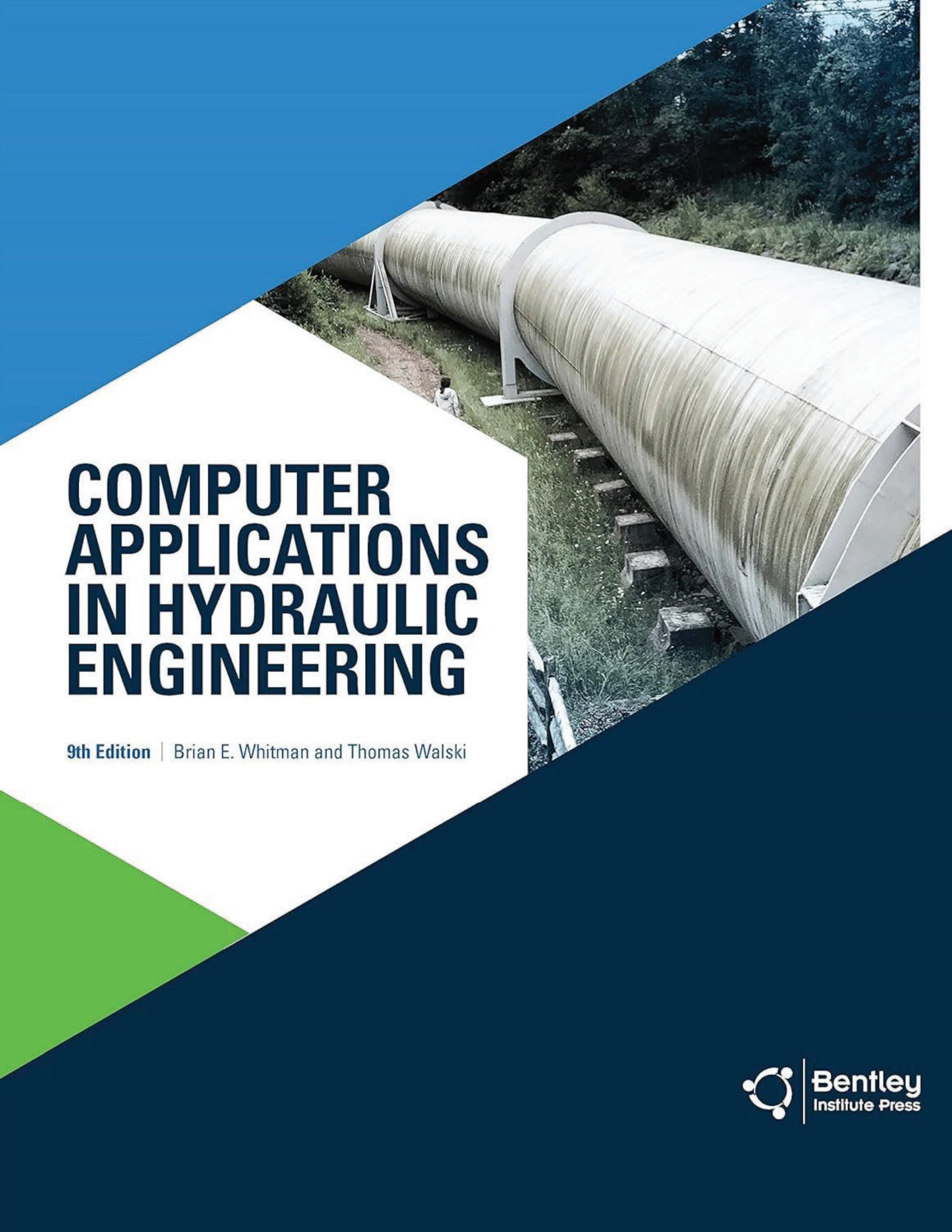




COMPUTER APPLICATIONS IN HYDRAULIC ENGINEERING

9th Edition | Brian E. Whitman and Thomas Walski

Computer Applications in Hydraulic Engineering

9th Edition

Brian E. Whitman
Thomas M. Walski
Thomas E. Barnard
S. Rocky Durrans
Steve Lowry
Michael E. Meadows



Computer Applications in Hydraulic Engineering

9th Edition

Copyright © 2021 Bentley Systems, Incorporated. All Rights Reserved.

Bentley, the “B” Bentley logo, and Bentley Institute Press are either registered or unregistered trademarks or service marks of Bentley Systems, Incorporated or one of its direct or indirect wholly owned subsidiaries. Other brands and product names are trademarks of their respective owners.

The publisher does not warrant or guarantee any of the products described herein or perform any independent analysis in connection with any of the product information contained herein. The publisher also does not assume, and expressly disclaims, any obligation to obtain and include information other than that provided to it by the manufacturer.

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by the activities herein and to avoid all potential hazards. By following the instructions contained herein, the reader willingly assumes all risks in connection with such instructions.

The publisher makes no representation or warranties of any kind, including but not limited to, the warranties of fitness for particular purpose of merchantability, nor are any such representations implied with respect to the material set forth herein, and the publisher takes no responsibility with respect to such material. The publisher shall not be liable for any special, consequential, or exemplary damages resulting, in whole or part, from the readers’ use of, or reliance upon, this material.

ISBN - 978-1-934493-70-0

ISBN (ePub) - 978-1-934493-71-7

LCCN – 2021930961



Published by:
Bentley Institute Press
Bentley Systems, Incorporated
685 Stockton Drive
Exton, PA 19341, United States

www.bentley.com

Printed in the United States

Authors and Editors



Brian E. Whitman, Ph.D. is a Professor of Environmental Engineering at Wilkes University in Wilkes-Barre, Pennsylvania where he was the recipient of the Wilkes University Outstanding Faculty Award in years 2002 and 2005. He holds a M.S. in Civil Engineering and Ph.D. in Environmental Engineering from Michigan Technological University. He has extensive experience teaching hydraulic modeling software for use in water distribution system design, wastewater collection system design, and water resources engineering. He is a contributing author to four books, and has several peer reviewed publications.

Thomas M. Walski, Ph.D., P.E. is Senior Project Manager at Bentley, and is registered as a professional engineer in Pennsylvania and Mississippi. He received his Master's Degree and Ph.D. in Environmental and Water Resources Engineering from Vanderbilt University. He has written a number of authoritative books and several hundred journal and conference papers on the subject of hydraulic modeling. Dr. Walski spent five years as an Associate Professor in the Environmental Engineering Program at Wilkes University.

Contributing Authors



Bentley's Civil Engineering Staff is an extremely diverse group of professionals from six continents with experience ranging from software development and engineering consulting, to public works and academia. This broad cross-section of expertise contributes to the development of the most comprehensive software and educational materials in the civil engineering industry. This book was truly a team effort and many at Bentley (formally Haestad Methods) contributed.

Thomas E. Barnard, Ph.D., P.E. had more than 25 years of experience in environmental engineering working as a consultant, researcher, educator, and author. He holds a B.S. in Civil Engineering from the University of Vermont, an M.S. in environmental engineering from Utah State University, and a Ph.D. in environmental engineering from Cornell University. His expertise includes water and wastewater treatment, hazardous waste management, surface water hydrology and water quality monitoring systems. He was registered professional engineer in Pennsylvania.

S. Rocky Durrans, Ph.D., P.E. was a Professor Emeritus of Civil and Environmental Engineering at the University of Alabama, and was responsible for the hydrologic and hydraulic engineering programs. He holds a Ph.D. from University of Colorado-Boulder. Prior to his academic career, Dr. Durrans spent many years in the consulting arena where he gained extensive experience in design of stormwater and major urban drainage systems. He is regarded as an authority on flood and rainfall analyses, and has many publications to his credit.

Steve Lowry, P.E. is President of Steven G. Lowery & Associates, Inc. and is a registered professional engineer in Colorado and Pennsylvania. He has over 30 years of civil engineering experience in all aspect of project management and performance on project involving hydraulic, water quality, and transient analysis of distribution systems; SCADA system design; hydrologic and safe yield of water supply systems; computer modeling and computer programming; and GIS applications. He has authored several publications regarding water systems, modeling, and model calibration. He holds a B.S. and M.S. in Civil Engineering from the Ohio State University.



Contributing Authors

Michael E. Meadows, Ph.D., P.E. is an Associate Professor in the Department of Civil and Environmental Engineering at the University of South Carolina where he instructs undergraduate and graduate courses on hydraulics and hydrology. He holds a M.S. in Environmental Engineering and Ph.D. in Civil Engineering from University of Tennessee. He has coauthored texts on stormwater modeling and has published numerous papers on improved methods for estimating stormwater discharge and evaluating drainage system performance. Dr. Meadows has also served as special consultant to SC DOT, USACE, USGS, EPA, and other state and municipal agencies in developing, evaluating, and applying water resource modeling methodologies.

Revision History



During the development of each edition of *CAiHE*, we encourage feedback from the hundreds of universities that have adopted this textbook and accompanying software. We are interested in learning more about what professors and students want added to this publication and the way it is being utilized within civil engineering curricula.

In the **Second Edition**, we added tutorial example problems with step-by-step instructions for solving the problems using the included computer software. More problems to be solved by students using the software were added. In addition, updates were incorporated into the software on the CD-ROM.

In the **Third Edition**, we added two new chapters: one chapter on drainage inlet design and the other on sanitary sewer design. These chapters included tutorials and problems. Also, a brief discussion of weirs and orifices as an application of the energy equation was added to Chapter 1. More problems were added to the existing chapters. The CD-ROM enclosed with this text contained new versions of FlowMaster and StormCAD, as well as a new product, SewerCAD. In addition, updates were incorporated into other software on the CD-ROM.

In the **Fourth Edition**, we added discussions on the use of hydrographs and hydrologic routing to the sanitary sewer modeling chapter (Chapter 6). An extended-period simulation problem was added to this chapter as well.

In the **Fifth Edition**, we added a chapter on basic hydrology and a chapter on detention pond design, accompanied by the latest academic version of PondPack. Tutorials throughout the text have been updated and expanded per the latest software versions, and current updates have been incorporated into the software on the CD-ROM. In response to requests from professors, the capacity of the software on the CD-ROM has been increased so that students may work with more complex problems.

In the **Sixth Edition**, we revised the tutorials to reflect revisions to our software.

In the **Seventh Edition**, we have revised the Pressure Piping and Water Quality Analysis chapter (Chapter 6) and updated the material for our newest software. We have also added a brand new Chapter 8 called Dynamic Modeling with full tutorials and problems. Appendix C was deleted and replaced by the previous Appendix D. Appendix A was modified to provide users with updated



Revision History

instructions on how to download the software licenses for each program contained on the academic CD.

In the **Eighth Edition**, we added chapter in transient analysis with tutorials using the HAMMER software (Chapter 7). Tutorials throughout the text have been updated and expanded per the latest software versions, and current updates have been incorporated into the software on the CD-ROM. CD-ROM has been replaced by a procedure to download software from Bentley website.

In this **Ninth Edition**, we added material about Low Impact Development (LID) for stormwater management to chapter 9, and the textbook CD-ROM has been replaced by a procedure to download the latest software from Bentley website. We took all the tutorials out of the print book and placed them on-line so they can be easily updated as the software changes and is improved with new features. We will also placed additional problems to go with all the chapters on this same on-line location. We would like to thank Brandon Whitman who completed, reviewed, and provided improvements for all the tutorials that were updated to the latest Bentley CONNECT editions.

The following table lists the software that accompanies each chapter.

Chapter	Accompanying Software
Chapter 1	FlowMaster
Chapter 2	not required
Chapter 3	StormCAD or SewerGEMS
Chapter 4	CulvertMaster
Chapter 5	PondPack
Chapter 6	WaterGEMS
Chapter 7	HAMMER
Chapter 8	SewerCAD or SewerGEMS
Chapter 9	SewerGEMS

Contacting Bentley



We welcome feedback regarding our products and services; we feel this is the best way for us to continue providing software, training, publications, and support that are tailored to the needs of professionals in the field of hydraulics and hydrology today.

Included with this book is access to download and install academic versions of our award-winning software, WaterGEMS, HAMMER, SewerGEMS, SewerCAD, StormCAD, PondPack, CulvertMaster, and FlowMaster. They have been included to assist you with working the problems contained in this book and to introduce you to the latest developments in computer-aided hydraulic and hydrologic modeling. Details on accessing the software is included in Appendix A.

We hope that once you try the companion software packaged with CAiHE, you'll discover many ways in which it can enhance your work outside of this book. You can get more products and upgrade information on any of our software by calling our sales department or by visiting the Bentley's OpenFlows website:

Sales

<https://www.bentley.com/en/about-us/contact-us/sales-contact-requestorcalling>

OpenFlows

<https://www.bentley.com/en/products/brands/openflows>

North America

1-800-BENTLEY (1 800 236 8539)
+1 610 458 5000

Europe, Middle East, and Africa

1800 948 368
+353 1 436 4600

Asia

400 842 7516
+86 10 5929 7000



Contacting Bentley

Bentley also works to bring the global civil engineering community together with our online forums. Get in touch with fellow professionals worldwide and discuss the latest tips and tricks in hydraulic and hydrologic modeling. Post your engineering questions and share your unique modeling experiences with an audience of thousands of professionals. Participants from around the world include many industry-recognized experts and users like you. Get connected today by visiting:

<http://communities.bentley.com>

https://communities.bentley.com/products/hydraulics___hydrology/

If you have any comments regarding this publication and/or any of Bentley products and services, please contact us at:

Bentley Systems, Inc.
685 Stockton Drive
Exton, PA 19341 USA
Voice: 1-800-BENTLEY (1 800 236 8539)
Email: sales@bentley.com
Internet: www.bentley.com

Foreword



Bentley Systems, Inc. has developed hydrology and hydraulics software for civil engineers and provides textbooks, training, and technical support to tens of thousands of professional civil engineers, modelers, and universities.

Along the way, we have learned a lot about our clients and their professional and educational backgrounds. Our continuing education program offers courses for professionals who need to quickly get up-to-speed with various numerical methods and practices.

Why is this important?

Our experience has shown us a great deal about the areas where engineers are being trained, and it has also revealed a significant gap in this training—the link between hydraulic theory and practical computer applications.

Hundreds of textbooks exist that offer enormous detail in the areas of engineering history, equation derivations, and hand calculation methods. There are also hundreds of published theses and articles that deal with computer applications—unfortunately, most of these are highly research oriented and are usually tied to a specific case study or an unusual set of circumstances. Both of these publication types are very important to the civil engineering industry, but in the majority of cases, they fail to address the issues most commonly encountered by professionals in day-to-day practice.

This publication is intended as an introduction to the more common applications of water resources engineering software, and it demonstrates the types of situations that an engineer will most likely come across on a daily basis in the real world. It shows the true benefits of computer software: increased efficiency, better flexibility, and—most important—an increased ability to try different and better designs.

It is our hope that engineers, technicians, and students will find this book to be challenging, but also easy to understand and very practical. Combined with standard hydraulic references, we believe that this text provides many of the tools needed to successfully proceed with a career in the fields of hydrology and hydraulics.

