7 ft/sec; (*b*) 13.37 hp; (*c*) 140.0 ft^2/sec; (*d*) 22.3 ft^2/sec.
9.14.1 (*a*) −451 ft/sec^2; (*b*) −471 ft/sec^2.
9.14.2 (*a*) −96.0 m/s^2; (*b*) −102.9 m/s^2.

Chapter 10

10.3.1 82.7, 308, 703, 1298 cfs.
10.3.2 1378 cfs.
10.3.3 1.101 m^3/s.
10.3.4 1.251 m.
10.3.5 0.000648, or 3.42 ft/mile.
10.3.6 0.00396.
10.3.7 0.01783.
10.3.8 0.01481.
10.3.9 8730 cfs.
10.3.10 134.6 m^3/s.
10.4.1 13.55 ft/sec, 2.58 ft.
10.4.2 5.22, 5.76, 6.30, 6.62, 7.02 (max) m/s.
10.5.1 3.33%.
10.5.2 2.80%.
10.5.3 22.9%; no.
10.6.1 7.57 ft; 11.88 ft versus 14 ft.
10.6.2 (*a*) 98.5 cfs; (*b*) 84.9 cfs; (*c*) form (*a*) has 16% more capacity yet requires only 80% of the lumber.
10.6.3 (*a*) 3.32 m^3/s; (*b*) 2.86 m^3/s; As Exer. 10.6.2.
10.7.1 Approx. 237 cfs, approx. 3.55 ft/sec.
10.7.2 (*a*) Approx. 330 mm; (*b*) approx. 354 mm. Larger n caused greater depth.
10.8.1 (*a*) 6.60 cfs; (*b*) 0.0001902.
10.8.2 (*a*) 0.225 m^3/s; (*b*) 0.001019.
10.8.3 8.11×10^{-4} cfs/ft; 76.6.
10.8.4 0.282 L/s per m; 281.
10.9.1 761 cfs.
10.9.2 35.4 m^3/s.
10.9.3 0.168 m; 0.216 m^3/s per m.
10.10.1 1.204 ft, 6.23 ft/sec, 0.00308.
10.10.2 0.00378; 0.00788; the second, flow is not wide and shallow; 52.0% small.
10.11.1 1.550 ft, 0.00305.
10.11.2 0.437 m, 0.00313.
10.11.3 (*a*) 6.58 ft/mile; (*b*) subcritical.
10.11.4 2.44 m.
10.13.1 0.414 m.
10.13.2 0.769-ft rise.
10.13.3 0.1446-m rise.
10.13.4 0.070-ft drop.
10.15.1 1377 ft downstream.
10.15.2 640 m downstream.
10.15.3 0.00988.
10.15.4 0.01141.
10.17.1 By S_2 or S_3 or C_3 profile.
10.17.2 M_2.
10.17.3 M_2.
10.17.4 A_2.
10.17.5 A_2.
10.17.6 Increase; S_3 after sluice gate.
10.17.7 Sketch $M_1 > y_0 = 0.279$ ft $> y_c = 0.1009$ ft.
10.17.8 Sketch $M_1 > y_0 = 0.0865$ m $> y_c = 0.0311$ m.
10.17.9 $M_1 > y_0 = 4.26$ ft $> y_c = 3.52$ ft; damming action, therefore y_c on dam.
10.18.1 (*a*) 0.535 ft; (*b*) 4.86 ft, 137.8 hp.
10.18.2 (*a*) 0.1564 m; (*b*) 3.70 m, 290 kW.
10.18.3 85.7 cfs, 1.906 ft.
10.20.1 13.10 ft/sec.
10.21.1 16.31 mm.
10.22.1 2.68 ft.
10.22.2 (*a*) 1.518 m or 0.869 m; (*b*) expanding or contracting, respectively.
10.23.1 1.5 m diameter (theoretically 1.409 m).

Chapter 11

11.1.1 1.420 slugs/ft^3.
11.1.2 1.699 g/cm^3 or 1699 kg/m^3.
11.1.3 76.7 pcf; surface tension.
11.1.4 11940 N/m^3; surface tension.
11.1.5 0.386 lb·sec/ft^2.
11.1.6 20.4 N·s/m^2.
11.1.7 0.0000227 ft^2/sec.
11.1.8 72.9 sec.
11.1.9 49.1 sec.
11.1.10 0.0806 lb·sec/ft^2; 0.0647.
11.3.1 0.444 m/s.
11.3.2 14.84 ft/sec.
11.3.3 4.25 m/s.
11.3.4 4.03 ft/sec.
11.3.5 1.156 m/s.
11.3.6 1.531 ft/sec.
11.3.7 0.470 m/s.
11.6.1 0.606, 0.618, 0.981.

1.6.2 (*a*) 0.606; (*b*) 0.969; (*c*) 0.625.
1.6.3 (*a*) 0.609; (*b*) 0.977; (*c*) 0.623.
1.6.4 39.6 ft/sec, 3.46 cfs.
1.6.5 39.3 ft/sec, −18.00 ft.
1.6.6 10.90 ft; separation, $p_o = p_a$, smaller Q.
1.6.7 28.3 ft/sec, −6.68 ft; less V, more p with friction.
1.6.8 13.43 m/s, −6.19 m.
1.6.9 4.39 m.
1.6.10 10.43 L/s.
1.7.1 0.316 cfs.
1.7.2 8.58 L/s.
1.7.3 1.019 m^3/s.
1.7.4 4.84 in.
1.8.1 (*a*) 0.0001548 m; (*b*) 0.014 65 m; (*c*) 1.465 m.
1.9.1 (*a*) 0.000 365 m; (*b*) 0.0383 m; (*c*) 3.83 m.
1.10.1 169.4 lb/sec.
1.10.2 0.1394 lb/sec.
1.10.3 5.34 lb/sec.
1.10.4 31.6 N/s.
1.10.5 5.87 lb/sec.
1.10.6 36.6 N/s.
1.10.7 (*a*) 52.8 psia, 1030 ft/sec; (*b*) 52.8 psia, about 1045 ft/sec.
1.10.8 (*a*) 370 kN/m^2 abs, 313 m/s; (*b*) 370 kN/m^2 abs, about 318 m/s.
1.11.1 23.6 cfs.
1.11.2 0.729 m^3/s.
1.11.3 (*a*) 22.7 cfs; (*b*) 2.00 ft.
1.11.4 (*a*) 0.708 m^3/s; (*b*) 0.733 m.
1.11.5 (*a*) 0.293 cfs; (*b*) 137.5% increase; 1.018 ft.
1.12.1 18.92 cfs/ft.
1.12.2 1.715 m^3/s per m.
1.12.3 2.46 ft.
1.12.4 0.705 m.
1.12.5 247 cfs.
1.12.6 6.66 m^3/s.
1.14.1 1.597 cfs/ft, 0.579.
1.14.2 0.237 m^3/s per m, 0.581.

Chapter 12

12.2.1 210 ft^2.
12.2.2 15.64 m^2.
12.2.3 791 sec.
12.2.4 605 s.
12.2.5 1276 sec.
12.2.6 962 s.
12.2.7 98.4 sec.
12.2.8 97.6 s.
12.3.1 −0.004 52 ft.
12.3.2 −0.450 cfs/sec.
12.4.1 5.96 ft/sec, 3.05 sec.
12.4.2 3.41 m/s, 1.593 s.
12.4.3 3.76 m/s, 1.772 s, 4.11 s.
12.4.4 11.00 ft/sec, 1.407 sec, 2.49 sec.
12.4.5 7.57 ft/sec, 6.58 sec.
12.4.6 3.20 m/s, 5.61 s.
12.5.1 (*a*) 4879 ft/sec; (*b*) 3761 ft/sec; (*c*) 77.1%.
12.5.2 (*a*) 1478 m/s; (*b*) 1126 m/s; (*c*) 76.2%.
12.5.3 3354 ft/sec.
12.5.4 1020 m/s.
12.6.1 (*a*) All 100.9 psi; (*b*) all 70.7 psi.
12.6.2 (*a*) All 800 kPa; (*b*) all 600 kPa.
12.6.3 (*a*) 100.9 psi, 100.9 psi, 74.8 psi; (*b*) 70.7 psi, 70.7 psi, 52.3 psi.
12.6.4 (*a*) 800 kPa, 800 kPa, 485 kPa; (*b*) 600 kPa, 600 kPa, 364 kPa.
12.6.5 (*a*) 80.7 psi, 53.8 psi, 26.9 psi; (*b*) 56.5 psi, 37.7 psi, 18.84 psi.
12.6.6 (*a*) 571 kPa, 381 kPa, 190.5 kPa; (*b*) 429 kPa, 286 kPa, 142.9 kPa.
12.7.1 65.3 ft.
12.7.2 1.837 m.