Contents



Authors and Editors	iii
Contributing Authors	v
Revision History	vii
Contacting Bentley	ix
Foreword	xi
Chapter 1: Basic Hydraulic Principles	1
Chapter 2: Basic Hydrology	29
Chapter 3: Inlets, Gravity Piping Systems, and Storm Sewer Design	81
Chapter 4: Culvert Hydraulics	113
Chapter 5: Detention Pond Design	127
Chapter 6: Pressure Piping Systems and Water Quality Analysis	161
Chapter 7: Hydraulic Transients in Distribution Systems	195
Chapter 8: Sanitary Sewer Design	211
Chapter 9: Low-Impact Development and Dynamic Modeling	235
Bibliography	261
Bentley Systems Software	267
Scenario Management	275
Index	289

1

Basic Hydraulic Principles

1.1 General Flow Characteristics

In hydraulics, as with any technical topic, a full understanding cannot come without first becoming familiar with basic terminology and governing principles. The basic concepts discussed in the following pages lay the foundation for the more complex analyses presented in later chapters.

System of Units

The basis for the problems in this book is real-world systems designed and constructed using U.S. customary units. Those units are retained in the problems.

In response to the goal to convert to a globally consistent system of units, many agencies in the United States changed their regulations to require engineering calculations and measurements in the International System of Units (SI). Even though a new design may require SI units, likely the existing engineering system, of which the new design is to become a part, was designed and constructed using U.S. customary units. Consequently, proper system evaluation will require the engineer to use both systems of units. This requirement will persist for many years. Therefore it is essential that students, engineers, regulators, and planners recognize and use contemporary units and, often, a dual system of units.

Flow Rate and Velocity

There are different measures for “how much” fluid is moving through a system. In hydraulic systems, the most common measure is the **volumetric flow rate**. This is a measure of the volume of fluid passing a location with time. Other measures of flow rate can be determined by multiplying the volumetric flow rate by the density of the fluid (mass flow rate), or the specific weight of the fluid (weight flow rate).

The **velocity** of a fluid is a measure of “how fast” a fluid is moving through a system. As shown in Figure 1.1, the **velocity** of a section is not constant throughout the cross-sectional area. Instead, it varies with location. The velocity is zero where the fluid is in contact with the conduit wall.

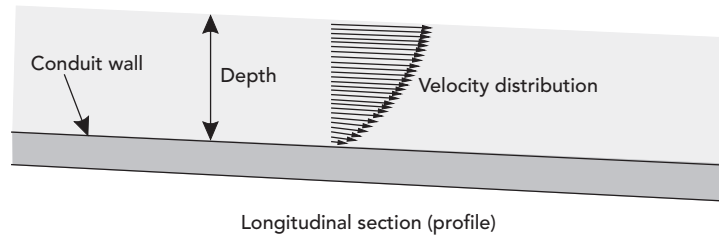


Figure 1.1: Velocity Distribution.

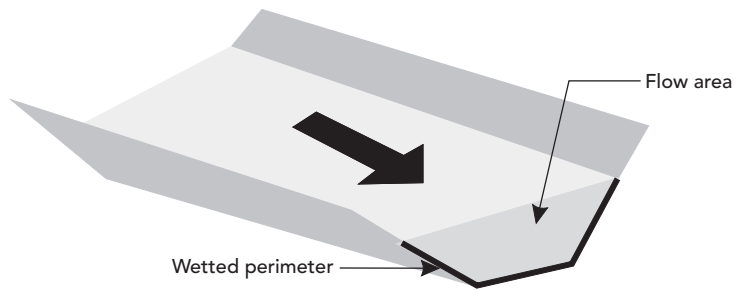


Figure 1.2: Flow Area and Wetted Perimeter.

The variation of flow velocity within a cross section complicates the hydraulic analysis, so the engineer usually simplifies the situation by looking at the average (mean) velocity of the section for analysis purposes. This average velocity is defined as the total flow rate divided by the cross-sectional area and is in units of length per time.

$$V = \frac{Q}{A}$$

where

V = average velocity (m/s, ft/s)

Q = flow rate (m³/s, ft³/s)

A = area (m², ft²).

Area, Wetted Perimeter, and Hydraulic Radius

The term **area** refers to the cross-sectional area of flow within a channel. When a channel has a consistent cross-sectional shape, slope, and roughness, it is called a **prismatic** channel.

A section's **wetted perimeter** is defined as the portion of the channel in contact with the flowing fluid. This definition is illustrated in Figure 1.2.

The **hydraulic radius** of a section is not a directly measurable characteristic, but it is used frequently during calculations. It is defined as the area divided by the wetted perimeter and therefore has units of length.

The hydraulic radius can often be related directly to the geometric properties of the channel. For example, the hydraulic radius of a full circular pipe (such as a pressure pipe) can be directly computed as:

$$R = \frac{A}{P_w}$$

or

$$R_{\text{circular}} = \frac{\pi \cdot D^2/4}{\pi \cdot D} = \frac{D}{4}$$

where

R = hydraulic radius (m, ft)

A = cross-sectional area (m², ft²)

P_w = wetted perimeter (m, ft)

D = inside pipe diameter (m, ft).

Flow Conveyance

Water travels downhill from points of higher energy to points of lower energy (unless forced to do otherwise) until it reaches a point of equilibrium, such as an ocean. This tendency is facilitated by the presence of natural conveyance channels such as brooks, streams, and rivers. The water's journey may also be aided by man-made structures such as drainage swales, pipes, culverts, and canals. Hydraulic concepts can be applied equally to both man-made structures and natural features.

If the flow in a conveyance section is open to the atmosphere, such as in a culvert flowing partially full or in a river, it is said to be **open-channel flow** or **free-surface flow**. If a channel is flowing completely full, as with a water distribution pipe, it is said to be operating under **full-flow** conditions. **Pressure flow** is a special type of full flow in which forces on the fluid cause it to push against the top of the channel as well as the bottom and sides. These forces may result from, for example, the weight of a column of water in a backed up sewer manhole or elevated storage tank.

Steady Flow

Speaking in terms of “flow,” the word **steady** indicates that a constant flow rate is assumed throughout an analysis. In other words, the flow velocity does not change with respect to time at a given location. For most hydraulic calculations, this assumption is reasonable. A minimal increase in model accuracy does not warrant the time and effort that would be required to perform an analysis with changing (**unsteady**) flows over time.

When analyzing tributary and river networks, storm sewers, and other collection systems in which it is desirable to vary the flow rate at different locations

throughout the system, the network can often be broken into segments that can be analyzed separately under steady flow conditions.

Uniform Flow

Uniform flow refers to the hydraulic condition in which the discharge and cross-sectional area (and therefore velocity) are constant throughout the length of the channel. For a pipe flowing full, the only required assumptions are that the pipe be straight and have no contractions or expansions. For an open channel, additional assumptions include the following:

- The depth of flow must be constant [i.e., the hydraulic grade line (HGL) must be parallel to the channel bed]. This depth of flow is called **normal** depth.
- Because the velocity is constant, the velocity head does not change through the length of the section; therefore the energy grade line (EGL) is parallel to both the HGL and the channel bed.

In channels that are prismatic, the flow conditions will typically approach uniform flow if the channel is sufficiently long. When this occurs, the net force on the fluid approaches zero because the gravitational force is equal to the opposing friction forces from the channel bottom and walls. **Nonuniform flow** is the hydraulic condition in which the discharge and cross-sectional area (and therefore velocity) are not constant throughout the length of the channel. In open channels, the velocity profile and cross-sectional area of flow will gradually change. The analysis of gradually varied flow is described in Chapters 3.6 and 9.4

Laminar Flow, Turbulent Flow, and Reynolds Number

Laminar flow is characterized by smooth, predictable **streamlines** (the paths of single fluid particles). An example of this type of flow is maple syrup being poured. In **turbulent** flow, the streamlines are erratic and unpredictable. Turbulent flow is characterized by the formation of eddies within the flow, resulting in continuous mixing throughout the section (see Figure 1.3).

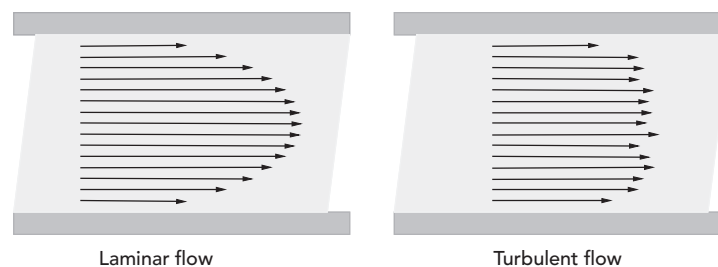


Figure 1.3: Instantaneous Velocity Distributions for Laminar and Turbulent Flow.

Eddies result in varying velocity directions as well as magnitudes (varying directions not depicted in Figure 1.3 for simplicity). At times, the eddies contribute to the velocity of a given particle in the direction of flow and at other times detract from it. The result is that velocity distributions captured at different times will be quite different from one another and will be far more chaotic than the velocity distribution of a laminar flow section.

By strict interpretation, the changing velocities in turbulent flow would cause it to be classified as unsteady flow. Over time, however, the average velocity at any given point within the section is essentially constant, so the flow is assumed to be steady.

The velocity at any given point within the turbulent section will be closer to the mean velocity of the entire section than with laminar flow conditions. Turbulent flow velocities are closer to the mean velocity because of the continuous mixing of flow, particularly the mixing of low-velocity flow near the channel walls with the higher velocity flow toward the center.

To classify flow as either turbulent or laminar, an index called the **Reynolds number** is used. It is computed as follows:

$$Re = \frac{4VR}{\nu}$$

where

Re = Reynolds number (unitless)

V = average velocity (m/s, ft/s)

R = hydraulic radius (m, ft)

ν = kinematic viscosity (m²/s, ft²/s).

If the Reynolds number is below 2,000, the flow is generally laminar. For flow in closed conduits, if the Reynolds number is above 4,000, the flow is generally turbulent. Between 2,000 and 4,000, the flow may be either laminar or turbulent, depending on how insulated the flow is from outside disturbances. In open channels, laminar flow occurs when the Reynolds number is less than 500 and turbulent flow occurs when it is above 2,000. Between 500 and 2,000, the flow is transitional.

Example 1.1: Flow Characteristics

A rectangular concrete channel is 3-m wide and 2-m high. The water in the channel is 1.5-m deep and is flowing at a rate of 30 m³/s. Determine the flow area, wetted perimeter, and hydraulic radius. Is the flow laminar or turbulent?

Solution

From the section's shape (rectangular), we can easily calculate the area as the rectangle's width multiplied by its depth. Note that the depth used should be the actual depth of flow, not the total height of the cross section. The wetted perimeter can also be found easily through simple geometry.

$$A = 3.0 \text{ m} \times 1.5 \text{ m} = 4.5 \text{ m}^2$$

$$P_w = 3.0 \text{ m} + 2 \times 1.5 \text{ m} = 6.0 \text{ m}$$

$$R = A/P_w = 4.5 \text{ m}^2/6.0 \text{ m} = 0.75 \text{ m}$$

In order to determine whether the flow is likely to be laminar or turbulent, we must determine the Reynolds number. To do this, first find the velocity of the section and a value for the kinematic viscosity.

$$V = Q/A = 30 \text{ m}^3/\text{s}/4.5 \text{ m}^2 = 6.67 \text{ m/s}$$

From fluids reference tables, we find that the kinematic viscosity for water at 20°C is $1.00 \times 10^{-6} \text{ m}^2/\text{s}$. Substituting these values into the formula to compute the Reynolds number results in

$$Re = (4 \times 6.67 \text{ m/s} \times 0.75 \text{ m})/(1.00 \times 10^{-6}) = 2 \times 10^7$$

This value is well above the Reynolds number minimum of 4,000 for turbulent flow.

1.2 Energy

The Energy Principle

The first law of thermodynamics states that for any given system, the change in energy (ΔE) is equal to the difference between the heat transferred to the system (Q) and the work done by the system on its surroundings (W) during a given time interval.

The energy referred to in this principle represents the total energy of the system, which is the sum of the potential energy, kinetic energy, and internal (molecular) forms of energy such as electrical and chemical energy. Although internal energy may be significant for thermodynamic analyses, it is commonly neglected in hydraulic analyses because of its relatively small magnitude.

In hydraulic applications, energy values are often converted into units of energy per unit weight, resulting in units of length. Using these length equivalents gives engineers a better “feel” for the resulting behavior of the system. When using these length equivalents, the engineer is expressing the energy of the system in terms of “head.” The energy at any point within a hydraulic system is often expressed in three parts, as shown in Figure 1.4:

- pressure head p/γ
- elevation head z
- velocity head $V^2/2g$

where

p = pressure (N/m^2 , lb/ft^2)

γ = specific weight (N/m^3 , lb/ft^3)

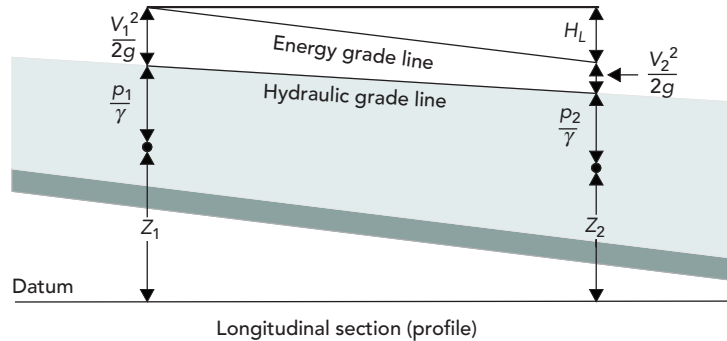


Figure 1.4: The Energy Principle.

z = elevation (m, ft)

V = velocity (m/s, ft/s).

Note that a point on the water surface of an open channel will have not only a pressure head of zero but also a positive elevation head higher than that of a point selected at the bottom of the channel for the same station.

The Energy Equation

In addition to pressure head, elevation head, and velocity head, energy may be added to a system by a pump, for example, and removed from the system by friction or other disturbances. These changes in energy are referred to as **head gains** and **head losses**, respectively. Because energy is conserved, the energy across any two points in the system must balance. This concept is demonstrated by the energy equation:

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} + H_G = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} + H_L$$

where

p = pressure (N/m², lb/ft²)

γ = specific weight of the fluid (N/m³, lb/ft³)

z = elevation above a datum (m, ft)

V = fluid velocity (m/s, ft/s)

g = gravitational acceleration (m/s², ft/s²)

H_G = head gain, such as from a pump (m, ft)

H_L = combined head loss (m, ft).

Hydraulic Grade

The **hydraulic grade** is the sum of the pressure head (p/γ) and elevation head (z). For open-channel flow (in which the pressure head is zero), the hydraulic grade

elevation is the same as the water surface elevation. For a pressure pipe, the hydraulic grade represents the height to which a water column would rise in a piezometer (a tube open to the atmosphere rising from the pipe). When the hydraulic grade is plotted as a profile along the length of the conveyance section, it is referred to as the **HGL**.

Energy Grade

The **energy grade** is the sum of the hydraulic grade and the velocity head ($V^2/2g$). This grade is the height to which a column of water would rise in a pitot tube (an apparatus similar to a piezometer, but also accounting for fluid velocity). When plotted in profile, this parameter is often referred to as the **EGL**. For a lake or reservoir in which the velocity is essentially zero, the EGL is equal to the HGL.

Energy Losses and Gains

Energy (or head) losses (H_L) in a system are due to a combination of several factors. The primary cause of energy loss is usually the internal friction between fluid particles traveling at different velocities. Secondary causes of energy loss are localized areas of increased turbulence and disruption of the streamlines, such as disruptions from valves and other fittings in a pressure pipe, or disruptions from a changing section shape in a river.

The rate at which energy is lost along a given length of channel is called the **friction slope** and is usually presented as a unitless value or in units of length per length (ft/ft, m/m, etc.).