Chapter 13

13.1.1 2.85×10^6 ft·lb.
13.1.2 4.55 MJ.
13.1.3 174.2°F, 273 psia, 3.15×10^6 ft·lb, $+4.42 \times 10^6$ ft·lb.
13.1.4 80.2°C, 1913 kPa abs, 4.89 MJ, +6.85 MJ.
13.1.5 37300 J/kg, 130200 J/kg checks closely.
13.3.1 1116, 1097, 1077, 1037, and 995 ft/sec.
13.3.2 340, 332, and 299 m/s.
13.5.1 17.89 psi, 89.0°F; 14.98 psi, 71.2°F; 5.59 psi, −17.8°F.
13.5.2 127.9 kPa abs, 34.9°C; 101.5 kPa abs, 21.9°C; 35.7 kPa abs, −30.0°C.
13.5.3 284 psia, 121°F, 1160 ft/sec.
13.5.4 1979 kPa abs, 51.2°C, 355 m/s.
13.6.1 1800 ft/sec, −170°F.
13.6.2 551 m/s, −111.1°C.
13.6.3 682 ft/sec.
13.6.4 208 m/s.
13.8.1 Q.E.D.
13.8.2 Q.E.D.
13.8.3 825 ft/sec, 22.0 psia, 13.68°F.
13.8.4 251 m/s, 153.8 kPa abs, −11.60°C.
13.8.5 644 ft/sec, 14.2 psia, 59.6°F.
13.8.6 201 m/s, 98 kPa abs, 16.51°C.
13.9.1 1.453.
13.9.2 1.459.
13.9.3 2.56 in, 0.224 slug/sec, (2) 1039 ft/sec, −10.00°F, (3) 1681 ft/sec, −155.3°F.
13.9.4 8.57 in.
13.9.5 (*a*) and (*b*) 0.0790 slug/sec.
13.9.6 (*a*) and (*b*) 1.101 kg/s.
13.10.1 (*a*) impossible situation; (*b*) 3.12.
13.10.2 (*a*) impossible situation; (*b*) 4.15.

13.11.1 1299 ft/sec, 3.98%.
13.11.2 1722 ft/sec.
13.11.3 812 m/s.
13.13.1 (*a*) 37,800 ft·lb/slug; (*b*) 48.6 Btu/slug; (*c*) added; (*d*) 15,450 Btu/hr.
13.13.2 (*a*) 3220 N·m/kg; (*b*) 3220 J/kg; (*c*) added; (*d*) 13.95×10^6 J/h.

Chapter 14

14.1.1 (*a*), (*b*), (*d*), (*e*).
14.1.2 (*a*), (*c*), (*d*), (*e*), (*g*).
14.2.1 (*a*), (*b*), (*c*), (*f*).
14.2.2 (*b*), (*c*), (*d*), (*g*).
14.3.1 (*a*) 0; (*b*) 0; (*c*) 0; (*d*) 1; (*e*) −5; (*f*) 0.
14.3.2 (*a*) −3; (*b*) 0; (*c*) 0; (*d*) 0; (*e*) $-2y - 4x$; (*f*) $4y + 2x$; (*g*) 0.
14.4.1 Yes. $-4x^2 + 2y$.
14.4.2 (*a*) $2y$; (*b*) $2y - 3x$; (*d*) $-1.5x^2 + y^2$; (*e*) $1.5x^2 + y^2$.
14.4.3 (*a*) $1.5y^2$; (*c*) $3xy$; (*d*) $4y + 6x + 2xy$; (*e*) $x^2y + 2xy^2$; (*g*) $x^3/3 - 2x^2y - xy^2 + 2y^3/3$.
14.5.1 (*a*) $12.5\theta/\pi + 5r\sin\theta$; (*b*) $(12.5/\pi)\arctan(y/x) + 5y$.
14.5.2 (*a*) $x = -0.796$ m; $y = 0$ ($r = 0.796$, $\theta = 180°$); (*b*) 5.51 m/s.
14.5.3 0.342 m.
14.5.4 2.46 ft.
14.6.1 Yes; no, because continuity is not satisfied.
14.6.2 Yes; $7x + 10y$; yes.
14.6.3 (*a*) Yes, $2y + 1.5y^2 - 1.5x^2$, yes; (*b*) no, yes.
14.6.4 (*a*) Yes, $3(y^2 - x^2)$, yes; (*b*) no, yes.
14.7.1 (*a*) $\psi = 2y$, $\phi = -2x$; (*b*) $\psi = 2y - 3x$, $\phi = -3y - 2x$.
14.7.2 (*c*) $\psi = 3xy$, $\phi = -1.5x^2 + 1.5y^2$; (*d*) $\psi = 4y + 6x + 2xy$, $\phi = 6y - 4x - x^2 + y^2$; (*g*) $\psi = x^3/3 - 2x^2y - xy^2 + 2y^3/3$, $\phi = 2x^3/3 + x^2y - 2xy^2 - y^3/3$.
14.8.1 1.735 ft^3/day per ft, 0.588 ft/day.
14.8.2 0.1643 m^3/d per m, 0.1880 m/d.
14.8.3 2.72 ft/day.
14.8.4 0.945 m/d.

Chapter 15

15.4.1 268 rpm, 25.1 m^3/s, 14000 kW.
15.4.2 267 hp; 156.3 ft, 25 cfs, 521 hp.
15.4.3 22.1 kW; 8.64 m, 360 L/s, 38.1 kW.
15.4.4 89.9%.
15.6.1 44.8 ft, 6000 gpm.
15.6.2 76.9 ft, 8640 gpm.
15.6.3 Shutoff 71.1 ft, 0 cfs; BEP 28.4 ft, 162.6 cfs; 75 cfs at 23 ft becomes 100 cfs at 40.9 ft, etc.
15.7.1 $h = \Delta z + 0.1061Q^2$; $h \approx 60$ ft with $Q \approx 10{,}500$ gpm, $h \approx 65$ ft with $Q \approx 9200$ gpm, $h \approx 76$ ft with $Q \approx 4500$ gpm. Approx. pump efficiencies: 83%, 79%, and 52%, respectively.
15.7.2 $h = \Delta z + 0.403Q^2$; $h \approx 73$ ft with $Q \approx 6500$ gpm, $h \approx 75$ ft with $Q \approx 5300$ gpm, $h \approx 78$ ft with $Q \approx 2500$ gpm. Approx. pump efficiencies: 63%, 56%, and 33%, respectively.
15.8.1 Raise Eq. (15.4) to the $\frac{1}{2}$ power and raise Eq. (15.5) to the $\frac{3}{4}$ power. Divide the first equation by the second to eliminate D, and find $N_s = nQ^{1/2}/h^{3/4}$.
15.8.2 $N_s = 13{,}163$: axial-flow pump.
15.8.3 1395, 1622, and 2140, respectively; all are radial-flow pumps.
15.8.4 $N_s = 1670$, radial-flow pump.
15.8.5 $(N_s)_{SI} = 0.318$; radial-flow pump.
15.9.1 Approx. 38 in.
15.9.2 Approx. 10 in.
15.9.3 Approx. 24.8 in = 630 mm.
15.10.1 Approx. 6.2 ft above the reservoir water surface. Approximate answer using Fig. 15.13: 10 ft above water surface.
15.10.2 Approx. 3.8 ft or more below the elevation of the reservoir water surface.
15.10.3 Place the pump no more than 5.01 m above the water surface.
15.12.1 $N_s = 2670$, no cavitation if pump is less than about 18 ft higher than the water surface.
15.12.2 $N_s = 1357$, no cavitation if pump is at least about 12 ft lower than the water surface.
15.12.3 10.9 m, $(N_s)_{SI} = 0.534$, no more than about 3.9 m above the water surface; hence cavitation will occur (6 m > 3.9 m).
15.13.1 Approx. 112 gpm, 179 gpm, 124 gpm.
15.13.2 Approx. 170 gpm; approx. 1650 rpm.

Chapter 16

16.3.1 278 hp. Horsepower transmitted to the runner, because $v_2 = v_1$.
16.3.2 (*a*) 1102 lb; (*b*) 5510 ft·lb; (*c*) 251.7 hp; (*d*) 10.69%.
16.3.3 (*a*) 72400 N; (*b*)72400 N·m; (*c*) 2840 kW.
16.4.1 (*a*) 1942 ft; (*b*) 88%; (*c*) 84%.
16.4.2 2.67 ft/sec, 385 ft/sec, 33.5 cfs, 166.5 ft/sec, 0.432, 2413 ft, 27,900 lb, 123,300 ft·lb, 8450 hp, 9170 hp, 92.1%.
16.4.3 90.6%.
16.5.1 About 1.0-in-diameter jet with about 8.95 hp.
16.5.2 About 167 hp with jet diameter of slightly less than 5 in.
16.7.1 8750 N·m.
16.7.2 30 poles.
16.8.1 3.23 ft of additional head.
16.8.2 97.2 ft.

6.8.3 6.04 m.
6.10.1 (*a*) 88.3%; (*b*) 189.7 rpm, 12.65 m^3/s, 5260 kW; (*c*) 38 poles.
6.10.2 (*a*) 20 poles; (*b*) 2093 hp.
6.10.3 22 poles: 273 rpm, 52.9 ft, 282 cfs, 1573 hp.
24 poles: 250 rpm, 44.4 ft, 258 cfs, 1209 hp.
6.11.1 4.01, approx. 8.8 ft.
6.11.2 25.1, approx. 6.4 ft, Francis turbine.
6.11.3 146.0, approx. 2.71 ft, propeller-type turbine.
6.12.1 Approx. 157 ft; fairly well.
6.12.2 (*a*) Approx. 42.5 ft; (*b*) approx. 30.2 ft.
16.12.3 Approx. 77 ft.
16.13.1 (*a*) It is possible, $N_s = 6.25 < 7.0$. (*b*) Range of N_s: from 6.25 (96 poles) to 6.99 (86 poles). Range of D: from approx. 6.8 ft to approx. 6.1 ft.
16.13.2 (*a*) It is possible, $N_s = 50.1 > 20$. (*b*) Range of N_s: from 20.1 (30 poles) to 50.1 (12 poles). Range of D: from approx. 3.7 ft to approx. 1.54 ft.
16.13.3 $N_s = 27.2$, D = approx. 13.1 ft.
16.13.4 21.8 ft, 11.06 ft.

Index

Page numbers in the index that are followed by an *n* refer to footnotes on those pages.

A
Abbreviations, xxv, 9
Absolute path, 203, 212–213
Absolute pressure, 53–54, 60
Absolute pressure head, 60
Absolute roughness, 280
  of pipes, 286–287
Absolute velocity, 200–201
Absolute viscosity, 29–35, 730
Absolute zero temperature, 22*n*
Acceleration, 117–123
  convective, 118
  local, 121
  normal, 119–120, 172
  tangential, 119–120
Accelerative head, 550–553
Acceleration scale ratio, 233
Acoustic flow meter, 502
Acoustic (sonic) velocity, 139, 585–592, 595–600
Actuating or actuated disk, 222–225
Adhesion, 37
Adiabatic exponent, 24
Adiabatic flow, 588, 614–618
Adiabatic process, 24, 581
Advection, 343
Adverse slope, 454
Affinity laws, 654
Aging of pipes, 320
Air, physical properties of, 733–734
Air entrainment, 410
Airfoil, 385–386, 389–400
  Clark Y, 396–398
Alternate depths, 431–437
Anemometer, 500–501
  hot-film, 501
  hot-wire, 501
  laser-Doppler, 502
  rotating, 500–501
Aneroid barometer, 55–56
Angle of attack, 391–398
Angle of repose, 423
Angular momentum, 176, 216
Answers to exercises, 769–776
Approach, velocity of, 529–530
Approach to steady flow, 554–557
Appurtenances, 481
Archimedes, 2
Areas, properties of, 738
Areas of circles, 737
Aspect ratio, 394
Atmosphere, standard, 27–29, 47–49, 734
Atmospheric pressure at sea level, 54, 56
Automobiles, drag on, 378–379
Avagadro's law, 23
Axial-flow, 214, 648
Axial-flow pumps, 214, 647–678
Axial-flow turbines, 697, 700–718, 721–722
Axisymmetric flow, 110–111, 122

B
Back pressure, 596
Backwater curve, 454, 456
Barker's mill, 221
Barometer, 55–56
Barometric pressure, 54–56
Base of nozzle, 505, 691
Bed slope, 409, 454
Bell-mouthed entrance, 303, 509
Bends:
  flow in, 312–314, 471–473
  force on, 194–195, 472
  loss in, 312–314
Bernoulli, Daniel, 133
Bernoulli constant, 133
Bernoulli equation, 133, 145, 385, 515, 626
  assumptions involved, 133
Best efficiency point (BEP), of pump, 657
BG (British Gravitational) units, 7, 8
Blades, force on, 198–206
Blasius, H., 281, 281*n*, 361–362
Blasius friction factor equation, 281
Blasius velocity distribution in laminar boundary layer, 361–362
Boiling, 40
  pressure, 40
Books on fluid mechanics, 764–768
Borda tube (mouthpiece), 509–510
Boundaries, smooth and rough, 272–275
Boundary layer, 3, 114, 265–267, 271–272, 357–372
  backflow in, 373
  drag of, 357–372
  on flat plate, 358–372
  laminar, 265–267, 271–272, 360–364
  in pipe, 265–267, 271–272
  and separation, 114–115, 372–378
  turbulent, 272, 360, 365–368
  velocity distribution in, 266, 361, 365–366
  viscous sublayer, 271–275, 365
Bourdon pressure gage, 57
Brake power, 652, 706

Branching pipes, 326–333
nonrigorous solutions, 331–333
Brater, Ernest F., 415*n*
British Gravitational (BG) units, 7
Broad-crested weir, 443, 534
Bubbler, 540
Bucket friction loss coefficient, 692
Buckingham, E., 247
Buckingham pi theorem, 247–249, 653
Bulk modulus of elasticity, 17–18, 25
of water, 17
Buoyancy, 81–86
center of, 83
Buoyant force, 82–83

C

Calculators, scientific, 746–747, 754–755
Capillarity, 37–39
Capillary rise, 37–39
Cauchy equations, 188
Cauchy number, 239
Cavitation, 40, 154–157, 517
on overflow spillways, 538
in pumps, 666–670
in turbines, 713–717
Cavitation number, 240
Cavitation parameter, 667–668, 714–715
critical, 667–670, 715
Celerity of wave, 468–471, 558–566, 585–588
Center of buoyancy, 83
Center of pressure, 68–74
Centipoise, 33
Centistoke, 33
Centrifugal pumps, 213–214, 647–678
cavitation in, 666–670
characteristics of, 655–657
efficiency of, 652, 654, 657, 663
specific speed of, 662–664, 669
types of, 647–650
Centripetal force, 172
Centroids, 738–739
CFD (computational fluid dynamics), 189, 748
Characteristic curves for pumps, 655–657, 675–677
Check (foot) valve, 650
de Chézy, Antoine, 411
Chézy formula, 411
Cipolletti weir, 531
Circular cylinder, flow around, 382–390
Circular motion, 172
Circulation, 385–390, 627–628
Clark Y airfoil, 396–398
Coefficient:
of contraction, 507–510
of discharge, 508–510, 515–517
of drag, 375–384, 394–401
for flow nozzle, 519–521
of lift, 394–398
for orifice meter, 522–523
for sluice gate, 539
of velocity, 508–510
for venturi meter, 515–518
of viscosity, 31
Cohesion, 37
Colebrook, C. F., 281*n*, 282
Complex variable potential, 637*n*
Compound gage, 57
Compound pipes:
in network, 339–343
in parallel, 336–339
in series, 333–335
Compressibility:
effects on drag, 399–400
of liquids, 17–19
of perfect gases, 25–26
Compressible flow, 147–149, 580–619
converging, 596–599
converging and diverging, 600–603
effect of area variation, 594–595
effect on drag, 399–400
expansion factors, 525–526
measurement of, 524–527
normal shock wave, 601–606
oblique shock wave, 607–609
stagnation pressure in, 589–592, 596–598
Compressible fluids:
equations for, 22–24, 580–585
flow of, 147–149, 580–619
flow measurement of, 524–527
Computational fluid dynamics, 2, 189, 748
Computational fluid mechanics, 2, 189, 748
Computer applications, 745–753
Computer models, 749–751
Conformal transformation, 390
Conical diffuser, 164, 309–311
Conjugate depth, 462–463
Conjugate functions, 637
Conservation of mass, 108–110
Continuity equation, 108–110, 584, 622–624
Contracted weir, 530–531
Contraction:
gradual, 306–307
of jet, 506–507
in open channels, 442–447
sudden, 305–306
suppressed, 528, 538–539
Contraction coefficient, 507–510
Control section, 441
Control surface, 106–108, 186–187
Control volume, 46, 106–108, 186–188
Convective acceleration, 118
Converging flow, 114, 302
Conversion factors, 10–11
Conveyance, 413
Critical cavitation parameter, 667–670
Critical contracted width, 443
Critical depth in open channels, 432–447
occurrence of, 441–442
Critical flow, 432, 436–437
Critical hump height, 443–447
Critical property ratios, 591, 597–599
Critical Reynolds number:
for flow around sphere, 375–377
for flow in boundary layer, 363–370
in open channels, 408
in pipes, 256–257
Critical slope, 436, 454
Critical submergence, of weir, 535
Critical velocity, 432, 439–440
Croley, T. E., 746*n*
Cross, Hardy, 340*n*
Cross section, most efficient, 422–425
Cross waves, 470, 481
Cubic equation, 71, 153, 324, 435, 445, 742
Culverts:
entrance control, 476–478
flow in, 476–480
free entrance, 476, 478–479
outlet control, 476–477
submerged entrance, 476–480
Current meter, 500–501
Curved path, flow in, 172
Curved surfaces, force components on, 77–79
Cylinder:
drag of, 382–383
flow field around, 382–390
lift force on rotating, 387–390
Cylindrical vortex, 173–177