Energy is generally added to a system with a device such as a pump. Pumps are discussed in more detail in Chapter 6.

Example 1.2: Energy Principles

A 1,200-mm diameter transmission pipe carries 126 l/s from an elevated storage tank with a water surface elevation of 540 m. Two kilometers from the tank, at an elevation of 434 m, a pressure meter reads 586 kPa. If there are no pumps between the tank and the meter location, what is the rate of head loss in the pipe? (Note: 1 kPa = 1,000 N/m².)

Solution

Begin by simplifying the energy equation. Assume that the velocity within the tank is negligible, and that the pressure head at the tank can be discounted because it is open to the atmosphere. Rewriting the energy equation and entering the known values, we can solve for head loss. The velocity can be calculated using the flow rate and pipe diameter.

$$Q = 126 \text{ l/s} \times (1 \text{ l/s} / 10^3 \text{ m}^3/\text{s}) = 0.126 \text{ m}^3/\text{s}$$

$$A = \pi \times (0.6 \text{ m})^2 = 1.13 \text{ m}^2$$

$$V = Q/A = 0.126 \text{ m}^3/\text{s} / 1.13 \text{ m}^2 = 0.11 \text{ m/s}$$

$$V^2/2g = (0.11 \text{ m/s})^2 / (2 \times 9.81 \text{ m/s}^2) = 0.0006 \text{ m (negligible)}$$

Neglecting the velocity simplifies the energy equation even further, and we can now solve for head loss as

$$H_L = 540 \text{ m} - 434 \text{ m} - (586,000 \text{ N/m}^2)/9,810 \text{ N/m}^3 = 46.27 \text{ m}$$

The rate of head loss (or friction slope) can now be computed as

$$\text{Friction slope} = 46.27 \text{ m}/(2 \times 1,000 \text{ m}) = 0.023 \text{ m/m, or } 23 \text{ m/km}$$

1.3 Friction Losses

There are many equations that approximate the friction losses associated with the flow of a liquid through a given section. Commonly used methods include-Manning's equation

- Chézy's (Kutter's) equation
- Hazen-Williams equation
- Darcy-Weisbach (Colebrook-White) equation.

These equations can be described by a generalized friction equation:

$$V = kCR^x S^y$$

where

V = mean velocity

C = flow resistance factor

R = hydraulic radius

S = friction slope

x, y = exponents

k = factor to account for empirical constants, unit conversion, etc.

The lining material of the flow channel usually determines the flow resistance or roughness factor, C . However, the ultimate value of C may be a function of the channel shape, depth, and fluid velocity.

Manning's Equation

Manning's equation is the most commonly used open-channel flow equation. The roughness component, C , is typically assumed to be constant over the full range of flows and is represented by a Manning's roughness value, n . These n -values have been experimentally determined for various materials and should not be used with fluids other than water. Manning's equation is:

$$V = \frac{k}{n} R^{2/3} S^{1/2}$$

where

V = mean velocity (m/s, ft/s)

k = 1.49 for U.S. customary units, 1.00 for SI units

n = Manning's roughness value

R = hydraulic radius (m, ft)
 S = friction slope (m/m, ft/ft).

Chézy's (Kutter's) Equation

The Chézy equation, in conjunction with Kutter's equation, is widely used in sanitary sewer design and analysis. The roughness component, C , is a function of the hydraulic radius, friction slope, and lining material of the channel. The Chézy equation is:

$$V = C\sqrt{RS}$$

where

V = mean velocity (m/s, ft/s)
 C = roughness coefficient (see following calculation)
 R = hydraulic radius (m, ft)
 S = friction slope (m/m, ft/ft).

The roughness coefficient, C , is related to Kutter's n through Kutter's equation. Note that the n -values used in Kutter's equation are actually the same as Manning's n coefficients.

U.S. Standard Units

$$C = \frac{41.65 + \frac{0.00281}{S} + \frac{1.811}{n}}{1 + \frac{\left(41.65 + \frac{0.00281}{S}\right)n}{\sqrt{R}}}$$

S.I. Units

$$C = \frac{23 + \frac{0.00155}{S} + \frac{1}{n}}{1 + \frac{\left(23 + \frac{0.00155}{S}\right)n}{\sqrt{R}}}$$

where

C = roughness coefficient
 n = Manning's roughness value
 R = hydraulic radius (m, ft)
 S = friction slope (m/m, ft/ft).

Hazen–Williams Equation

The Hazen–Williams equation is most frequently used in the design and analysis of pressure pipe systems. The equation was developed experimentally and therefore should not be used for fluids other than water (and only within temperatures normally experienced in potable water systems). The Hazen–Williams equation is:

$$V = kCR^{0.63}S^{0.54}$$

where

V = mean velocity (m/s, ft/s)
 k = 1.32 for U.S. customary units, or 0.85 for SI units

C = Hazen–Williams roughness coefficient (unitless)
 R = hydraulic radius (m, ft)
 S = friction slope (m/m, ft/ft).

Darcy–Weisbach (Colebrook–White) Equation

The Darcy–Weisbach equation is a theoretically based equation commonly used in the analysis of pressure pipe systems. It applies equally well to any flow rate and any incompressible fluid and is general enough to be applied to open-channel flow systems. In fact, the ASCE Task Force on Friction Factors in Open Channels (1963) supported the use of the Darcy–Weisbach equation for free-surface flows. This recommendation has not yet been widely accepted because the solution to the equation is difficult and not easily computed using noncomputerized methods. With the increasing availability of computer solutions, the Darcy–Weisbach equation will likely gain greater acceptance because it successfully models the variability of effective channel roughness with channel material, geometry, and velocity.

The roughness component in the Darcy–Weisbach equation is a function of both the channel material and the Reynolds number, which varies with velocity and hydraulic radius.

$$V = \sqrt{\frac{8g}{f} RS}$$

where

V = flow velocity (m/s, ft/s)
 g = gravitational acceleration (m/s², ft/s²)
 f = Darcy–Weisbach friction factor (unitless)
 R = hydraulic radius (m, ft)
 S = friction slope (m/m, ft/ft).

The Darcy–Weisbach friction factor, f , can be found using the Colebrook–White equation for fully developed turbulent flow, as follows:

Free surface

$$\frac{1}{\sqrt{f}} = -2 \log \left(\frac{k}{12R} + \frac{2.51}{Re\sqrt{f}} \right)$$

Full flow (closed conduit)

$$\frac{1}{\sqrt{f}} = -2 \log \left(\frac{k}{14.8R} + \frac{2.51}{Re\sqrt{f}} \right)$$

where

k = roughness height (m, ft)
 R = hydraulic radius (m, ft)
 Re = Reynolds number (unitless).

This iterative search for the correct value of “ f ” can become quite time-consuming for hand computations and computerized solutions of many pipes. Another

method, developed by Swamee and Jain, solves directly for f in full-flowing circular pipes. This equation is:

$$f = \frac{1.325}{\left[\ln \left(\frac{k}{3.7D} + \frac{5.74}{Re^{0.9}} \right) \right]^2}$$

where

f = friction factor (unitless)

k = roughness height (m, ft)

D = pipe diameter (m, ft)

Re = Reynolds number (unitless).

Typical Roughness Factors

Typical pipe roughness values for each of these methods are shown in Table 1.1. These values will vary depending on the manufacturer, workmanship, age, and other factors. For this reason, Table 1.1 should be used only as a guideline.

1.4 Pressure Flow

For pipes flowing full, many of the friction loss calculations are greatly simplified because the flow area, wetted perimeter, and hydraulic radius are all functions of pipe radius (or diameter). Table 1.2 presents the three pipe friction loss equations that are commonly used to design pressure pipe systems.

There is much more information presented about pressure piping systems in Chapter 6, including further discussion on pumping systems, minor losses, and network analysis.

Example 1.3: Pressure Pipe Friction Losses

Use the FlowMaster program to compare the head loss computed by the Hazen–Williams equation to the head loss computed by the Darcy–Weisbach equation for a pressure pipe having the following characteristics: 12-in diameter cast iron pipe (new) one mile in length with a flow rate of 1,200 gal/min (with water at 65°F).

Solution

Although there are no elevations or pressures given, these values are not needed to determine the head loss in the pipe. Setting up FlowMaster to solve for the “Elevation at 1” allows us to use zero elevation and zero pressure assumptions and fill in the rest of the pipe characteristics.

For the Hazen–Williams equation, a C coefficient of 130 is assumed. This value results in 18.8 ft of head loss (which agrees with the computed 18.8-ft elevation at point 1). Using Darcy–Weisbach, a roughness height of 0.00085 ft is assumed. The solution indicates a head loss of 18.9 ft, which is only a 0.1-ft difference from the value predicted by Hazen–Williams.

Table 1.1: Typical Roughness Coefficients

Material	Manning's Coefficient n	Hazen-Williams C	Darcy-Weisbach Roughness Height	
			k (mm)	k (ft)
Asbestos cement	0.011	140	0.0015	0.000005
Brass	0.011	135	0.0015	0.000005
Brick	0.015	100	0.6	0.002
Ductile (cast) iron, new	0.012	130	0.26	0.00085
Concrete:				
Steel forms	0.011	140	0.18	0.006
Wooden forms	0.015	120	0.6	0.002
Centrifugally spun	0.013	135	0.36	0.0012
Copper	0.011	135	0.0015	0.000005
Corrugated metal	0.022	–	45	0.15
Galvanized iron	0.016	120	0.15	0.0005
Glass	0.011	140	0.0015	0.000005
Lead	0.011	135	0.0015	0.000005
Plastic	0.009	150	0.0015	0.000005
Steel:				
Coal-tar enamel	0.010	148	0.0048	0.000016
New unlined	0.011	145	0.045	0.00015
Riveted	0.019	110	0.9	0.003
Wood stave	0.012	120	0.18	0.0006

Table 1.2: Three Pipe Friction Loss Equations

Equation	Q (m ³ /s); D (m)	Q (cfs); D (ft)	Q (gpm); D (in.)
Darcy-Weisbach	$S_f = \frac{0.083 f Q^2}{D^5}$	$S_f = \frac{0.025 f Q^2}{D^5}$	$S_f = \frac{0.031 f Q^2}{D^5}$
Hazen-Williams	$S_f = \frac{10.7}{D^{4.87}} \left(\frac{Q}{C} \right)^{1.852}$	$S_f = \frac{4.73}{D^{4.87}} \left(\frac{Q}{C} \right)^{1.852}$	$S_f = \frac{10.5}{D^{4.87}} \left(\frac{Q}{C} \right)^{1.852}$
Manning	$S_f = \frac{10.3(nQ)^2}{D^{5.33}}$	$S_f = \frac{4.66(nQ)^2}{D^{5.33}}$	$S_f = \frac{13.2(nQ)^2}{D^{5.33}}$

Discussion

If the same system is analyzed with 2,000–3,000 gal/min of flow, the difference in head loss between the two equations becomes almost 10 ft.

Why such a big difference? For starters, the two methodologies are completely unrelated, and the estimated roughness coefficients were taken from a list of approximate values. If the Hazen–Williams equation is used with a roughness value of 125, the results are much closer. This difference should emphasize the fact that models are only as good as the data that is input into them, and the engineer needs to fully understand all of the assumptions that are being made before accepting the results.

1.5 Open-Channel Flow

Open-channel flow analysis is more complex than pressure flow analysis because the flow area, wetted perimeter, and hydraulic radius are not necessarily constant as they are in a uniform pipe section under full-flow conditions. Because of this considerable difference, additional characteristics become important when dealing with open-channel flow.

Specific Energy and Critical Flow

Of course, channels do not always flow at normal depth. If they did, it would make the engineer's task quite simple. A more in-depth at nonuniform flow is presented in Chapters 3.6 and 9.4, but this chapter will continue by focusing on another important concept—"specific energy."

For any flow section, the *specific energy* is defined as the sum of the depth of flow and the velocity head.

$$E = y + \frac{V^2}{2g}$$

where

E = specific energy (m, ft)

y = depth of flow (m, ft)

V = mean velocity (m/s, ft/s)

g = gravitational acceleration (m/s², ft/s²).

If we assume the special case of an infinitely short section of open channel (with essentially no friction losses and no change in elevation), we see that the general energy equation can be reduced to an equality of specific energies. In other words,

$$E_1 = y_1 + \frac{V_1^2}{2g} = y_2 + \frac{V_2^2}{2g} = E_2$$

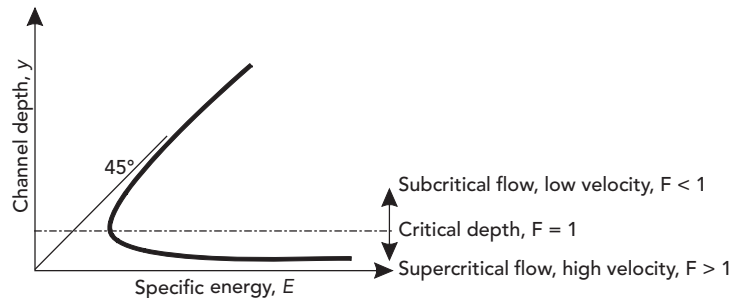


Figure 1.5: Specific Energy.

Recall that the velocity of the section is directly related to the area of flow, and that the area of flow is a function of channel depth. This means that, for a given discharge, the specific energy at each point is solely a function of channel depth and more than one depth may exist with the same specific energy. If the channel depth is plotted against specific energy for a given flow rate, the result is similar to the graph shown in Figure 1.5.

As this figure shows, a depth exists for which the specific energy is at a minimum. This depth is called the **critical** depth. If the velocity is higher than critical velocity (that is, the depth is less than critical depth), the flow is considered **supercritical**. If the velocity is lower than critical velocity (the depth is greater than critical depth), the flow is **subcritical**.

The velocity at critical depth is equal to the **wave celerity**—the speed at which waves will ripple outward from a pebble tossed into the water. A unitless value called the **Froude number**, F , represents the ratio of actual fluid velocity to wave celerity. The Froude number is computed as follows:

$$F = \frac{V}{\sqrt{gD}}$$

where

F = Froude number (unitless)

D = hydraulic depth of the channel, defined as A/T

A = flow area (m^2 , ft^2)

T = top width of flow (m , ft)

V = fluid velocity (m/s , ft/s)

g = gravitational acceleration (m/s^2 , ft/s^2).

By definition, when the flow is at critical depth (that is, the velocity is equal to the wave celerity), the Froude number must be equal to 1. The equation can therefore be rewritten and refactored to form the following equality:

$$\frac{A^3}{T} = \frac{Q^2}{g}$$

where

A = flow area (m^2 , ft^2)

T = top width of flow (m , ft)

Q = channel flow rate (m^3/s , ft^3/s)

g = gravitational acceleration (m/s^2 , ft/s^2).

This equation can now be used to determine the depth for which this equality holds true, which is critical depth. For simple geometric shapes, the solution is relatively easy to determine. However, quite a few iterations may be required to find the solution for an irregularly shaped channel such as a natural streambed. In fact, several valid critical depths may exist for irregular channels.

Example 1.4: Uniform Open-Channel Flow

A concrete trapezoidal channel has a bottom width of 4 m and 45-degree side slopes. If the channel is on a 1% slope and is flowing at a depth of 1 m throughout its length, how much flow is being carried (use Manning's equation)? How much flow would the same channel carry if it were a rectangular channel 4 m wide?

Solution

Because the channel is flowing at the same depth throughout, we can assume that normal depth has been achieved (that is, the friction slope is equal to the channel slope). We will assume a Manning's n of 0.013 for concrete.