D

D'Alembert's paradox, 356*n*, 394
Dalton's law of partial pressures, 23
Damming action, 443–447
Darcy, Henry, 640
Darcy friction factor, 261
Darcy's law, 640
Darcy-Weisbach equation, 261, 316
Daugherty, Robert L., xv
da Vinci, Leonardo, 2, 108
Deep-well pump, 649–650
Delivery curves, 480
Density, 14
measurement of, 491
Depression in open channel, 447
Deriaz turbine, 701
Differential manometer, 61–64, 517
Diffuser, conical, 164, 309–311
Diffuser (manifold), 344
Diffusion, 343
Dimensional analysis, 245–251
of pipe friction, 260
pi theorem, 247–249, 653
Rayleigh method, 246–247
of weir, 529
Dimensionless relations, 234–240
Dimensions, 6–11
commonly used, 8
five basic, 7
Direction-finding tube, 496–498
Discharge:
mass rate of, 103–105
measurement of, 503–541
(volume) rate of, 103–105
under varying head, 546–548
weight rate of, 103–105
Discharge coefficient, 508–510, 515–517
Discharge curve, 433–435
Discharge problem, 481
Dispersion, 344
Disturbance line, 470
Disturbed flow, 100, 114–115, 301, 307–308
Diverging flow, 114–115, 301, 307
Doublet, 634
Downwash velocity, 393
Draft head, 704
Draft tubes, 702–705
Drag, 356, 372–384, 392–400
on automobiles, 378–379
on bodies in compressible flow, 399–400
on cylinder, 382–384
form, 357
friction, of boundary layer, 357–372
induced, 392–394
pressure, 357, 372–374
profile, 394
projectile, 399–400
on sphere, 375–381, 493
surface, 357
on three-dimensional bodies, 374–384
on two-dimensional bodies, 382–384
Drag coefficient, 375–384, 393–401
for automobiles, 378–379
for bodies of revolution, 376
for smooth plate, 370
for two-dimensional bodies, 382–384
Drag diagrams, 395–396
Drawdown curve, 442, 454–457
Drops, of liquid, 39
Dyes, 344
Dynamic pressure, 139, 240, 591
Dynamic similarity, 234–240
Dynamic viscosity, 31
Dyne, 9

E

Earth's rotation, 8
Eddies, 100, 114–115, 268, 301, 373
Eddy currents, 301
Eddy diffusion, 344
Eddy formation, 100, 114–115, 268, 301, 373
Eddy viscosity, 269
Eddy rotation, 100
Effective head, 691, 704
Efficiency, 151
hydraulic, 692
Moody step-up formula, 654, 707
of propellers, 224–225
of pumps, 652, 654, 657, 663
of turbines, 691–694, 706–708
of windmills, 225
Efficiency values:
for pumps, 663
for turbines, 706–707
Efficient cross section, 422–425
Elasticity, modulus of, 17–18, 25, 587

Elbow, loss in, 312–314
  vaned, 313
Elbow meter, 313, 540
Electric analogy, 339
Electrical gages, 540
Elevation head, 150
Empirical equations, 298–300, 740
  for pipe flow, 298–300
Energy:
  internal, 129, 142–144
  kinetic, 127–128
  mechanical, 141–145, 219
  potential, 128–129
  specific, 431–435
  thermal, 141–145
Energy equation, 131–136, 140–149, 584
  for compressible fluid, 147–149, 584
  for gradually varied flow, 449–452
  for unsteady flow, 551
Energy gradient, 261, 299, 409, 453
Energy head, 158
Energy line, 158–169, 409
Energy loss, 144–150
  in flow in conduit, 136
  in fluid flow, 136
  in hydraulic jump, 462–464
  in nozzles and orifices, 510–513
  in pipe flow, 258–265, 301–314
English Engineering units, 7, 9
Enlargement:
  gradual, 309–311
  sudden, 307–311
Enthalpy, 147–148, 581, 584
Entrance control, 476–478
Entrance length, 266–267, 272
Entrance loss, 302–303
Entropy, 581
Equation of continuity, 108–110
Equations:
  cubic, 71, 153, 324, 435, 445, 742
  empirical, 298–300, 740
  explicit, 281, 286, 294, 315–316, 740–741
  in fluid mechanics, 740–744
  implicit, 281, 286, 294, 315–316, 741
  linear, 636, 741
  nonlinear, 743
  simultaneous, 285, 343, 744–747, 753–754, 760
  of state for gases, 22, 580
  summary of types, 740–744
Equation solver, 71, 153, 296–298, 315–319, 325, 329–330, 335, 339, 343, 415–416, 445, 475, 480, 573, 611, 613, 746–748, 754–763
Equipotential lines, 111–114, 635–639
Equivalent diameter, 263
Equivalent length, of pipe to minor loss, 301
Equivalent velocity head, 334
Established flow, 266
Euler, Leonhard, 132, 239
Eulerian approach, 107
Euler number, 239–240
Euler's equation(s) of motion, 132, 188, 584, 593
Excel, xvii, 3, 71, 297–298, 316, 329, 335, 339, 343, 754–759
Exercises, answers to, 769–776
Expansion:
  gradual, 309–311
  in open channels, 447
  sudden, 307–311
Expansion factor, 525–526
Explicit equation, 281, 286, 294, 315–316, 740–741
External flows, 356
Eye, of pump, 677, 719

F

Fanning equation, 261*n*
Fanno flow, 615
Fanno line, 615–617
Filament line, 102–103
Finnemore, E. John, v, xv
Fittings, losses in, 312–314
Flat plate:
  boundary layer on, 358–372
  Reynolds number for, 361–372
Flettner, A., 390*n*
Floating bodies, 83–85
Float measurements, 501
Flow:
  adiabatic, 588, 614–618
  axial, 214, 647–678
  axially-symmetric, 110–111, 122
  in bends, 312–314, 471–473
  classification of, 97–99
  of compressible fluids, 147–149, 580–619
  converging, 114, 302
  critical, 432, 436–437
  in culverts, 476–480
  in curved path, 172
  around cylinder, 382–390
  disturbed, 100, 114–115, 301, 307–308
  diverging, 114–115, 301, 307
  established, 266
  external, 356
  free surface, 114, 407
  fully-rough-pipe, 274–275, 281, 286
  gradually varied, 448–459
  gravity, 407
  hypersonic, 239
  ideal, 131–134, 622–642
  irrotational, 625–627, 636
  isentropic, 593
  isothermal, 609–613
  kinematics, 97–123
  laminar (*see* Laminar flow)
  mean steady, 101–102
  measurement of, 503–541
  mixed, 214, 647–648
  nonuniform, 101, 448–459
  one-dimensional, 110
  in open channels (*see* Open channel flow)
  through porous media, 639–642
  potential, 635–639
  pulsating, 410, 481
  radial, 212–218, 647–678
  rapidly varied, 448–449
  rectilinear, 631
  from reservoir, 317–319
  reverse, 114
  rotational, 625–627
  rough-pipe, 274–275, 281, 286
  secondary, 312–313, 472–473
  sheet, 429–430
  single-pipe (*see* Single-pipe flow problems)
  smooth-pipe, 274–275, 280–281, 286
  around sphere, 375–381
  over spillways, 536–538
  spiral, 312, 472–473
  around square block, 189
  steady, 101–102, 117–120, 131–137, 140–149
  subsonic, 239, 399, 585, 594
  supersonic, 239, 399, 585, 594
  three-dimensional, 110–111, 374–381
  through porous media, 639–642
  through rotating channel, 175, 219
  transitionally rough, 274
  transonic, 399
  turbulent, 99–101, 255–256, 268–282
  two-dimensional, 110–111, 382–384, 628, 630, 638
  types of, 97–99
  unestablished, 265–266
  uniform, 101, 407–417
  unsteady, 101–102, 121–123, 546–574
  varied, 448–459
  viscous, 98, 144
  visualization of, 502
  "wide and shallow," 421–422
Flow coefficient, 519–523
Flow fields, 119, 627, 630–634
  superposition of, 388, 632
Flow measurement of compressible fluids, 524–527
Flow net, 111–116, 637–639
  mathematical basis, 112*n*, 636–639
Flow nozzle, 519–521, 526
Flow problems, method of solution, 165–168
Flow rate, 103–105
  mass, 103–105
  relative, 202
  volume, 103–105
  weight, 103–105
Flow visualization, 502
Flow work, 141
Fluid, 13
  ideal, 29, 97, 111, 622–642
  inviscid, 29
  Newtonian, 32
  non-Newtonian, 31–32, 32*n*
  real, 29, 97
Fluid dynamics, 2
Fluid friction, 143
  in conduits, 258–261
Fluid mechanics, 2
  books on, 764–768
  development of, 2–3
  journals on, 764
Fluid power, 652
Fluid properties, 732–736
  measurement of, 491–494
Fluid statics, 2, 45–91
Fluid system, 106–108, 186
Flush entrance, 303, 509
Foot (check) valve, 650
Force:
  on accelerating static fluid, 88–91
  on bend, 194–195, 472
  on blades, 198–206
  buoyant, 82–83
  centripetal, 172
  on curved surface, 77–79
  drag (*see* Drag)
  on flat plate, 358–372
  lift (*see* Lift)
  on nozzle, 196–197
  on open flow structures, 190–192
  on plane area, 66–74
  on pressure conduits, 193–197
  on reducer, 193–196
  on vanes, 198–206
Force main, 344
Force scale ratio, 234
Forced vortex, 173–176
Form drag, 357
Fourier, Baron Joseph, 246
Frame of reference, 117
Francis, James B., 697
Francis formula, 530
Francis turbine, 697–701, 706–708, 712–718, 721
Franzini, Joseph B., v, xv
Free discharge from reservoir, 317
Free entrance, 476, 478–479
Free overfall, 442, 457, 468, 535
Free surface flow, 114, 407, 749
Free vortex, 176–179
French, Richard H., 448*n*
Friction coefficient. *See* Friction factor
Friction drag, 357–372
Friction factor, 261, 282–284, 410, 413
  chart for, 282–284
  Darcy, 261
Friction head loss, 258–265
Friction velocity, 273
Froude, William, 237
Froude number, 237–238, 244, 437–438, 453–454, 464
Fully developed flow, 266, 272
Fully-rough-pipe flow, 274–275, 281, 286