From the trapezoidal geometry, we can easily calculate the area and wetted perimeter, and then the hydraulic radius, as follows:

$$A = (4 \text{ m} \times 1 \text{ m}) + 2 \times (0.5 \times 1 \text{ m} \times 1 \text{ m}) = 5.00 \text{ m}^2$$

$$P_w = 4 \text{ m} + 2 \times (1 \text{ m} \times 2^{0.5}) = 6.83 \text{ m}$$

$$R = A/P_w = 5.00 \text{ m}^2/6.83 \text{ m} = 0.73 \text{ m}$$

Manning's equation for velocity can then be solved. The discharge can be computed as

$$V = (1.00/0.013) \times 0.73^{2/3} \times 0.01^{1/2} = 6.25 \text{ m/s}$$

$$Q = V \times A = 6.25 \text{ m/s} \times 5.00 \text{ m}^2 = 31.2 \text{ m}^3/\text{s}$$

To answer the second part of the question, we simply repeat the steps for a rectangular section shape.

$$A = (4 \text{ m} \times 1 \text{ m}) = 4 \text{ m}^2$$

$$P_w = 4 \text{ m} + 2 \times (1 \text{ m}) = 6 \text{ m}$$

$$R = 4 \text{ m}^2/6 \text{ m} = 0.67 \text{ m}$$

$$V = (1.00/0.013) \times 0.67^{2/3} \times 0.01^{1/2} = 5.87 \text{ m/s}$$

$$Q = 5.87 \text{ m/s} \times 4 \text{ m}^2 = 23.5 \text{ m}^3/\text{s}$$

As we would expect, this discharge is less than the discharge of the trapezoidal section.

Example 1.5: Critical Depth

What is the critical depth for a grassy triangular channel with 2H:1V side slopes and a 0.5% slope when the flow is $3.00 \text{ m}^3/\text{s}$? If the channel is actually flowing at a depth of 1.2 m, is the flow critical, subcritical, or supercritical?

Solution

For such a simple geometry, we can quickly create a relationship between the flow area, top width, and depth of flow:

$$T = 4y, A = 0.5 \times T \times y = 0.5 \times 4y \times y = 2y^2 \text{ m}^2$$

Inserting these values into the previous equation for critical depth, we can algebraically solve for the channel depth:

$$(2y^2)^3/4y = Q^2/g$$

$$8y^6/4y = Q^2/g$$

$$2y^5 = Q^2/g$$

$$y^5 = Q^2/2g$$

$$y = (Q^2/2g)^{1/5} = [(3.00 \text{ m}^3/\text{s})^2/(2 \times 9.8 \text{ m/s}^2)]^{0.2} = (0.46 \text{ m}^5)^{0.2} = 0.86 \text{ m}$$

The critical depth for this section is 0.86 m. The actual flow depth of 1.2 m is greater than critical depth, so the flow is subcritical.

1.6 Orifices and Weirs

The energy equation serves as the foundation for calculating the flow through and over hydraulic structures based on the size of the opening associated with the structure and the difference in energy on either side of it. The flow exiting the structure can be calculated by solving the energy equation for velocity, V^2 , and multiplying the resulting formula by the flow area and a coefficient to account for different hydraulic and physical variables. These variables include head loss, the shape and nature of the opening, the contraction of the flow after it leaves the structure, and countless indefinable variables that are difficult to measure but produce quantifiable effects.

Two common devices for which equations are derived in this manner are weirs and orifices. They are important not only because of their widespread usage in the industry, but also because the equations that describe them serve as the foundation for mathematical descriptions of more complicated hydraulic devices such as drainage inlets and culverts.

Orifices

Orifices are regularly shaped, submerged openings through which flow is propelled by the difference in energy between the upstream and downstream sides of the opening. The stream of flow expelled from the orifice is called the **jet**. When the jet exits the orifice, adverse velocity components cause it to contract to a point after which the flow area remains relatively constant and the flow lines become parallel (see Figure 1.6). This point is called the **vena contracta**.

Orifices and the orifice equations have the following applications:

- regulating the flow out of detention ponds,
- regulating the flow through channels in the form of radial and sluice gates,

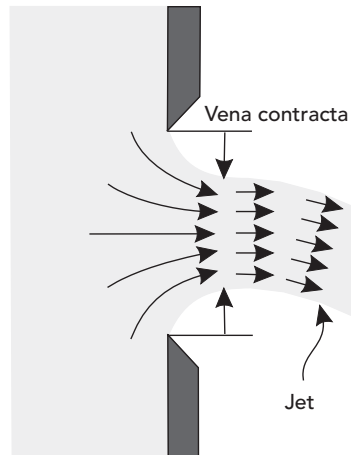


Figure 1.6: Cross-Sectional View of Typical Orifice Flow.

- approximating the interception capacity of submerged drainage inlets in sag (see Chapter 3),
- approximating the flow allowed through a submerged culvert operating under inlet control (see Chapter 4),
- measuring flow.

Example 1.6: The Orifice Equation

For the structure in Figure 1.7, derive the orifice equation for an orifice of area A .

Solution

First, start with the energy equation from Section 1.2:

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} + H_G = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} + H_L$$

List known variables and assumptions:

- The datum is at the centerline/centroid of the orifice.
- $p_1/\gamma = H$.
- Point 2 occurs at the vena contracta.
- Elevation heads, z_1 and z_2 , are equivalent.
- The velocity in the tank at point 1 is negligible.
- The jet is open to the air, so the pressure at point 2 is 0.
- There is no head gain.

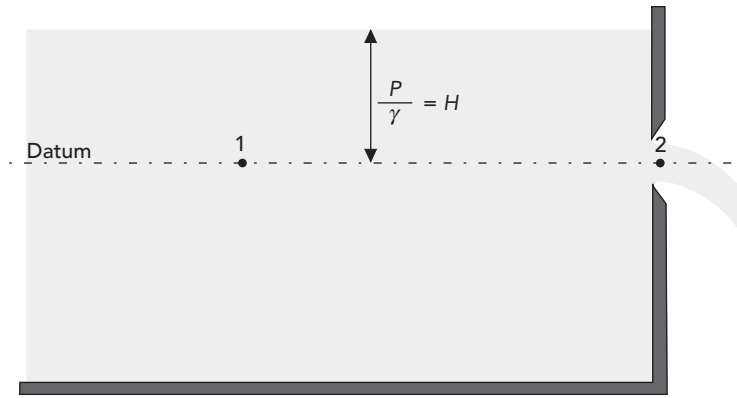


Figure 1.7: Orifice Example.

Taking these known variables and assumptions into account and solving for V_2 , the energy equation becomes:

$$V_2 = \sqrt{2g(H - H_L)}$$

To find the flow exiting the structure at point 2, multiply both sides of the equation by the orifice area, A .

$$AV_2 = Q = A\sqrt{2g(H - H_L)}$$

where Q = discharge (m^3/s , ft^3/s).

The point of discharge is the vena contracta, where the flow area is usually contracted from the original orifice area. Also, computations can be simplified by eliminating the head loss term, H_L . Both of these variables are accounted for by applying an orifice coefficient, C , to the right side of the equation. The final form of the orifice equation becomes:

$$Q = CA\sqrt{2gH}$$

where C = orifice coefficient.

When dealing with storm sewer design, the orifice coefficient is generally about 0.6. For more in-depth information on orifice coefficients for different situations, see Brater and King's *Handbook of Hydraulics* (1996).

Weirs

Weirs are notches or gaps over which fluid flows. The lowest point of structure surface or edge over which water flows is called the **crest**, whereas the stream of

water that exits over the weir is called the *nappe* (see Figure 1.8). Depending on the weir design, flow may contract as it exits over the top of the weir, and, as with orifices, the point of maximum contraction is called the vena contracta.

This contraction can be counteracted or suppressed by designing the weir such that its shape conforms to the shape of the channel. This type of weir is called a *suppressed weir*. With a *contracted weir*, the crest and nappe vary from the channel to such a degree that a significant contraction of flow area does occur.

In addition to suppressed and contracted weir types, weirs are also distinguished as either *sharp-crested* or *broad-crested*. A sharp-crested weir has a sharp upstream edge formed so that the nappe flows clear of the crest. Broad-crested weirs have crests that extend horizontally in the direction of flow far enough to support the nappe and fully develop hydrostatic pressures for at least a short distance.

Weirs can also be distinguished by their shapes. The most common shapes are shown in Figure 1.9. The effects of weir shape and other factors previously mentioned are accounted for with modifications to the weir equation (derived in Example 1.7), such as adjustments the weir coefficient. Table 1.3 contains information on coefficients for V-notch weirs.

Weirs have the following applications:

- serving as emergency spillways for regulating high-return event flows overtopping dams and detention ponds,
- regulating the flow in channels,
- measuring flow,
- approximating the flow over roadways acting as broad-crested weirs when flow exceeds a culvert's capacity (see Chapter 4),
- approximating the interception capacity of unsubmerged drainage inlets in swales (see Chapter 3),
- approximating the flow allowed through an unsubmerged culvert operating under inlet control (see Chapter 4).

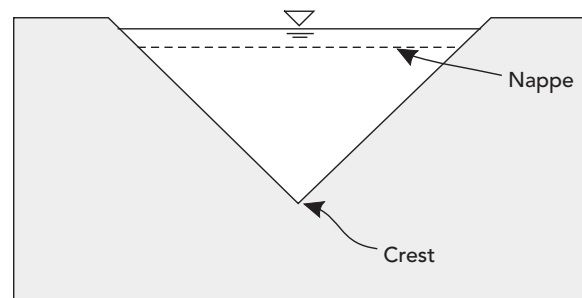
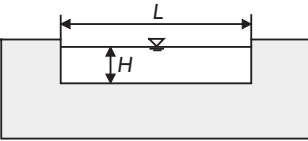
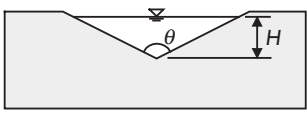
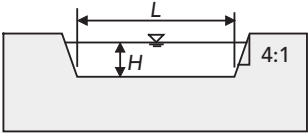
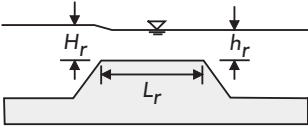


Figure 1.8: Front View of Common Weir.

	Weir type	Figure	Equation	Coefficients
Sharp crested	Rectangular		Contracted $Q = C(L - 0.1iH) H^{3/2}$ Suppressed $Q = CLH^{3/2}$ i = number of iterations	Metric $C = 1.84$ English $C = 3.367$
	V-notch		$Q = C \left(\frac{8}{15} \right) \sqrt{2g} \tan \theta \left(\frac{H}{2} \right)^{3/2}$	C varies between 0.611 and 0.570 depending on H and Q^*
	Cipolletti		Metric $Q = CLH^{3/2}$ English $Q = CLH^{3/2}$	Metric $C = 1.86$ English $C = 3.367$
Non sharp-crested	Broad (side view)		$Q = C_d L H_r^{3/2}$	C_d is a function of H_t , h_t and L_t ranging between 1.25 and 3.1*

*Refer to FlowMaster help documentation for more information.

Figure 1.9: Front View of Common Weir.

Table 1.3: V-Notch Weir Coefficients of Discharge—English Units

Head (ft)	Weir Angle, θ (°)					
	22.5	30	45	60	90	120
0.5	0.611	0.605	0.569	0.590	0.584	0.581
1.0	0.593	0.590	0.583	0.580	0.576	0.575
1.5	0.586	0.583	0.578	0.575	0.572	0.572
2.0	0.583	0.580	0.576	0.573	0.571	0.571
2.5	0.580	0.578	0.574	0.572	0.570	0.570
3.0	0.579	0.577	0.574	0.571	0.570	0.570

Derived from: Van Havern, Bruce P. *Water Resources Measurements*, American Water Works Association, 1986

Example 1.7: The Weir Equation

Derive the weir equation for the rectangular weir with a crest of length L and head H , which discharges from a free outfall as shown in Figure 1.10.

Solution

Begin with the energy equation:

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} + H_G = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} + H_L$$

List the known variables and assumptions:

- The datum is at the crest of the weir.
- $p_1/\gamma = H$.
- Point 2 occurs at the vena contracta.
- The elevation heads, z_1 and z_2 , are equivalent.
- Assume the velocity in the tank is negligible.
- The exiting stream pressure at point 2 is 0.
- There is no head gain.

After applying the known variables and assumptions and solving for V_2 , the energy equation becomes:

$$V_2 = \sqrt{2g(H - H_L)}$$

To find the flow, multiply both sides by the flow area, which in this case is the length of the weir, L , multiplied by the height of the head, H .

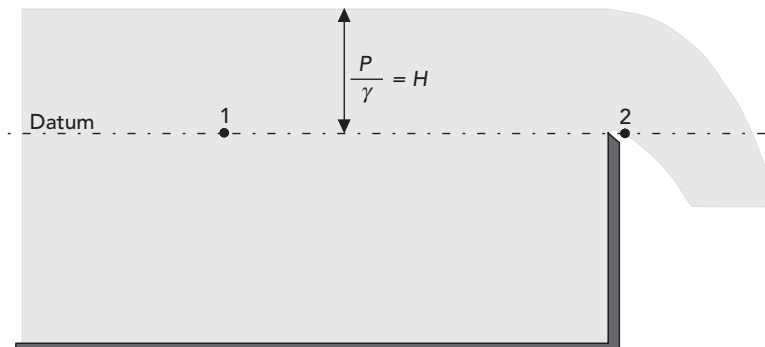


Figure 1.10: Weir Example.

$$AV_2 = Q = LH\sqrt{2g(H - H_L)}$$

where L = length of weir (m, ft).

To account for head loss, contraction, and other variables, a weir coefficient, C , is applied. Unlike the orifice coefficient, the weir coefficient generally takes into account the constant $2g$. Finally, the weir equation becomes:

$$Q = CLH^{3/2}$$

where C = weir coefficient ($\text{m}^{1/2}/\text{s}$, $\text{ft}^{1/2}/\text{s}$)

Unlike the orifice coefficient, the weir coefficient is not unitless. Care has to be taken that the correct coefficient is applied when using a specific unit system. Typical coefficient values for different weir shapes, as well as variations in the equation, can be seen in the previously shown in Figure 1.9.

1.7 Computer Applications

It is very important for students (and practicing engineers) to fully understand the methodologies behind hydraulic computations. Once these concepts are understood, the solution process can become repetitive and tedious—the type of procedure that is well suited to computer analysis.

There are several advantages to using computerized solutions for common hydraulic problems:

- The amount of time to perform an analysis can be greatly reduced.
- Computer solutions can be more detailed than hand calculations. Performing a solution manually often requires many simplifying assumptions.
- The solution process may be less error-prone. Unit conversion and the rewriting of equations to solve for any variable are just two examples of mistakes frequently introduced with hand calculations. A well-tested computer program helps to avoid these algebraic and numeric errors.
- The solution is easily documented and reproducible.
- Because of the speed and accuracy of a computer model, more comparisons and design trials can be performed. The result is the exploration of more design options, which eventually leads to better, more efficient designs.

In order to prevent an “overload” of data, this chapter deals primarily with steady-state computations. After all, an introduction to hydraulic calculations is tricky enough without throwing in the added complexity of a constantly changing system.

The assumption that a system is under steady-state conditions is often perfectly acceptable. Minor changes that occur over time or irregularities in a channel

cross section are frequently negligible, and a more detailed analysis may not be the most efficient or effective use of time and resources.

There are circumstances when an engineer may be called upon to provide a more detailed analysis, including unsteady flow computations. For a storm sewer, the flows may rise and fall over time as a storm builds and subsides. For water distribution piping, a pressure wave may travel through the system when a valve is closed abruptly (the same “water-hammer” effect can probably be heard in your house if you close a faucet quickly).

As an engineer, it is important to understand the purpose of an analysis; otherwise, appropriate methods and tools to meet that purpose cannot be selected.

1.8 FlowMaster and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software.

What Does FlowMaster Do?

FlowMaster is an easy-to-use program that helps civil engineers with the hydraulic design and analysis of pipes, gutters, inlets, ditches, open channels, weirs, and orifices. FlowMaster computes flows and pressures in conduits and channels using common head loss equations such as Darcy–Weisbach, Manning’s, Kutter’s, and Hazen–Williams. The program’s flexibility allows the user to choose an unknown variable and automatically compute the solution after entering known parameters. FlowMaster also calculates rating tables and plots curves and cross sections. You can view the output on the screen, copy it to the Windows clipboard, save it to a file, or print it on any standard printer. FlowMaster data can also be viewed and edited using tabular reports called FlexTables.

FlowMaster enables you to create an unlimited number of worksheets to analyze uniform pressure pipe or open-channel sections, including irregular sections (such as natural streams or odd-shaped man-made sections). FlowMaster does not work with networked systems such as a storm sewer network or a pressure pipe network. For these types of analyses, StormCAD, WaterGEMS, or SewerCAD should be used instead.

The theory and background used by FlowMaster have been reviewed in this chapter and can be accessed via the FlowMaster on-line help system. General information about installing and running Bentley Systems software can be found in Appendix A.

FlowMaster replaces solutions such as nomographs, spreadsheets, and BASIC programs. Because FlowMaster gives you immediate results, you can quickly generate output for a large number of situations. For example, you can use FlowMaster to:

- analyze various hydraulic designs,
- evaluate different kinds of flow elements,
- generate professional-looking reports for clients and review agencies.

1.9 Problems

Solve the following problems using the FlowMaster computer program. Unless stated otherwise, assume water is at 20°C.

1. The cross section of a rough, rectangular, concrete ($k = 0.2 \times 10^{-2}$ ft) channel measures 6×6 ft. The channel slope is 0.02 ft/ft. Using the Darcy–Weisbach friction method, determine the maximum allowable flow rate through the channel to maintain one foot of freeboard (freeboard is the vertical distance from the water surface to the overtopping level of the channel). For these conditions, find the following characteristics (note that FlowMaster may not directly report all of these):
 - (a) Flow area
 - (b) Wetted perimeter
 - (c) Hydraulic radius
 - (d) Velocity
 - (e) Froude number
2. A 450-mm circular concrete ($n = 0.013$) pipe constructed on a 0.6% slope carries $0.1 \text{ m}^3/\text{s}$.
 - (a) Using Manning's equation and normal depth assumptions, what are the depth and velocity of flow?
 - (b) What would the velocity and depth be if the pipe were constructed of corrugated metal ($n = 0.024$) instead of concrete?
3. A trapezoidal channel carries $2.55 \text{ m}^3/\text{s}$ at a depth of 0.52 m. The channel has a bottom width of 5 m, a slope of 1.00%, and 2H:1V side slopes.
 - (a) What is the appropriate Manning's roughness coefficient?
 - (b) How deep would the water be if the channel carried $5 \text{ m}^3/\text{s}$?
4. Use Manning's equation to analyze an existing brick-in-mortar ($n = 0.015$) triangular channel with 3H:1V side slopes and a 0.05 longitudinal slope. The channel is intended to carry 7 cfs during a storm event.
 - (a) If the maximum depth in the channel is 6 in., is the existing design acceptable?

- (b) What would happen if the channel were replaced by a concrete ($n = 0.013$) channel with the same geometry?
5. A pipe manufacturer reports that it can achieve Manning's roughness values of 0.011 for its concrete pipes, which is lower than the 0.013 reported by its competitors. Using Kutter's equation, determine the difference in flow for a 310-mm circular pipe with a slope of 2.5% flowing at one-half of the full depth.
6. A grass drainage swale is trapezoidal, with a bottom width of 6 ft and 2H:1V side slopes. Using the friction method you feel is appropriate, answer the following questions:
 - (a) What is the discharge in the swale if the depth of flow is 1 ft and the channel slope is 0.005 ft/ft?
 - (b) What would the discharge be with a slope of 0.010 ft/ft?
7. A paved highway drainage channel has the geometry shown in the following figure. The maximum allowable flow depth is 0.75 ft (to prevent the flow from encroaching on traffic), and the Manning's n -value is 0.018 for the type of pavement used.

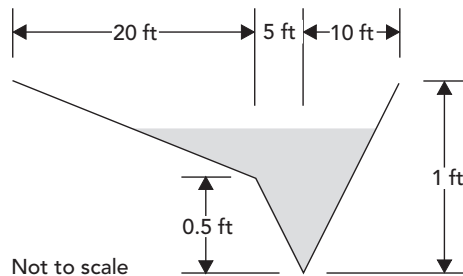


Figure for Problem 7

- (a) What is the capacity of the channel given a 2% longitudinal slope?
 - (b) Create a rating curve to demonstrate how the capacity varies as the channel slope varies from 0.5% to 5%. Choose an increment that will generate a reasonably smooth curve.
8. Using the Hazen–Williams equation, determine the minimum diameter of a new cast iron pipe ($C = 130$) for the following conditions: the upstream end is 51.8 m higher than the downstream end, which is 2.25 km away. The upstream pressure is 500 kPa, and the desired downstream pressure and flow rate are 420 kPa and 11,000 l/min, respectively. What is the minimum diameter needed? Assume pipes are available in 50-mm increments.
9. 2,000 gal of water per minute flow through a level, 320-yard-long, 8-in.-diameter cast iron pipe ($C = 130$, $k = 2.5908 \times 10^{-4}$ m) to a large industrial site. If the pressure at the upstream end of the pipe is 64 psi,

Basic Hydraulic Principles

Problems

what will the pressure be at the industry? Is there a significant difference between the solutions produced by the Hazen–Williams method and the Darcy–Weisbach method?

10. Develop a performance curve for the pipe in Problem 9 that shows the available flow to the industry with residual pressures ranging from 20 to 80 psi (assume the source can maintain 64 psi regardless of flow rate). Create similar curves for 10- and 12-in. diameter pipes and compare the differences in flow.
11. Using the Darcy–Weisbach equation, find the minimum size of circular corrugated metal storm drain ($k = 1.0 \times 10^{-3}$ ft) that will carry 1.5 cfs with a maximum depth of 6 in. The drain carries water down a hill 3-ft high to a pond with a free outfall 75 ft away. What pipe size should be used? Assume pipes are available in 3-in. increments. What would the maximum capacity of this pipe be? What would the capacity of the pipe be when it is flowing full?
12. A channel with the cross section shown in the following figure has a Manning's coefficient of 0.040 from stations 0 to 3 and 0.054 from stations 3 to 8. The flow through the channel is $13 \text{ m}^3/\text{s}$, and the water surface is 1.7-m high. Find the following:
 - (a) Weighted Manning's coefficient
 - (b) Slope of the channel
 - (c) Top width
 - (d) Wetted perimeter
 - (e) Flow regime (supercritical or subcritical)

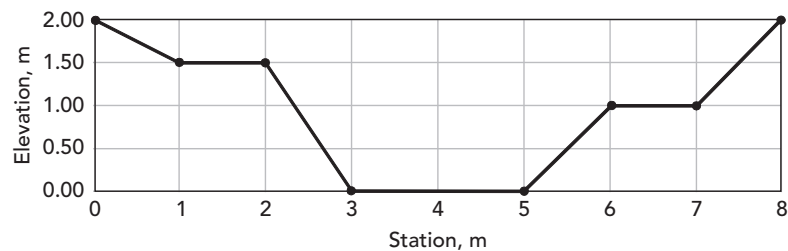


Figure for Problems 12 and 13

13. A stream with the cross section shown in the previous figure has a flow rate of $5 \text{ m}^3/\text{s}$. The stream has a longitudinal slope of 0.002 m/m and a natural stony bottom ($n = 0.050$, stations 0–8).
 - (a) Using Manning's equation, what is the water surface elevation of the stream?
 - (b) What is the maximum capacity of the channel?

(c) How would the capacity of the channel be affected if you were to pave the center of the channel ($n = 0.013$) between stations 3 and 5?

14. A rectangular concrete channel with a width of 1 m and a height of 0.5 m is on a slope of 0.008 m/m. Design a concrete circular channel for which the depth is half of the diameter and the flow area is the same as that of the rectangular channel. Which channel is more efficient and by how much?
15. A weir is placed in a rectangular channel to measure the flow. The discharge from the rectangular channel enters a trapezoidal channel with a stony bottom. The trapezoidal channel is 0.50-m wide at the base with 2:1 ($H:V$) equal side slopes. The weir is a sharp-crested, V-notch weir with a crest 0.43 m above the channel bottom, a weir coefficient of 0.58, and a notch angle of 1.57 radians. The height of the water above the weir is 0.70 m, and the depth of water in the trapezoidal channel is measured to be 0.40 m.

What is the flow rate? What is the slope of the trapezoidal channel (using Manning's formula)? If the discharge is increased until the elevation of the water surface in the trapezoidal channel reaches 0.61 m, what will the headwater elevation be at the weir?

16. The outlet structure on a pond is used to regulate the flow out of the pond for different storm events. An outlet structure must be designed to discharge 2.20 m³/min when the water surface elevation in the pond reaches 1.52 m, and 6.29 m³/min when the water surface elevation reaches 2.60 m. The outlet structure will be a circular orifice and a sharp-crested rectangular weir combination, with the centroid of the orifice at an elevation of 0.90 m and the weir crest at an elevation of 2.50 m. Both will discharge to free outfall conditions.

Assume an orifice coefficient of 0.6. Find the orifice diameter needed to supply the correct discharge when the water surface reaches the first specified elevation. What will the discharge from the orifice be when the water surface reaches the second specified elevation? Find the width of the weir needed to supply the extra discharge necessary to meet the requirement. Use Manning's formula where necessary.

17. An approximately trapezoidal, clean, natural stream carries the discharge from a pond down a 0.001 slope. The maximum depth in the channel is 0.5 m. The channel has equal side slopes of 3.0 ($H:V$) and a bottom width of 1.0 m. The pond discharges water through a circular orifice into the channel. The centroid of the orifice is located 1.0 m above the bottom of the channel.

Assume an orifice coefficient of 0.6. Design the orifice to discharge the maximum flow rate possible without exceeding the maximum allowed depth in the channel when the water surface in the pond reaches 4.6 m above the channel bottom. Use Manning's formula when necessary.

2

Basic Hydrology

Hydrology is the study of water on and below the Earth's surface and in the atmosphere. The discussion in this chapter is limited to rainfall and the resulting surface runoff. Data on surface runoff flow rates and volumes provides the loads used to design and evaluate storm sewers, culverts, ditches, and detention ponds. Several of these structures are covered in more detail in other chapters.

2.1 Rainfall

Rainfall data is fundamental for determining the amount of stormwater generated by a particular storm event. This section describes the properties of rainfall used in modeling.

Basic Rainfall Characteristics

For the design engineer, the most important characteristics of rainfall are

- the *depth*, or *volume*, of rainfall during a specified time interval (or its average *intensity* over that time interval),
- the *duration* of the rainfall,
- the *area* over which the rainfall occurs,
- the average *recurrence interval* of a rainfall amount, and
- the *temporal* and *spatial distributions* of rainfall within the storm.

Return Period and Frequency

The probability that a storm event of a certain magnitude will occur in any given year is expressed in terms of event frequency and return period.

The *return period*, or the recurrence interval, represents, on average, the length of time expected to elapse between rainfall events of equal or greater

magnitude. Although a recurrence interval is expressed in years, it is actually based on probability of occurrence.

For example, a 5-year return period represents a storm event that is expected to occur once every 5 years on average. This does not mean that two storm events of that size will not occur in the same year, nor does it mean that the next storm event of that size will not occur for another 5 years. It just means that the average will be once every 5 years (that is, there is a 20% chance of occurrence in any given year).

The *frequency*, or exceedance probability, is a measure of how often a specific rainfall event will be equaled or exceeded and is simply the inverse of the return period.

Example 2.1: Computing a Recurrence Interval

What is the recurrence interval for a storm event that has a 20% probability of being equaled or exceeded within any given year? For one that has a 2% probability of exceedance?

Solution

For a storm event with a 20% probability of being equal to or exceeded in a given year, the recurrence interval is computed as

$$1/0.20 = 5 \text{ years}$$

For an event with a 2% probability, the recurrence interval is

$$1/0.02 = 50 \text{ years}$$

Types of Rainfall Data

Historic rainfall data is compiled and analyzed to predict storm characteristics. Rainfall data is available from a variety of sources, including governmental organizations and agencies. The data can be presented in various formats, including intensity–duration–frequency (IDF) curves, cumulative rainfall depths, and rainfall hyetographs. In infrastructure design, a synthetic rainfall distribution is often applied to the total rainfall depth for a storm of given duration and recurrence frequency. In the United States, the most frequently applied synthetic rainfall distributions are four 24-h distributions developed by the Natural Resources Conservation Service, U.S. Department of Agriculture.

Intensity–Duration–Frequency Data

For a selected storm duration, a rainfall intensity exists that corresponds to a given exceedance probability or recurrence interval. A rainfall *IDF curve* illustrates the average rainfall intensities corresponding to a particular storm

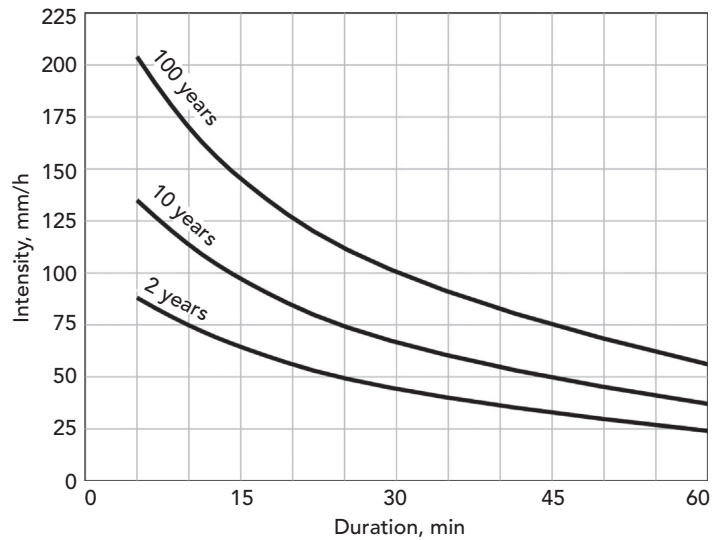


Figure 2.1: Example set of IDF curves.

Table 2.1: Example IDF Table

Storm	Rainfall Intensities (mm/h)		
	Return Period		
	2 Years	10 Years	100 Years
Durations			
5 min	88	135	204
10 min	75	114	168
15 min	65	97	142
30 min	44	66	100
60 min	24	37	56

recurrence interval for various storm durations (see Figure 2.1). These curves are the result of the statistical analysis of rainfall data for a particular area.

Given the information on the graph shown in Figure 2.1, you can determine that the average 1-h rainfall intensity expected to be equaled or exceeded, on average, once every 100 years is 56.0 mm/h.