G

Gage:
  Bourdon, 57
  compound, 57
  electrical, 540
  hook, 528, 540
  point, 528, 540
  pressure, 57
  staff, 540
  vacuum, 57
Gage pressure head, 59
Gage pressures, 53–54, 59
Gas, 13
  perfect, 22–24
  property relations for, 22–24
Gas constant, 22
  universal, 23
Gas dynamics, 580
Gases:
  adiabatic flow with friction, 588, 614–618
  equation of state for, 22, 580

Gases—*Cont.*
isentropic flow of, 597
isothermal flow of, 609–613
process equations for, 23–24
properties of, 13–17, 22–31, 40–41, 733–736
property relations, 22, 580
Gas laws, 22–24, 580–582
Generator poles, 685, 709–710
Geometric similarity, 232–233
Governor, 686, 697–698
Gradient of velocity potential, 638
Gradual contraction, 306–307
Gradual expansion (enlargement), 309–311
Gradually varied flow, 448–459
standard step method for, 450–452
Gravity flow, 407
Gravity waves, 468–471
Gross head of power plant, 691
Guide vanes, 212, 697–698

H

Haaland, S. E., 282*n*
Hagen, G., 265
Hagen-Poiseuille law, 265
Hardy Cross method, 340–342
Hazen-Williams formula, 299, 315
Head:
accelerative, 550–553
added (developed) by pump, 150, 651–652
definition of, 51–52, 150
effective, 691, 704
elevation, 150
energy, 158
gross, 691
hydraulic, 640
imparted to (removed by) turbine, 150, 219
on impulse turbine, 691–694
limiting, for pump, 669
net, 691, 704
normal, of turbine, 706*n*
piezometric, 150, 158
potential, 150
pressure, 51–52, 150
on reaction turbine, 702–705
shutoff, 657
static (pressure), 150, 158, 691
total, 150, 158
utilized (by turbine), 219
varying, 546–548
velocity, 150
Head loss, 137, 144–150, 258–265
in bends and elbows, 312–314
at changes in pipe section, 302–311
due to friction, 136–137, 258–265
at entrance, 302–303
in hydraulic jump, 462–464
nonrigorous equations for, 300–301
in nozzles, 307, 319, 510–513
in orifices, 510–513
in pipe fittings, 312–314
at submerged discharge, 160–161, 303–305
total, in pipes, 137, 144–146, 150–151, 161–163, 258–265, 285, 301–314, 321
Headwater, 477–479, 478*n*
Herschel, Clemens, 515
Homologous machines, 652, 708–710
Hook gage, 528, 540
Horsepower, 151
Hot-film anemometer, 501
Hot-wire anemometer, 501
Hubbert, M. King, 640*n*
Hump in open channel, 442–447
critical height, 443–447
effect of, 442–447
Hydraulically smooth, 274–275, 280
Hydraulic conductivity, 640
of common geologic deposits, 640
Hydraulic cross section, best, 422–425
Hydraulic depth, 438
Hydraulic diameter, 258
Hydraulic drop, 441
Hydraulic efficiency, 692
Hydraulic grade line, 158–168
Hydraulic gradient, 640
Hydraulic head, 640
Hydraulic jump, 460–468
depth relations, 461–463
energy loss, 462–464
length, 464
location of, 465–468
types, 464–465
Hydraulic mean depth, 438
Hydraulic models, 232–245
Hydraulic radius, 258, 263, 427
Hydraulics, 2
Hydrodynamics, 2, 622–642
Hydroelectric power, 685
Hydrogen bubbles, 502
Hydrometer, 83–84, 491
Hydrostatics, 45–95
Hypersonic flow, 239

I

ICAO standard atmosphere, 27–29, 734
Ideal fluid, 29, 97, 111, 622–642
Ideal velocity, 508, 511, 515, 539
Impeller, 212–213, 647–648
trimming of, 654, 672
Imperial gallon, 10
Implicit equation, 281, 286, 294, 315–316, 741
Implosion, 155
Impulse-momentum principle, 186. *See also* Momentum principle
Impulse turbines, 686–696, 706–707, 711–714, 717–718, 720
cavitation in, 713–714
efficiency of, 692, 706–707
selection of, 717–718
specific speed of, 711–712
Incompressible fluids, 16–17, 255
Induced drag, 392–394
Inertia, moment of, 68–74, 738
Inferential meter, 541
Interferometer, 502
Internal energy, 129, 142–144
International system (SI) of units, 7

Inverted siphon, 343
Inviscid, 29
Inviscid core, 266, 272
Ionosphere, 28
Irrotational flow, 625–627, 636
Irrotational vortex, 176–179
ISA flow nozzle, 519–521
Isentropic compressibility, 587
Isentropic flow, 593
Isentropic process, 24, 581
ISO standard atmosphere, 27
Isothermal flow, 609–613
Isothermal process, 24, 581

J

Jet:
action of, 689–691
definition of, 505–506
force of, 201–206
free, 506
power of, 151, 167, 205
pressure in, 507
reaction of, 206–210
submerged, 160–161, 506, 511–512
trajectory of, 169–170
velocity distribution in, 507
Jet contraction, 506–507
Jet deflector, 686
Jet engine, 211
Jet propulsion, 210–211
Jet trajectory, 169–170
Joukowski, N. E., 388, 391
Joukowski transformation, 391
Journal bearing, 32
Journals on fluid mechanics, 764
Jump, hydraulic. *See* Hydraulic jump

K

Kaplan turbine, 698, 700, 706, 721–722
Kármán vortex street, 382–384
Kiel probe, 498
Kinematic eddy viscosity, 269
Kinematics, 2, 97–123
of flow, 97–123
Kinematic similarity, 233
Kinematic viscosity, 33, 731
Kinetic energy, 127–128
correction factor, 128, 130, 267, 278, 431, 449–450
King, Horace W., 415*n*
Kutta, W. M., 388
Kutta-Joukowski theorem, 388
Kutter formula, 411

L

Lagrangian approach, 107
Laminar boundary layer, 271–272, 360–364
Laminar flow, 98–99, 102, 255–256
in boundary layer, 272, 360–364
in circular pipes, 264–267
entrance condition in, 265–267
fully developed, 266
in open channels, 429–430
around spheres, 375–377
Laminar flow wing, 371
Langhaar, H. L., 266*n*
de Laplace, Pierre S., 636
Laplace equation, 636, 638
Lapse rate, 28, 47
Laser-Doppler anemometry (LDA), 502
Laser Doppler velocimetry (LDV), 502
Law of the wall, 274
Length scale ratio, 233
Lift, 356, 385–400
of airfoils, 390–400
of rotating cylinders, 387–390
Lift coefficient, 394–398
Lift diagrams, 395–396
Line integral, 386, 627
Liquid, 13
Liquid properties, 13–21, 29–39, 732, 735
Local acceleration, 121
Local friction coefficient, 367
Local Reynolds number, 361

M

Mach, Ernst, 238
Mach angle, 608–609
Mach number, 238–239, 400, 585, 589–597, 604–613
Magnetic flowmeter, 502
Magnus, Heinrich G., 388
Magnus effect, 388
Manifold (diffuser), 344
Manning, Robert, 411
Manning formula, 299, 411–412
Manning roughness coefficient, 411–414
variation, 413–414
Manometer, 59–64, 517
differential, 61–64, 517
negative pressure, 61
simple, 59–61
Mass, units of, 7
Mass balance, 108, 110
Mass density. *See* Density
Mass flow rate, 103–105, 109, 584
Mathcad, xvii, 3, 71, 297–298, 316–319, 329, 335, 339, 343, 754, 760–763
Mathematics software, 748–751
Mean steady flow, 101–102
Mean velocity, 105
Measurement:
of compressible flow, 524–527
of flow (discharge), 503–541
of fluid properties, 491–494
of liquid surface elevation, 540
of pressure, 55–64, 495–496
of river discharge, 503–504
of static pressure, 495–496
of velocity, 496–502
of viscosity, 491–494
Mechanical energy, 141–145, 219
Mechanical work, 219
Meniscus, 37
Mesosphere, 28
Meter:
anemometer, 500–501
barometer, 55–56
current, 500, 503
elbow, 540
hydrometer, 83–84, 491
inferential, 541
orifice, 522–526
VDI, 522–523
pitometer, 498
rotameter, 541