Although graphical rainfall curves are acceptable for hand calculations, they are not well-suited to computer analyses. Data is therefore input into hydrologic software either as equations or in a tabular format. Creating a rainfall table from a set of IDF curves is a simple matter of manually picking values from the curves. For example, Table 2.1 can be created from the data presented in the IDF curves of Figure 2.1.

Computer programs commonly access IDF data in the form of an equation. Several forms have been developed to analytically describe the graphical IDF relationships. The most common forms of these equations are:

$$V = \frac{Q}{A} \quad R = \frac{A}{P_w} \quad i = a + b(\ln D) + c(\ln D)^2 + d(\ln D)^3$$

where

i = intensity of rainfall (mm/h, in/h);

D = rainfall duration (min or h);

R_p = return period (years); and

a, b, c, d, m , and n are coefficients.

When applying IDF data to system design, you must use data developed for the specific geographic location where the system is to be constructed. A single set of IDF curves can normally be used for areas as large as a city or small county. Many drainage jurisdictions and agencies such as weather bureaus can provide the engineer with IDF data recommended for their particular geographical location. Engineers should understand when and by whom the IDF curves were created, as more recently updated resources may be available.

In the United States, such information can be found in several National Weather Service publications. For example, the NWS publication TP 40 (Hersfield, 1961) presents maps showing precipitation depths over the United States for storm durations from 30 min to 24 h and for recurrence intervals from 1 to 100 years. TP 40 was partially superseded by a later publication known as HYDRO-35 for the central and eastern United States (Frederick et al., 1977), and by the NOAA Atlas 2 for the 11 coterminous western states (Miller et al., 1973). Updated atlases for the Midwestern United States (Huff and Angle, 1992) and the northeastern United States and southeastern Canada (McKay and Wilkes, 1995) have also been published.

Temporal Distributions and Hyetographs for Design Storms

Some types of hydrologic analyses require the distribution of precipitation over the duration of the storm. A **temporal rainfall distribution**, such as the one shown in Figure 2.2, shows the cumulative progression of rainfall depth throughout a storm. The y -axis is represented by a simple rain gauge that fills over the period represented on the x -axis. **Total depth** is simply the final depth in the gauge. The average intensity (i) during any time segment is represented by the slope of the rainfall curve during that interval. The steeper the slope, the greater the average intensity is for a given segment.

The temporal distribution shown in Figure 2.2 can also be represented using a bar graph that shows how much of the total rainfall occurs within each time interval during the course of an event. A graph of this nature is called a **rainfall hyetograph**. Hyetographs can be displayed in terms of incremental rainfall depth measured within each time interval as shown in Figure 2.3, or as the average intensity calculated for each interval by dividing incremental depth by the time interval.

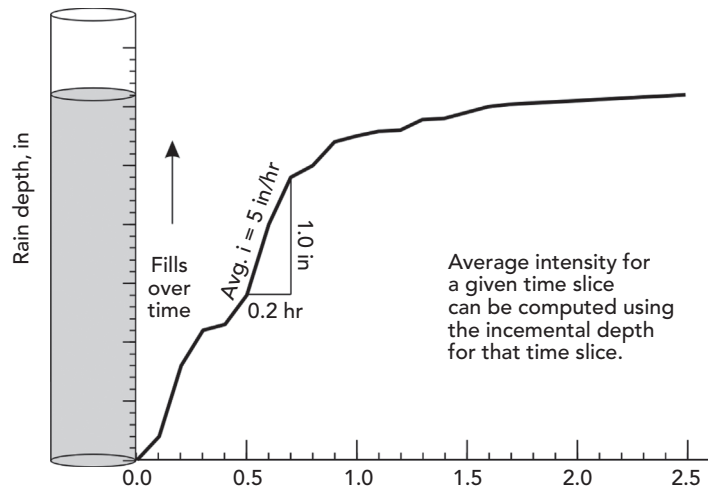


Figure 2.2: Temporal distribution of rainfall.

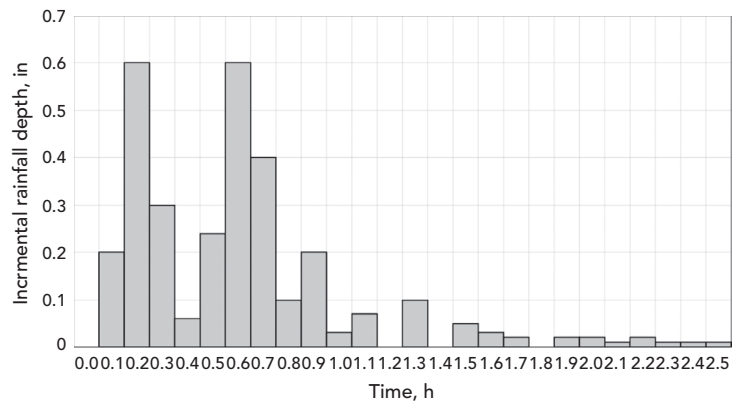


Figure 2.3: A hyetograph of incremental rainfall depth versus time.

The design of stormwater management facilities typically requires a complete rainfall hyetograph. For such situations, engineers commonly use *synthetic* temporal distributions of rainfall, which are essentially systematic and reproducible methods for varying the rainfall intensity throughout a design event.

The selected length Δt of the time increment between the data points used to construct a temporal rainfall distribution depends on the size (area) and other characteristics of the drainage basin. As a rule of thumb, the time increment should be no larger than about one-fourth to one-fifth of the basin lag time (t_l), or about one-sixth of the *time of concentration* (t_c) of the basin. In any case, the smallest time increment for which rainfall data is generally available is

about 5 min. In small urban drainage basins where it is often necessary to use time increments as small as 1 or 2 min, the data must be extrapolated.

In addition to selecting an appropriate Δt , the engineer must select the total duration to be used when developing a design storm hyetograph. In many cases, the storm duration will be specified by the review agency having jurisdiction over the area in which a stormwater conveyance facility will be built; this approach promotes consistency from one design to another.

Many methods have been proposed for distributing a total rainfall depth throughout a storm to develop a design storm hyetograph. The NRCS developed one of the more commonly used methods in the United States (SCS, 1986). With this method, Table 2.2 is used to find fractions of the total

Table 2.2: SCS Dimensionless Storm Distributions (SCS, 1986)

t (h)	Type I	Type IA	Type II	Type III
0	0.000	0.000	0.000	0.000
1	0.017	0.020	0.011	0.010
2	0.035	0.050	0.022	0.020
3	0.055	0.082	0.035	0.031
4	0.076	0.116	0.048	0.043
5	0.099	0.156	0.063	0.057
6	0.126	0.206	0.080	0.072
7	0.156	0.268	0.098	0.091
8	0.194	0.425	0.120	0.114
9	0.254	0.520	0.147	0.146
10	0.515	0.577	0.181	0.189
11	0.624	0.624	0.235	0.250
12	0.682	0.664	0.663	0.500
13	0.728	0.701	0.772	0.750
14	0.766	0.736	0.820	0.811
15	0.799	0.769	0.854	0.854
16	0.830	0.800	0.880	0.886
17	0.857	0.830	0.902	0.910
18	0.882	0.858	0.921	0.928
19	0.905	0.884	0.937	0.943
20	0.926	0.908	0.952	0.957
21	0.946	0.932	0.965	0.969
22	0.965	0.956	0.978	0.981
23	0.983	0.978	0.989	0.991
24	1.000	1.000	1.000	1.000

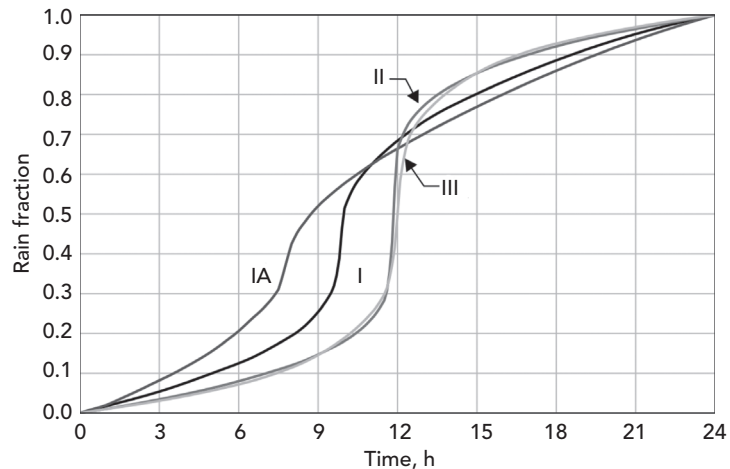


Figure 2.4: Graphical representation of NRCS (SCS) rainfall distributions.

accumulated rainfall depth for storms with 24-h durations. (Figure 2.4 depicts Table 2.2 graphically.) The storms are classified into various types, with each type being recommended for use in a certain U.S. geographical region, as shown in Figure 2.5. If necessary, interpolation may be used to obtain values not shown in Table 2.2. Nonlinear interpolation methods are recommended for this purpose.

Example 2.2: Developing a Design Storm Hyetograph from SCS Distributions

Develop a design storm hyetograph for a 50-year, 24-h storm in Boston, Massachusetts. Assume that $\Delta t = 0.1$ h is a reasonable choice for the drainage basin to which the design storm will be applied.

Solution

Figure 2.5 illustrates that a Type III storm distribution is a reasonable choice for Boston. From TP 40, the total depth of a 50-year, 24-h storm in Boston is estimated to be 6.0 in. Table 2.3 illustrates the calculation of the storm hyetograph.

The first column of the table is the time, in hours, since the beginning of the storm, and is tabulated in 1-h increments for the total storm duration of 24 h. (In actuality, the Δt used in the calculations would be 0.1 h; the 1-h increment is used here for brevity.) The second column is the fraction of the total storm depth that has accumulated at each time during the storm. These values are obtained by interpolation from Table 2.2 for the Type III storm distribution. The third column contains the cumulative rainfall depths for each time during the storm and is obtained by multiplying each fraction in the second column by the total storm depth of 6.0 in. The fourth column contains the incremental

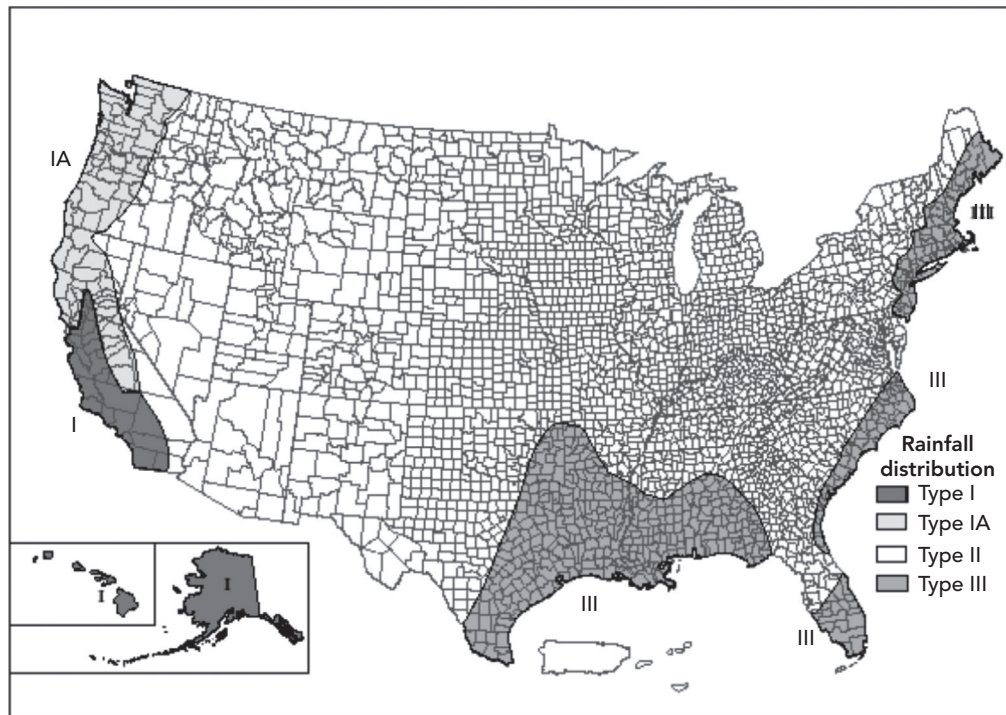


Figure 2.5: Coverage of NRCS (SCS) rainfall distributions (Adapted from SCS, 1986).

Table 2.3: 50-Year, 24-H Storm Hyetograph for Boston, Massachusetts

t (h)	Fraction	Cum. P (in)	Incr. P (in)
0	0.000	0.000	
1	0.010	0.060	0.060
2	0.020	0.120	0.060
3	0.031	0.186	0.066
4	0.043	0.258	0.072
5	0.057	0.342	0.084
6	0.072	0.432	0.090
7	0.091	0.546	0.114
8	0.114	0.684	0.138
9	0.146	0.876	0.192
10	0.189	1.134	0.258
11	0.250	1.500	0.366
12	0.500	3.000	1.500

Table 2.3: (Continued)

13	0.750	4.500	1.500
14	0.811	4.866	0.366
15	0.854	5.124	0.258
16	0.886	5.316	0.192
17	0.910	5.460	0.144
18	0.928	5.568	0.108
19	0.943	5.658	0.090
20	0.957	5.742	0.084
21	0.969	5.814	0.072
22	0.981	5.886	0.072
23	0.991	5.946	0.060
24	1.000	6.000	0.054

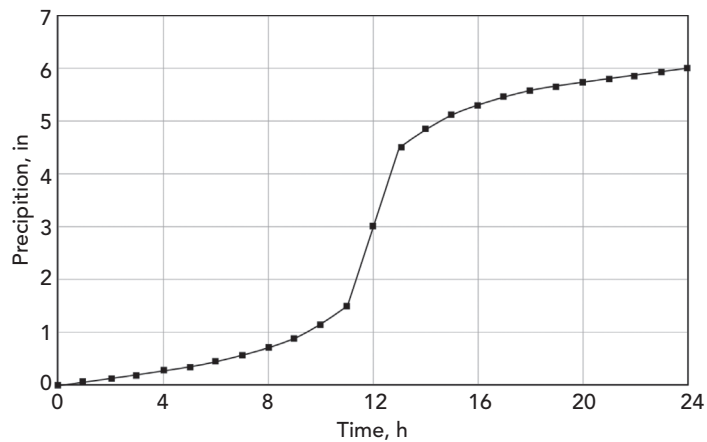


Figure 2.6: Graph of derived design storm cumulative precipitation.

depths of rainfall within each time interval during the storm; these values are computed as the difference between the current and preceding values in the third column.

The resulting graph of cumulative precipitation is shown in Figure 2.6, and the hyetograph is shown in Figure 2.7. The height of each bar on the hyetograph is the average rainfall intensity during that time interval, and the area of each bar is the incremental rainfall depth during that time interval. Because the time increment is 1 h (1-h increment is shown for simplicity; actual Δt is 0.1 h), the value for the height of the bar (in units of in./h) is equal to the incremental depth for that time increment (in in).

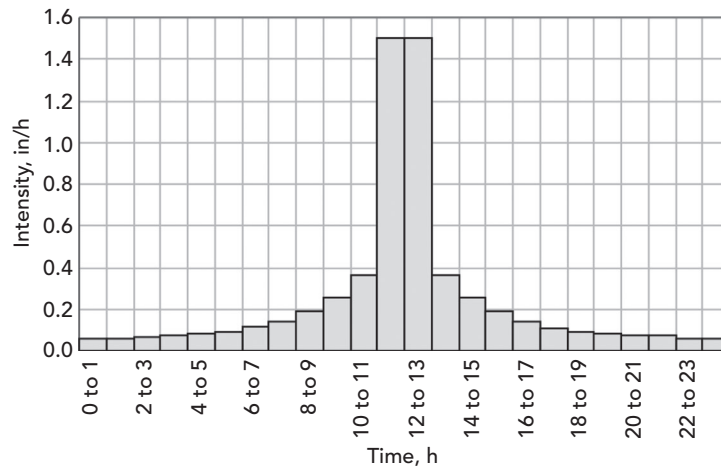


Figure 2.7: Derived design storm hyetograph.

2.2 Rainfall Abstractions and Runoff Volume

Only a portion of the total rainfall occurring over a drainage basin contributes to surface runoff and stream flow. In fact, a simple comparison of rainfall and runoff records for most locations in the United States shows that the equivalent depth of runoff is typically about 30%–50% of the precipitation depth.

To obtain the loading information necessary to design and model structures such as storm sewers, culverts, and detention facilities, it is necessary to calculate the runoff volumes and/or flow rates resulting from the storm events of interest. This section presents three techniques for computing total runoff volume: (1) the *Horton equation*, (2) the *runoff coefficient*, and (3) the *NRCS curve number equation*.

Watershed Area

A *watershed* is an area that drains to a single point of discharge. The first step in computing runoff at a point of interest (such as a culvert, inlet, or property outfall) is to delineate its watershed on a contour map. Because water flows downhill, delineating a watershed is simply a matter of identifying an outfall and locating the watershed boundary such that any rain that falls within the boundary will be directed toward that point of discharge. The delineated area is then measured using a planimeter, or graphical or computer-aided methods. Because river and stream systems collect water, a watershed may have any number of subwatersheds within it. The focus of the analysis and the determination of how large a watershed must be to be analyzed separately depend on the scope and purpose of the project at hand. Figure 2.8 shows the collection channels for two typical natural watersheds.

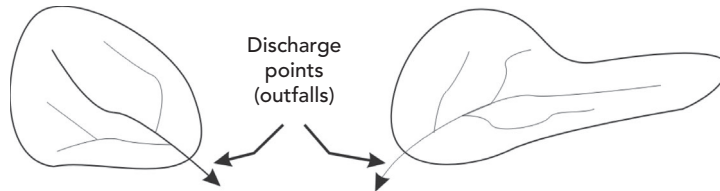


Figure 2.8: Typical natural watersheds with collection channels.

After the watershed is delineated and measured, the modeler can proceed to the next step of computing rainfall abstractions and runoff depth. To determine the total runoff volume, runoff depth is multiplied by the watershed area.

Rainfall Abstractions

Rainfall that does not contribute to direct surface runoff may be intercepted by vegetation, infiltrated into the ground surface, retained as depression storage in puddles and small irregularities in the land surface, or returned to the atmosphere through transpiration and evaporation. Collectively, these **losses** of rainfall are called **abstractions**. The rainfall that remains after abstractions have occurred comprises the surface runoff and is called **effective precipitation**.

The four basic types of abstractions are interception, depression storage, infiltration, and evaporation. Evaporation, however, is not typically considered in modeling stormwater conveyances.

Interception

Interception refers to the capture of rainfall on the leaves and stems of vegetation before it reaches the ground surface. Water intercepted by vegetation is returned to the atmosphere by evaporation during dry-weather periods.

Depression Storage

Excess water begins to pond on the land surface when the rainfall intensity exceeds the infiltration capacity of a soil during a storm event. The ponded water fills small depressions and irregularities in the ground surface, and additional water is held on the surface through the phenomenon of surface tension. The water held in depressions is called **depression storage**, and it either evaporates during dry-weather periods or infiltrates into the soil. Noninfiltrated rainfall that remains after surface depressions and irregularities have been filled contributes to surface runoff.

Infiltration

When rainfall occurs on a **pervious** surface, some of the rainwater infiltrates into the ground. The infiltrated water may contribute to groundwater recharge, or it may be taken up by the roots of vegetation and subsequently transpired through the leaves of plants. Infiltrated water also may be evaporated from the

soil during dry-weather periods between storm events, or it may move laterally through the near-surface soils and reappear as surface water in a stream.

Determining Runoff Volume

In its most general form, the relationship of runoff (effective precipitation) to infiltration and abstractions is (see Figure 2.9):

$$D_r = D_p - D_{li} - D_i - D_s - D_e \quad \text{for } D_p > D_{li} \quad \text{and} \quad D_r > D_s$$

$$D_r = 0 \quad \text{for } D_p \leq D_{li} \quad \text{or} \quad D_r \leq D_s$$

where

D_r = total depth of runoff (effective precipitation),

D_p = total depth of precipitation (rainfall),

D_{li} = total initial loss (interception),

D_i = total depth infiltrated after initial losses,

D_s = total depression storage depth, and

D_e = transpiration and evaporation losses (often ignored for short-duration stormwater events).

Volume of runoff is then expressed by

$$V_r = D_r \times A$$

where

V_r = runoff volume,

A = contributing drainage area, and

D_r = total depth of runoff.

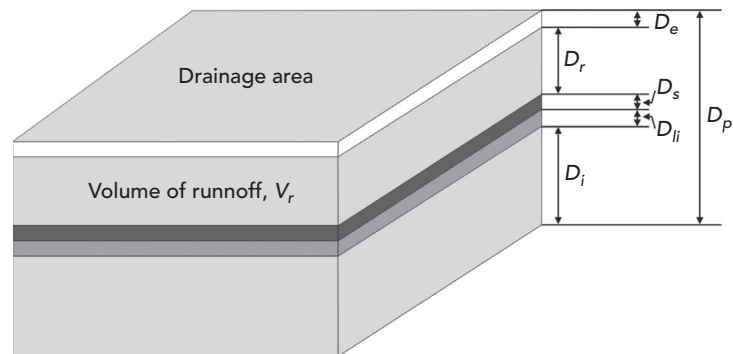


Figure 2.9: Precipitation, abstraction, and runoff volume.

Note: Some hydrology literature refers to runoff volume in terms of depth (in in. or mm), in which case the depth would be multiplied by the contributing area to get the true volume.

Example 2.3: Computing Runoff Volume

A 2.5-h storm deposits 2.0 in. of rainfall over a catchment (watershed) area of 3 ac. If the interception capacity is 0.3 in., the depression storage is 0.2 in., the calculated total infiltration is 0.7 in., and evaporation is negligible, find the total depth of effective precipitation and the total runoff volume for the drainage area.

Solution

Total runoff depth is computed as

$$D_r = 2.0 - 0.3 - 0.7 - 0.2 - 0 = 0.8 \text{ in}$$

And total runoff volume is

$$V_r = (0.8 \text{ in.} \div 12 \text{ in/ft}) \times 3 \text{ ac} = 0.2 \text{ ac} \cdot \text{ft} = 8,712 \text{ ft}^3$$

Although the concept of computing runoff volume just described is a simple one, the problem of determining the amount of rainfall lost to abstractions remains. The following subsections describe a few of the methods used in accounting for abstractions—the Horton equation, the runoff coefficient, and the NRCS (SCS) curve number equation.

Horton Infiltration Equation

A widely used method of representing the infiltration capacity of a soil is the **Horton equation** (Horton, 1939). The Horton equation models a decreasing rate of infiltration over time, which implies that the rate of infiltration decreases as the soil becomes more saturated. For conditions in which the rainfall intensity is always greater than the infiltration capacity (that is, the rainwater supply for infiltration is not limiting), this method expresses the infiltration rate as

$$f(t) = f_c + (f_0 - f_c)e^{-k(t-t_0)}$$

where

$f(t)$ = the infiltration rate (in/h or mm/h) at time t (min),

f_c = a steady-state infiltration rate that occurs for sufficiently large t ,

f_0 = the initial infiltration rate at the time that infiltration begins (that is, at time $t = t_0$), and

k (min⁻¹) = a decay coefficient.

It can be shown theoretically that the steady-state infiltration rate f_c is equal to the saturated vertical hydraulic conductivity of the soil.

Table 2.4: Typical Values of Horton Infiltration Parameters (Rawls et al., 1976)

Soil Type	f_0 (in/h)	f_c (in/h)	k (min ⁻¹)
Alphalpha loamy sand	19.0	1.4	0.64
Carnegie sandy loam	14.8	1.8	0.33
Dorhan loamy sand	3.5	2.6	0.02
Fuquay pebbly loamy sand	6.2	2.4	0.08
Leeffield loamy sand	11.3	1.7	0.13
Tooup sand	23.0	1.8	0.55

Estimation of the parameters f_c , f_0 , and k in the preceding equation can be difficult because of the natural variabilities in antecedent moisture conditions and soil properties. Table 2.4 provides some values recommended by Rawls et al. (1976), although such tabulations should be used with caution. Singh (1992) recommends that f_0 be taken as roughly five times the value of f_c .

For large rain events with a high rainfall intensity that is greater than the infiltration rate, it is possible to estimate the runoff by integrating the Horton equation. The area under the Horton curve would be the maximum potential infiltration, and then by subtracting the maximum potential infiltration from the rainfall depth, the runoff can be estimated. The integrated form of the Horton equation to estimate the maximum potential infiltration is

$$F_{\max} = f_c t + \frac{f_0 - f_c}{k} (1 - e^{-kt})$$

where $F_{\max}$ = maximum potential infiltration depth (in. or mm).

Often, the rainfall intensity during the early part of a storm is less than the potential infiltration capacity of the soil; thus the supply of rainwater is a limiting factor on the infiltration rate. During the time period when the water supply is limiting, the actual infiltration rate is equal to the rate at which rainwater is supplied to the ground surface. This effect is illustrated in Figure 2.10. Later in the storm when the rainfall rate is greater than the infiltration rate, the actual infiltration rate will be greater than that predicted by the preceding equation because infiltration was limited early in the storm. In this case, the runoff is estimated by integration of the rainfall profile above the infiltration curve.

Land Use and Cover Considerations

When applying a physically based equation such as Horton's, care should be taken to consider the impacts of land use and cover conditions. If the parameters used to solve for infiltration consider soil conditions only, the computed runoff may not be applicable.

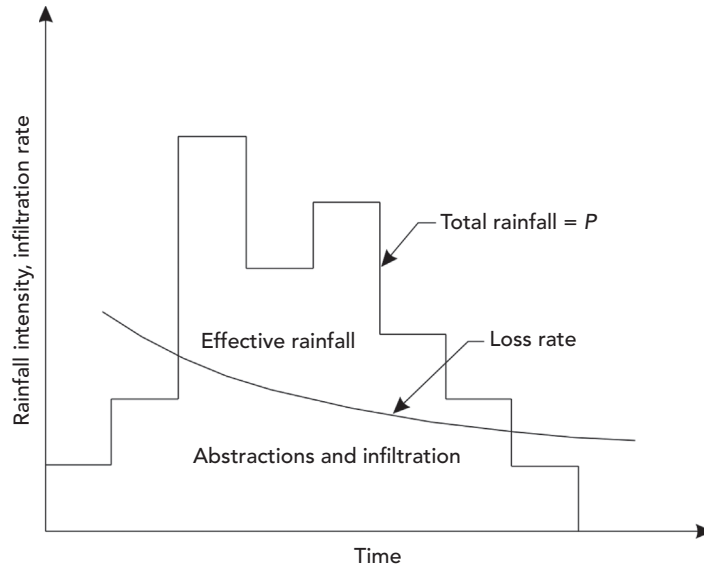


Figure 2.10: Rainfall intensity can limit infiltration rate during the early part of the storm.

Example 2.4: Using Horton's Equation to Determine Infiltration Rate

Using Horton's equation, find the infiltration rate at time 2.0 h for Dothan loamy sand. The rainfall duration is 4 h, and for this event, it takes 0.5 h to reach the interception capacity, at which time infiltration begins.

Solution

From Table 2.4, $f_0 = 3.5 \text{ in/h}$; $f_c = 2.6 \text{ in/h}$; and $k = 0.02 \text{ min}^{-1}$. Thus $f(2\text{h}) = 2.6 \text{ in/h} + (3.5 - 2.6 \text{ in/h})e^{-0.02(2-0.5)} = 3.47 \text{ in/h}$.

Example 2.5: Using Horton's Equation to Estimate Runoff

Using integrated form of Horton's equation, estimate the runoff depth for Dothan loamy sand for a 1-h rain event that produced 3.55 in. of rainfall. Assume the rainfall intensity is always greater than the infiltration rate.

Solution

From Table 2.4, $f_0 = 3.5 \text{ in/h}$; $f_c = 2.6 \text{ in/h}$; and $k = 0.02 \text{ min}^{-1}$. Thus:

$$F_{\max} = 2.6 \text{ in/h} \times 1 \text{ hr} + \frac{3.5 \text{ in/h} - 2.6 \text{ in/h}}{0.02/\text{min} \times 60 \text{ min/h}} \left(1 - e^{-0.02/\text{min} \times 60 \text{ min/h} \times 1 \text{ hr}} \right)$$

$$= 3.12 \text{ inches}$$

Runoff = 3.55 in - 3.12 in = 0.43 inches.

Runoff Coefficient

The concept of the **runoff coefficient** assumes that the effective rainfall intensity i_e is a fraction of the gross rainfall intensity i , or

$$i_e = Ci$$

where C = runoff coefficient ($0 \leq C \leq 1$).

A good way to visualize the C coefficient is to treat it in terms of percent of rainfall. For example, a C of 0.85 would yield a runoff amount (effective precipitation) that is 85% of the gross rainfall. The C coefficient is used with the rational method for peak runoff rate estimation, which is described in Section 2.3.

Example 2.6: Using the Runoff Coefficient to Compute Effective Rainfall Intensity

For a rainfall event with an average intensity of 2.0 in/h falling onto a drainage area having a C coefficient = 0.70, determine the effective rainfall intensity.

Solution

$$i_e = C_i = 0.70 \times 2.0 \text{ in/h} = 1.4 \text{ in/h}$$

Table 2.5 is a listing of recommended runoff coefficients corresponding to various types of land uses. It should be noted, however, that some locales have developed runoff coefficient tables that also consider soil type. Coefficients should be selected carefully for proper application to the particular locale and soil conditions.

When a drainage basin consists of a mixture of land uses, a composite runoff coefficient may be computed for the basin by weighting individual runoff coefficients for each land use by their respective areas, as demonstrated in Example 2.7.

Table 2.5: Runoff Coefficients for Use in the Rational Method (Schaaake et al., 1967)

Type of Area or Development	C
<i>Types of Development</i>	
Urban business	0.70–0.95
Commercial office	0.50–0.70
Residential development	
Single-family homes	0.30–0.50
Condominiums	0.40–0.60

Table 2.5: (Continued)

Type of Area or Development	C
Apartments	0.60–0.80
Suburban residential	0.25–0.40
Industrial development	
Light industry	0.50–0.80
Heavy industry	0.60–0.90
Parks, greenbelts, cemeteries	0.10–0.30
Railroad yards, playgrounds	0.20–0.40
Unimproved grassland or pasture	0.10–0.30
<i>Types of Surface Areas</i>	
Asphalt or concrete pavement	0.70–0.95
Brick paving	0.70–0.80
Roofs of buildings	0.80–0.95
Grass-covered sandy soils	
Slopes 2% or less	0.05–0.10
Slopes 2%–8%	0.10–0.16
Slopes over 8%	0.16–0.20
Grass-covered clay soils	
Slopes 2% or less	0.10–0.16
Slopes 2%–8%	0.17–0.25
Slopes over 8%	0.26–0.36

Example 2.7: Determining the Weighted Runoff Coefficient

Estimate the runoff coefficient for a drainage basin that is made up of 15 ac of park and 30 ac of medium-density, single-family housing.