Meter—*Cont.*
turbine, 541
venturi, 515–518, 525–526
Method of characteristics, 568
Metrification, 7
Mild slope, 437, 441, 454
Minor losses, 301–320
equivalent length for, 301
Mixed flow, 214, 647, 656–659, 662–663
turbines, 698
Mixing length, 269–271, 276
Model ratio, 233*n*
Model scale, 233*n*
Models, 232–245, 654
Model studies, 242–245
Modulus of elasticity, 17–18, 25, 587
joint (combined), 558, 587
Molar mass, 23, 736
Molecular weight. *See* Molar mass
Moment of momentum, 216
Moments of inertia, 68–74, 738
Momentum, 185–188, 461, 584
correction factor for, 189–190, 278
rate of change of, 186–187, 223
Momentum equation, 584
Momentum principle, 185–225, 461, 584, 689
Moody, Lewis F., 278*n*, 284*n*, 654, 707*n*, 714
Moody diagram, 283–284
Moody step-up formula, 654, 707
Multi-stage pumps, 649–650

N

Nappe of weir, 528
Navier, C. L. M. H., 188
Navier-Stokes equations, 188–189
Needle nozzle, 695
Net head, 691, 704
Net positive suction head (NPSH), 667
Network of pipes, 339–343
Newton, Isaac, 2
Newton (unit), 7
Newtonian fluid, 31–32
Newton's equation of viscosity, 31–32
Newton's Law of Gravitation, 8
Newton's second law, 2, 7, 131, 185–187, 223, 245
Nikuradse, J., 280
Noncircular conduit, 263
Non-Newtonian fluid, 31–32, 32*n*
Nonrectangular channels, 438–440
Nonuniform flow, 101, 448–459
Normal acceleration, 119–120, 172
Normal capacity of pump, 656–657
Normal depth, 408
Normal load, of turbine, 706, 706*n*
Normal shock wave, 601–606
No-slip condition, 30, 114, 265, 271
Nozzles, 196, 505–506, 519–521, 526, 695–696
base of, 505, 691
converging-diverging, 600–603
flow, 519–521, 526
forces on, 196–197
loss coefficient, 307
losses in, 510–513, 692
needle, 695
with pipe, 317
types of, 506
NPSH (net positive suction head), 667

O

Oblique shock wave, 607–609
Ogee spillway, 536–538
One-dimensional flow, 110
One-seventh power law, 281
Open channel flow, 407–481
alternate depths of, 431–437
backwater curve, 456
Chézy formula for, 411
critical depth in, 432–447
critical velocity in, 432, 439–440
drawdown curve, 442, 454–457
hydraulic jump (*see* Hydraulic jump)
Kutter formula for, 411
laminar, 429–430
Manning formula for, 411–412
normal depth of, 408
overbank, 417
secondary, 472–473
spiral, 472–473
standing wave in, 468–471, 481
subcritical, 436–437
supercritical, 436–437, 472
uniform, equation for, 407–417
varied, 448–459, 481
equation for, 450–452
velocity distribution in, 419–420
water surface profiles, 452–459
"wide and shallow," 421–422
Open channels, 407–481
celerity of gravity waves in, 468–471
circular section not full, 426–428
contractions in, 442–447
control section, 441
depression, 447
discharge curve, 433–435
expansion, 447
free overfall, 442, 457, 468, 535
humps in, 442–447
most efficient section, 422–425
nonrectangular, 438–440
specific energy, 431–435
trapezoidal sections, 439
Optical methods of velocity measurement, 501–502
Orifice meter, 522–526
Orifices, 505, 512, 522–526
coefficients of, 505
losses in, 510–513
Orthogonality, 636–639
Outfall sewers, 170, 344
Outlet control, 476–477
Overbank flow, 417
Overflow spillway, 536–538

P

Parallel pipes, 336–339
Partial pressure, 23–24
Particle image velocimetry, 502
Pascal, Blaise, 50
Pascal (unit), 45
Pascal's law, 50
Path:
absolute, 203, 212–213
relative, 203, 212–213
Path lines, 102
Pelton wheel, 686
Penstock, 319, 569, 686–687
Perfect (ideal) gases, 22–24, 27, 580
compressibility of, 25–27
process equations for, 581
property relations for, 22–24, 580
Perfect-gas law, 22, 580
Peripheral velocity factor, 665–666, 711–712
Photographic methods of velocity measurement, 501–502
Physical properties of fluids, 732–736
measurement of, 491–494
Piezometer (column, tube), 58–59, 158
Piezometer connections, 495
Piezometer ring, 495
Piezometric head, 150, 158
Pigott, R. J. S., 284
Pipe bends:
as flow meter, 540
forces on, 194–195, 472
head loss in, 312–314
secondary flow in, 312–313
Pipe factor, 277
Pipe fittings, losses in, 312–314
Pipe flow, 264–301
empirical equations for, 298–300
fully rough, 274–275, 281, 286
unsteady, 550–557
Pipe-flow problems, solution of. *See* Single-pipe flow (problems)
Pipe friction, 137, 145, 255–265
equation, 261
head loss, 136, 261
Pipe network, 339–343
Pipe roughness, 280–298
Pipes:
aging of, 320
branching, 326–333
nonrigorous solutions, 331–333
commercial, 286
compound (*see* Compound pipes)
laminar flow in, 264–267
noncircular, 263
in parallel, 336–339
partly full, 426–428
with pump, 321–325
sag, 343
in series, 333–335
size required, 320
with turbine, 321–325
turbulent flow in, 271–282
unsteady flow in, 550–557
Pipe size required, 320
Pitch diameter, 688
Pi theorem, 247–249, 653
Pitometer, 498
Pitot, Henri, 496
Pitot-static tube, 497–498
Pitot tube, 158–160, 496–500, 524
Pitting, 156, 666, 713
PIV (particle image velocimetry), 502
Plane area, force on, 66–74
Plate, flat, drag of, 358–372
Plumes, 170–171
Point gage, 528, 540
Point of optimum efficiency (BEP), 657
Poise (unit), 33
Poiseuille, Jean Louis, 33, 265
Polar diagram, 395–396
Poles of generator, 685, 709–710
Polynomial solver, 71, 153, 325, 416–417, 435, 445, 475, 746, 748, 754–755, 760–762
Pore velocity, 641
Porous media flow, 639–642
Positive displacement meter, 503
Potential energy, 128–129
Potential flow, 636–639
Potential functions, 635–639
Potential head, 150
Powell R. W., 413*n*
Power:
brake, 652, 706
expressions for, 150–151, 652, 706
fluid, 652
horsepower, 151
of jet, 151, 167, 205
rated, 706*n*
shaft, 652, 701, 706
water, 652, 706
Prandtl, Ludwig, 3, 267, 271, 276, 280, 395
Prandtl-Glauert rule, 400*n*
Prandtl tube, 498
Pressure:
absolute, 53–54, 60
atmospheric, 53–56
barometric, 53–56
boiling, 40
center of, 68–74
critical ratio, 591, 597–599
dynamic, 139, 240, 591
in fluid flow, 138–139
gage, 53–54
measurement of, 55–64, 495–496
negative, 37, 53, 61
partial, 23–24
saturation, 40
stagnation, 139, 589–592, 596–598
static, 139
vapor, 40–41
variation in static fluid, 46–50
water hammer, 560–569
Pressure coefficient, 239
Pressure conduits, force on, 193–197
noncircular, 263
Pressure drag, 357, 372–374
Pressure gage, 57
Pressure head, 50–52, 58, 60, 129, 150
Pressure head gage, 59
absolute, 60
Pressure prism, 67–68
Pressure transducer, 58
Pressure waves, 558–566, 585–588

'ressure waves—*Cont.*
velocity, 585–587
in pipes, 558–566, 588
'ressure zone, 344
'reston tube, 498–499
'rinciple of dimensional homogeneity (PDH), 246
'roblem solving methods, 3–6
'rocess equations, 24, 581
'rofile drag, 394
'rogrammable calculator, 71, 746–747, 754–755
'rogramming, 745–753
'rogramming languages, 751–752
'rojectiles:
drag of, 399–400
flow around, 410–411, 607–609
'ropeller pumps, 648
'ropellers, theory of, 222–225
'ropeller turbines, 697, 700–718, 721–722
'roperty relations, 22, 580
'roportional flow weirs, 532–545
'ropulsion:
jet, 210–211
propeller, 222–225
rocket, 211
'rototype, 232–245, 654
'ulsating flow, 410, 481
'ulsating machine, 211
'ump efficiency, 652, 654, 657, 663
'ump eye, 677, 719
'umps, 212–220, 647–678
axial-flow, 214, 647–678
centrifugal (*see* Centrifugal pumps)
characteristic curves for, 655–657, 675–677
deep-well, 649–650
efficiency of, 652, 654, 657, 663
head developed by, 150, 651–652
mixed-flow, 214, 647, 656–659, 662–663
multi-stage, 649–650
normal capacity of, 656–657
operating point of, 660–662
in parallel, 675–677
in pipeline, 321–325
radial-flow, 212–216, 647–678
rated capacity of, 657
selection of, 671–674
in series, 675–677
specific speed of, 662–664, 669
Pump turbine, 719

R

Radial flow, 212–218, 647–678
Radial-flow pumps, 212–216, 647–678
Radius, hydraulic, 258, 263, 427
Ramjet, 211
Ramp on overflow spillway, 536, 538
Rao, N. S. Lakshmana, 534*n*
Rapid flow, 436
Rapidly varied flow, 448–449
Rated capacity of pump, 657
Rated capacity of turbine, 706*n*, 714
Rated power output of turbine, 706*n*
Rated specific speed, 712
Rate of discharge, 103–105
Rayleigh, Lord, 236, 247, 384
Reaction, of jet, 206–210
with rotation, 220–221
Reaction turbines, 686, 697–722
cavitation in, 713–717
efficiency of, 706–708
selection of, 717–718
specific speed of, 711–712
Real fluid, 29, 97
Rectangular weir, 528–530
Rectilinear flow, 631
Reentrant tube, 303, 509
References, 764–768
Rehbock weir formula, 529
Relative flow rate, 202
Relative path, 203, 212–213
Relative roughness, 280–283
Relative velocity, 117, 200–201
Reservoir, flow from, 317–319
Reverse flow, 114
Reversible process, 581
Reynolds, Osborne, 37*n*, 98, 98*n*, 235, 255
Reynolds number, 235–236, 256–257, 363–384
critical, 256–257, 363, 367–370, 377, 408
local, 361
Reynolds stress, 270
Reynolds transport theorem, 107
River flow, measurement of, 503–504
Rocket propulsion, 211
Roll waves, 481
Rotameter, 541
Rotating channel, equation for flow in, 220
Rotating machines, 212–225
Rotating sprinkler, 220–221
Rotational flow, 625–627
Rotational vortex, 173–176
Rotor, 175, 212, 685
Rotorship, Flettner, 390*n*
Roughness, 280–298
of commercial pipe, 286–287
critical, 370
in open channels, 411–414
Rough-pipe flow, 274–275, 281, 286
Rouse, Hunter, 2*n*, 539*n*
Runaway speed, 221
Runner, turbine, 212, 685, 687, 699–700

S

Sag pipe, 343
Salt-velocity method, 541
Saturation pressure, 40
Saybolt viscometer, 492
Scale ratio, 233–234, 241–242
acceleration, 233
force, 234
length, 233
time, 233
velocity, 233
Schlichting, H., 271*n*, 367*n*, 397*n*
Schlieren photograph, 502
Scientific calculators, 746–747, 754–755
Scroll case, 697
Secondary flow, 312–313, 472–473
Separation of boundary layer, 114–115, 307, 310, 372–378
point of, 115, 373–378
shift in point of, 374–378
Series pipes, 331–335
Seventh-root law, 281
Shadowgraph, 502
Shaft power, 652, 701, 706
Shaft work, 141
Sharp-crested weir, 527–533
Shear stress, 31–32, 268–271
in boundary layers, 358–370
measurement of, 499
in open channel flow, 409–410
in pipe flow, 259–260
in turbulent flow, 268–271
Shear-stress velocity, 273
Sheet flow, 429–430
Shock loss in turbines, 704
Shock stall, 400
Shock wave:
normal, 601–606
oblique, 607–609
Shroud, 648
Shutoff head, 657
SI (International System) units, 7
Similarity:
dynamic, 234–240
geometric, 232–233
kinematic, 233
laws of, for turbomachinery, 652–655, 708–710
restrictions on use of, 653
Similitude, 232–245
Simultaneous equations, 285, 343, 744–747, 753–754, 760
Single-pipe flow (problems):
automated solutions, 296–298
direct solutions, 293–296
empirical equations for, 298–300
fully-rough-pipe flow, 286
governing equations, 285
laminar flow, 285
smooth-pipe flow, 285–286
solution basics, 285–287
solution by trials, 287–292
solution with minor losses, 315–320
Sink, 631–634
Siphon, 134, 243
Size of pipe required, 320
Slipstream, 222
Slope of open channel, 409–410, 454
adverse, 454
critical, 436, 454
mild, 437, 441, 454
steep, 436, 441, 454
Slug (unit), 7
Sluice coefficient, 539
Sluice gate, 433, 538–539
Smooth-pipe flow, 275, 280–281
Software, 748–751
Solid bodies, properties of, 739
Sonic (acoustic) velocity, 139, 558, 585–592, 595–600
Sound wave, 585–588
Source, 631–634
Spatially varied flow, 481
Specific discharge, 641
Specific energy, 431–435
Specific energy diagram, 432
Specific gravity, 15
Specific heat, 23, 144, 580–583
effect on temperature, 144
temperature dependence, 23
of water, 144*n*
Specific heat ratio, 24, 580–582, 736
Specific speed, 662–664, 669, 711–712
limiting values of, 669, 713–714
nominal, 712
normal, 712
rated, 712
SI, 662–663
suction, 668
Specific volume, 15
Specific weight, 14, 19–21
of liquids, 19–21
of water, 20
Speed of sound, 585–588
Speed regulation of turbines, 686, 695–698
Spheres, drag on, 375–381, 493
Spillways:
cavitation on, 538
coefficient for, 537
flow over, 536–538
ogee, 536–538
ramps on, 536, 538
Spiral flow, 312, 472–473
Spiral (volute) casing, 648
Spiral vortex, 175–176, 178–179
Spreadsheets, 71, 153, 452, 747, 755
Square-edged entrance, 303, 509
Stability:
of floating bodies, 84–85
of submerged bodies, 82–83
Stable equilibrium, 83
Staff gage, 540
Stagnation density, 589–591
Stagnation enthalpy, 584
Stagnation hypothesis, 391
Stagnation point, 113–116, 139, 375, 495, 589, 633–634
Stagnation pressure, 139, 589–592, 596–598
Stagnation temperature, 588–592
Stall, point of, 395–396
Standard atmosphere, 27–29, 47–49, 734
Standing wave, 468–471, 481
Stanton diagram, 283, 284*n*
Stanton tube, 498–499
Starting vortex, 392
State, equations of, for gases, 22, 580
Static pressure, 139
measurement of, 495–496
Static (pressure) head, 150, 158, 691
Static tube, 496
Stay ring, 698
Stay vanes, 698
Steady flow, 101–102, 117–120, 131–137, 140–149
acceleration in, 117–120
approach to, 554–557
velocity in, 117–120
Steep slope, 436, 441, 454

Stilling basin, 465
Stilling well, 540
Stoke (unit), 33
Stokes, George G., 33, 188
Stokes' law, 375–376, 493
Stratosphere, 28
Streak line, 102–103
Stream function, 112, 629–630, 638
Streamlined weirs, 533–535
Streamlines, 102, 627, 629, 636–639
Streamlining, 378, 399
Stream tube, 108
Study methods, 3–6
Subcritical flow, 436–437
Submerged bodies, 81–83
Submerged discharge, 160–161, 303–305
Submerged entrance, 476–480
Submerged jet, 160–161, 506, 511–512
Submerged orifice, 512
Submergence of weir, 534–535
  critical, 535
Subsonic velocity, 239, 399, 585, 594
Suction lift, 669
Suction specific speed, 668
Sudden contraction, 305–306
Sudden expansion (enlargement), 307–311
Supercavitation, 401
Supercritical flow, 436–437, 472
Superposition of flow fields, 388, 632
Supersonic flow, 239, 399, 585, 594
Supersonic velocity, 239, 399, 585, 594
Suppressed weir, 528
Surface drag, 357
Surface elevation, measurement of, 540
Surface profiles, 452–459
Surface tension, 37–39, 509*n*
Surge, 465, 481
Surge tanks (chambers), 569–573
Sutro weir, 532
Swirl, 100
Symbols, list of, xix–xxiii
Système Internationale d'Unités (SI), 7

T

Tailrace, 702–704
Tailwater, 477, 479
Tangential acceleration, 119–120
Taylor, G. I., 384
Temperature, absolute zero, 7, 22*n*
Thermal energy, 141–145
Thermodynamics, 14, 580–583
  first law of, 141, 581
  second law of, 144, 581
Thin-plate weirs, 527–533
Thomson's theorem, 392
Three-dimensional flow, 110–111, 374–381
Three-reservoir problem, 328, 330–331
Throat, 157, 163
Tidal bore, 465, 481
Time scale ratio, 233
Tip vortices, 392–393
Torque, 32, 173, 216–218, 626
Total enthalpy, 584
Total head, 133, 150, 158
Trajectory of jet, 169–170
Tranquil flow, 436
Transitional flow, 369–372
Transitionally rough flow, 274
Transition from laminar to turbulent flow, 255–256, 369–372
Transitions, 474–475
Transonic flow, 399
Trapezoidal channels, 439
Trapezoidal weir, 531
Trials, trial and error, 4, 6, 71, 153, 287–292, 316–320, 325–342, 415–416, 435, 475, 480, 573, 613
Triangular weir, 531–532
Troposphere, 28
Tube coefficients, 509
Tubes, 505, 509–510
  Borda (mouthpiece), 509–510
  direction-finding, 496–498
  draft, 702–705
  losses in, 510–513
  piezometer, 58–59, 158
  pitot, 158–160, 496–500, 524
  pitot-static, 497–498
  Prandtl, 498
  Preston, 498–499
  Stanton, 498–499
  static, 496
  venturi, 515–518, 525–526
Tube turbine, 700–701
Turbine meter, 541
Turbine pump, 648
Turbines, 212–220, 685–722
  axial flow, 214, 697, 700–718, 721–722
  cavitation in, 713–717
  Deriaz, 701
  efficiency of, 691–694, 706–708
  Francis, 697–701, 706–708, 712–718, 721
  impulse (*see* Impulse turbines)
  Kaplan, 698, 700, 706, 721–722
  mixed-flow, 698
  Pelton, 686
  in pipeline, 321–325
  propeller, 697, 700–718, 721–722
  reaction (*see* Reaction turbines)
  selection of, 717–718
  specific speed of, 711–712
  tube, 700–701
Turbojet, 211
Turbomachinery, similarity laws for, 652–655, 708–710
Turbulence, 104, 268–269
  level of, 377
  measurement of, 501
Turbulent boundary layer, 272, 360, 365–368
Turbulent flow, 99–101, 255–256, 268–282
  in boundary layers, 360, 365–368
  in circular pipes, 268–282
  fully developed, 272
  around spheres, 375–381
Turbulent fluctuation of velocity, 104, 269–271, 501
Turbulent mixing, 344
Turbulent shear, 268–271
Turbulent wake, 115, 374–378
Two-dimensional flow, 110–111, 382–384, 628, 630, 638

U

Unestablished flow, 265–266
Uniform flow, 101, 407–417
Units, 6–11
  abbreviations for, xxv, 9
  conversion of, 9–11
  derived, 7
  Newton, 7
  popular usage, 9
  prefixes, 10
  slug, 7
Unit systems, 6–11
  absolute, 8
  Absolute Metric (cgs), 9
  British Gravitational (BG), 7
  consistent, 9
  English, 7
  English Engineering (EE), 9
  gravitational, 8
  inconsistent, 9
  International System (SI), 7
  mks metric, 9
  U.S. Customary, 7
Universal gas constant, 23
Unsteady flow, 101–102, 121–123, 546–574
  acceleration in, 121–123
  in pipes, 550–557
  velocity in, 121–123
U.S. Customary (USC) units, 7
U.S. standard atmosphere, 28

V

Vacuum, 53
Vacuum gage, 57
Valve closure, 559–569
Valve opening, 554–556
Valves:
  air, 569
  automatic relief, 569
  head loss in, 312
  types, 312, 650
Vaned elbow, 313
Vanes, force on, 198–206
Vanoni, V. A., 419*n*
Vapor, 13–14
Vapor pressure, 40–41
  saturation, of selected liquids, 40
Vapor trails, 394
Varied flow, 448–459
Varying head, 546–548
VDI orifice meter, 522–523
Velocity:
  absolute, 200–201
  acoustic, 139, 585–592, 595–600
  of approach, 529–530
  critical, 432, 439–440
  downwash, 393
  fluctuation, 104, 269–271
  friction, 273
  of gravity waves, 468–471
  ideal, 508, 511, 515, 539
  jet, 507
  mean, 105
  measurement of, 496–502
    by optical methods, 501–502
    by photographic methods, 501–502
  in open channels, 411, 419–420
  pore, 641
  of pressure waves, 558–559, 585–588
  relative, 117, 200–201
  shear-stress, 273
  sonic, 139, 558, 585–592, 595–600
  subsonic, 239, 399, 585, 594
  supersonic, 239, 399, 585, 594
Velocity coefficient, 508–510
Velocity defect law, 276
Velocity distribution:
  in boundary layers, 266, 361, 365–366
  in open channels, 419–420
  in pipes, 264, 276–279
  seventh-root law of, 281
Velocity factor, peripheral, 665–666, 711–712
Velocity gradient, 31
Velocity head, 150
Velocity measurement, 496–502
Velocity potential, 635–639
Velocity profile, 30, 33, 264–281, 419–420
Velocity scale ratio, 233
Vena contracta, 506–507
Venturi, Giovanni B., 515
Venturi tube (meter), 515–518
  with compressible flow, 525–526
Viscometers, 491–494
Viscosity:
  absolute, 29–35
  coefficient of, 31
  dynamic, 31
  eddy, 269
  effect of temperature, 29–30, 730–731
  kinematic, 33, 731
  kinematic eddy, 269
  measurement of, 491–494
  Newton's equation of, 32
  pressure independence, 33
  units of, 33
Viscous flow, 98, 144
Viscous sublayer, 271–275, 365
  thickness of, 272–275, 371
Visualization of flow, 502
V-notch weir, 531–532
Volume flow rate, 103–105
Volume modulus of elasticity, 17–18, 25
Volumes, properties of, 739
Volumetric losses, of turbine, 692
Volute (spiral) casing, 648
von Kármán, T., 281
Vortex:
  cylindrical, 173–177
  forced, 173–176
  free, 176–179
  irrotational, 176–179
  rotational, 173–176
  spiral, 175–176, 178–179
  starting, 392
  strength of, 386
  tip, 392–393

Vortex center, 386
Vortex street, 382–384
Vorticity, 627–629

W

Wake, 115, 374–378
Wall effect, 493
Wall friction, 137, 145
  head loss, 136
Water, physical properties of, 732
Water hammer, 559–569
  computer techniques, 568
  pressures, 560–569
  protection from, 568–569
Water power, 706, 752
Water surface profiles, 452–459
Water surface slope, 409
Water table, 641
Wave angle, 470
Waves:
  celerity of, in pipes, 558–566, 585–588
  gravity, 468–471
Weber, Moritz, 239
Weber number, 239, 529
Wedges, 481
Weight flow rate, 103–105
Weirs, 443, 527–535
  broad-crested, 443, 534
  Cipolletti, 531
  contracted, 530–531
  dimensional analysis of, 529
  errors in use of, 528
  nappe of, 528
  proportional-flow, 532, 545
  rectangular, 528–530
  streamlined, 533–535
  submergence of, 534–535
  suppressed, 528
  Sutro, 532
  thin-plate, 527–533
  trapezoidal, 531
  triangular, 531–532
  V-notch, 531–532
Weisbach, 261
Wicket gates, 697–698
"Wide and shallow" flow, 421–422
Windmill, 222–225
Wind tunnel, 243, 400
Work, 141
Work-energy relation, 141

Z

Zagustin, K., 538*n*

UNIVERSIDAD POLITECNICA SALESIANA
BIBLIOTECA

DIAZ DE SANTOS - Importador N.60
Alm. 2
Finnemore, E. John
Fluid mechanics: with engineering applications. (
International edition)
9.000 Ptas.
54,09 € Iva Inc. =
SP1109105619
SP04034/022506
Fisica de fluidos
F.E.:02/04/2002

Use the most up-to-date, superiorly accurate textbook available.

Civil engineering instructors and professionals alike will want this title. It explains and applies the phenomena and principles of fluid mechanics in the simplest and clearest manner without using complicated mathematics. No practicing engineer in any branch of civil engineering should be without this classic! Many agricultural, mechanical, and aeronautical engineers will find it valuable, too. While it targets students at the undergraduate level, professionals will want the most current information available—Finnemore and Franzini's *Fluid Mechanics with Engineering Applications*, Tenth Edition.

This highly regarded textbook has been extensively revised but still maintains its traditional approach to teaching fundamental concepts by solving engineering problems. This tenth edition is extraordinarily accurate, and its coverage of pipe flow and open channel flow is superior. There is a clearer and more integrated coverage of all topics in this edition of *Fluid Mechanics with Engineering Applications.* Furthermore, this edition has numerous challenging problems that can optionally be solved with computers and programmable calculators; new example solutions use Excel, Mathcad, and the HP-48G.

The tenth edition includes:

- An increased number of exercises and problems in the book (1354 in total)
  - *Take the theory and put it into practice.*
- 110 new figures that have been added to help present exercises and problems and solutions
  - *Visualize your solutions.*
- A thoroughly revised chapter on Energy in Steady Flow
  - *Currency is the key to success in academia and the professional world.*
- New coverage on computational fluid dynamics, single-pipe flow, hydraulic efficiency of channel flow, methods of measuring fluid velocity using lasers, the types of equations used in fluid mechanics, and software that models flow systems, components, processes, and flow fields
  - *More information ties the ideas together.*
- Expanded descriptions of head losses in pipes, submerged discharge into moving water, conveyance in open channels, hydraulic conductivities of geologic deposits, affinity laws for pumps, and ways to solve trial-and-error problems
  - *Reference material never goes unused.*
- A Solutions Manual, which is available to Instructors
- A website at **www.mhhe.com/finnemore** for book information, author information, web links for further study and reference, and a photo gallery

ISBN 978-007-127011-3
MHID 007-127011-6
9 780071 270113
www.mhhe.com