Solution

From Table 2.5, the runoff coefficients for the park and residential areas are estimated to be 0.20 and 0.40, respectively. The composite runoff coefficient for the entire drainage basin of 45 ac is therefore

$$C = [15(0.20) + 30(0.40)]/45 = 0.33$$

NRCS (SCS) Curve Number Method

In the 1950s, the U.S. Department of Agriculture Soil Conservation Service (SCS; now the Natural Resource Conservation Service or NRCS) developed a procedure to partition the total depth of rainfall represented by a design storm hyetograph into initial abstractions, retention, and effective rainfall (SCS, 1969).

Initial rainfall abstractions consist of all rainfall losses occurring before the beginning of surface runoff, including interception, infiltration, and depression storage. **Retention** refers to the continuing rainfall losses following the initiation of surface runoff, which are predominantly due to continuing infiltration. Conservation of mass requires that

$$F = P - I_a - P_e$$

where

F = equivalent depth of retention (in., mm),

P = total rainfall depth in storm (in., mm),

I_a = equivalent depth of initial abstractions (in., mm), and

P_e = depth of effective precipitation (in., mm).

An assumption made in the development of the curve number method is

$$\frac{F}{S} = \frac{P_e}{P - I_a}$$

where S = maximum possible retention (in., mm).

The value of the maximum possible retention, S , depends on the soil type and land use condition in the drainage basin. The value of S does not include I_a . In essence, the assumption represented by this equation is that the ratio of actual retention to maximum possible retention of water during a storm is equal to the ratio of effective rainfall to maximum possible effective rainfall (total rainfall less initial abstractions). Substitution of $F = P - I_a - P_e$ into the previous equation yields

$$P_e = \frac{(P - I_a)^2}{(P - I_a) + S}$$

for values of $P > I_a$. Data analyzed by the NRCS indicated that I_a is related to S and on average supported the use of the relationship $I_a = 0.2S$. Thus the equation becomes

$$P_e = \frac{(P - 0.2S)^2}{P + 0.8S}$$

where $P > 0.2S$ ($P_e = 0$ when $P \leq 0.2S$). Because the initial abstraction I_a consists of interception, depression storage, and infiltration prior to the onset of direct runoff, it may be appropriate in some applications to assume that $I_a = 0.1S$ or $I_a = 0.3S$ instead of $I_a = 0.2S$. For example, the relationship $I_a = 0.1S$ might be appropriate in a heavily urbanized area where there is little opportunity for initial abstractions to occur. The preceding equation must be modified when the relationship between I_a and S is assumed to be different from $I_a = 0.2S$.

The use of this equation for estimating the depth of effective rainfall during a storm requires an estimate of the maximum possible retention S . NRCS conducted research to approximate S for various soil and cover conditions. In order to provide engineers with tables having a manageable range of coefficients that varying between 0 and 100, the original values for S were rearranged using the following simple relationship:

$$CN = \frac{1000}{S + 10}$$

where

CN = runoff curve number and

S = maximum possible retention (in).

Rearranging, S is related to the **curve number**, CN , as

$$S = \frac{1000}{CN} - 10$$

Practical values of CN range from about 30 to 98, with larger values being associated with more impervious land surfaces. The NRCS has tabulated curve numbers as a function of soil type, land use, hydrologic condition of the drainage basin, and antecedent moisture condition.

Soil Group

Soils in the United States have been classified by the NRCS into four hydrologic groups: Groups A, B, C, and D. **Group A** soils have high infiltration rates (low runoff potential), even when they are thoroughly wetted. Typical Group A soils are well-drained sands and gravels. **Group D** soils are at the opposite end of the spectrum, having low infiltration rates (high runoff potential). Typical Group D soils are clays, shallow soils over nearly impervious material, and soils with a high water table. **Group B** and **Group C** soils are in the midrange of the spectrum.

In the United States, hydrologic soil group information can be determined from the NRCS soil survey of the county in which the project is located. Note that when a drainage basin undergoes urbanization, the hydrologic soil group may change due to compaction of the soil by heavy construction equipment or mixing of soils as a consequence of grading operations.

Cover Type

The surface conditions of a drainage area have a significant impact on runoff. For example, in the exaggerated case of a sandy (Group A) soil completely paved with asphalt, the soil itself will have no impact on the amount of runoff. Even for pervious conditions, cover type plays a significant role in the amount of runoff from a site. For example, a heavily forested area will yield runoff volumes that differ from those of a lawn or plowed field.

Hydrologic Condition

The *hydrologic condition* of rangeland, meadow, or pasture is defined to be “good” if it is lightly grazed and has vegetative cover on more than 75% of the area. Conversely, a “poor” hydrologic condition corresponds to a heavily grazed area with vegetation covering less than 50% of the surface.

Antecedent Moisture Condition

When rainfall events occur in quick succession, the time period between storms may be too short for the soils to dry to their average or normal moisture conditions. When rainfall occurs on soils that are already wet, the net result is that runoff volumes and peaks will be higher than normal. The NRCS (SCS) method accounts for this possibility by allowing the curve number to depend on an *antecedent moisture (AMC) condition*. Three AMC classifications exist. Normal conditions correspond to AMC-II. AMC-I corresponds to a drier condition, and AMC-III to a wetter condition.

Curve numbers corresponding to AMC-I and AMC-III conditions can be computed from AMC-II condition curve numbers using the following equations (Chow, 1959). The computed CN should be rounded to the nearest whole number.

$$CN_I = \frac{4.2CN_{II}}{10 - 0.058CN_{II}}$$

$$CN_{III} = \frac{23CN_{II}}{10 + 0.13CN_{II}}$$

where CN_I , CN_{II} , and CN_{III} = curve numbers for AMC-I, -II, and -III, respectively.

Curve Number Tables

Table 2.6 provides a listing of curve numbers that account for both cover conditions and soil type for normal antecedent moisture conditions (AMC-II). The curve numbers shown for the urban and suburban land use conditions are based on the percentages of directly connected impervious areas in the drainage basin as shown in the table and should be used with caution when the actual percentage of imperviousness in a drainage basin differs from this

Table 2.6: Runoff Curve Numbers for Urban Areas (SCS, 1986)¹

Cover Type and Hydrologic Condition	Average Percent Impervious Area ²	Curve Numbers for Hydrologic Soil Group			
		A	B	C	D
<i>Fully developed urban areas (vegetation established)</i>					
Open space (lawns, parks, golf courses, cemeteries, etc.) ³ :					
Poor condition (grass cover<50%)		68	79	86	89
Fair condition (grass cover 50%–75%)		49	69	79	84
Good condition (grass cover>75%)		39	61	74	80
Impervious areas:					
Paved parking lots, roofs, driveways, etc. (excluding right-of-way)		98	98	98	98
Streets and roads:					
Paved; curbs and storm sewers (excluding right-of-way)		98	98	98	98
Paved; open ditches (including right-of-way)		83	89	92	93
Gravel (including right-of-way)		76	85	89	91
Dirt (including right-of-way)		72	82	87	89
Western desert urban areas:					
Natural desert landscaping (pervious area only) ⁴		63	77	85	88
Artificial desert landscaping (impervious weed barrier, desert shrub with 1–2 in. sand or gravel mulch and basin borders)		96	96	96	96
Urban districts:					
Commercial and business	85	89	92	94	95
Industrial	72	81	88	91	93
Residential districts by average lot size:					
1/8 ac or less (town houses)	65	77	85	90	92
1/4 ac	38	61	75	83	87
1/3 ac	30	57	72	81	86
1/2 ac	25	54	70	80	85
1 ac	20	51	68	79	84
2 ac	12	46	65	77	82
<i>Developing urban areas</i>					
Newly graded area (pervious areas only no vegetation) ⁵		77	86	91	94
Idle lands (CNs are determined using cover types similar to those in 2-2(c) (SCS, 1986)					

¹ Average runoff condition, and $I_a = 0.2$ s.

² The average percent impervious area shown was used to develop the composite CNs. Other assumptions are as follows: impervious areas are directly connected to the drainage system, impervious areas have a CN of 98, and pervious areas are considered equivalent to open space in good hydrologic condition.

³ CNs shown are equivalent to those of pasture. Composite CNs may be computed for other combinations of open space cover type.

⁴ Composite CNs for natural desert landscaping should be computed using Figures 2.3 or 2.4 (in SCS, 1986) based on the impervious area percentage (CN = 98) and the pervious area CN. The pervious area CNs are assumed equivalent to desert shrub in poor hydrologic condition.

⁵ Composite CNs to use for the design of temporary measures during grading and construction should be computed using Figures 2.3 or 2.4 (in SCS, 1986) based on the degree of development (impervious area percentage) and the CNs for the newly graded pervious areas.

assumed value. When necessary, a composite curve number can be developed as an area-weighted average of individual curve numbers. The composite CN should be rounded to the nearest whole number.

Additional information on hydrologic condition and curve numbers for land uses other than those contained in Table 2.6 can be found in the National Engineering Handbook, Section 4: Hydrology (NEH-4) (SCS, 1969) and TR-55 (SCS, 1986).

Example 2.8: Estimating Runoff Depth and Volume Using the NRCS (SCS) Curve Number Method (Modified from SCS, 1986)

Estimate the curve number, depth of runoff (effective precipitation), and runoff volume for a 1,000-ac drainage basin if the total depth of precipitation is 5.0 in. All soils in the basin are in hydrologic soil group C. The proposed land use is 50% detached houses with 0.25-ac lots; 10% townhouses with 0.125-ac lots; 25% schools, parking lots, plazas, and streets with curbs and gutters; and 15% open space, parks, and schoolyards with good grass cover. Use an antecedent soil moisture condition of AMC-III. The detached housing and townhouse areas have directly connected impervious area percentages similar to those assumed in Table 2.6.

Solution

The composite curve number corresponding to AMC-II conditions is computed as a weighted average, as shown in Table 2.7.

Thus the composite CN is computed as

$$CN = 86,100 \div 1,000 = 86$$

The curve number corresponding to AMC-III moisture conditions is found to be

$$CN_{III} = \frac{23(86)}{10 + 0.13(86)} = 93$$

Table 2.7: Calculations for Example 2.8

Land Use	Area (ac)	CN	Product
Detached houses (0.25-ac lots)	500	83	41,500
Townhouses (0.125-ac lots)	100	90	9,000
Streets, plazas, etc.	250	98	24,500
Open space, parks, etc.	150	74	11,100
Sum	1,000		86,100

The maximum possible retention for this basin at AMC-III is

$$S = (1,000/93) - 10 = 0.75 \text{ in}$$

Initial abstractions are estimated to be

$$I_a = 0.2S = 0.15 \text{ in}$$

Because $P > I_a$, the depth of runoff (effective precipitation) is estimated as

$$P_e = [5.0 - 0.2(0.75)]^2 / (5.0 + 0.8(0.75)) = 4.20 \text{ in}$$

The total volume of runoff is therefore

$$V_r = (4.2 \text{ in} / 12 \text{ in/ft}) (1,000 \text{ ac}) = 350 \text{ ac}\cdot\text{ft}$$

2.3 Computing Peak Runoff Flow Rate

To design stormwater conveyance or detention systems, the engineer must obtain information on runoff flow rates. Two basic levels of analysis exist. The first level is a peak flow calculation to determine the maximum runoff flow rate at a given point resulting from a storm event. This level of analysis is often sufficient for designing storm sewers and culverts whose only function is to convey runoff away from areas where it is unwanted. The second level, which is more complex, consists of the generation of a runoff **hydrograph**, which provides information on flow rate versus time and runoff volume. This type of information is necessary in the analysis of detention and retention facilities where time and volume considerations are critical. In some situations, hydrograph analysis is also required for storm sewer analyses.

After you determine the required analysis approach, you must choose techniques for computing how much of the rainfall will contribute to surface runoff and, in the case of a hydrograph analysis, how this flow will be distributed through time. These techniques are sometimes dictated by local regulations.

Time of Concentration

The maximum amount of flow coming from any watershed is related to the amount of time it takes for the entire watershed to be contributing to the flow. In other words, it may start raining right now, but it could be several minutes (or even hours) before the water that falls on some parts of the watershed actually makes its way to the discharge point.

Some places in a watershed are hydraulically closer to the discharge point than others, but for peak flow generation only the most hydraulically remote

location is considered crucial. The amount of time that it takes for the first drop of water falling on this location to work its way to the discharge point is called the **time of concentration**.

Many methods for calculating time of concentration are given in various private, federal, and local publications. Although each of these methods is different (in some cases only slightly), they are all based on the type of ground cover, the slope of the land, and the distance along the flow path. In most locales, there is also a minimum time of concentration (typically 5–10 min) recommended for small watersheds, such as a section of a parking lot draining to a storm sewer. Some methods predict the response time directly, whereas others predict the velocity of flow. The predicted velocity coupled with estimates of the flow path length can then be used to estimate the response time.

With few exceptions, methods for prediction of basin response time are empirical in nature. Consequently, large errors in response time estimates can be expected to occur if these methods are not carefully selected and applied, and these errors can significantly affect peak runoff estimates. The method selected for estimation of t_c should be one that was developed for basin conditions similar to those in the drainage basin for which an estimate is desired.

Table 2.8 lists several commonly used methods for estimation of the time of concentration of a drainage basin. Figure 2.11 illustrates average overland flow velocities as a function of land use characteristics and surface slope. When flows

Table 2.8: Commonly Used Methods for Estimation of the Time of Concentration, in minutes

Equation	Source	Remarks
$t_c = 60LA^{0.4}/DS^{0.2}$	Williams (1922)	L = basin length (mi), A = basin area (mi ²), D = diameter (mi) of a circular basin of area A , and S = basin slope (%). The basin area should be smaller than 50 mi ²
$t_c = KL^{0.77}/S^n$	Kirpich (1940)	Developed for small drainage basins in Tennessee and Pennsylvania, with basin areas from 1 to 112 ac. L = basin length (ft), S = basin slope (ft/ft), K = 0.0078 and n = 0.385 for Tennessee; K = 0.0013 and n = 0.5 for Pennsylvania. The estimated t_c should be multiplied by 0.4 if the overland flow path is concrete or asphalt, or by 0.2 if the channel is concrete-lined

Table 2.8: (Continued)

Equation	Source	Remarks
$t_c = (2LN/3S^{0.5})^{0.47}$	Hathaway (1945), Kerby (1959)	Drainage basins with areas of less than 10 ac and slopes of less than 0.01. This is an overland flow method. L = overland flow length from basin divide to a defined channel (ft), S = overland flow path slope (ft/ft), and N is a flow retardance factor ($N = 0.02$ for smooth impervious surfaces; 0.10 for smooth, bare packed soil; 0.20 for poor grass, row crops, or moderately rough bare surfaces; 0.40 for pasture or average grass; 0.60 for deciduous timberland; and 0.80 for conifer timberland, deciduous timberland with deep ground litter, or dense grass)
$t_c = 300(L/S)^{0.5}$	Johnstone and Cross (1949)	Developed for basins in the Scotie and Sandusky River watersheds with areas between 25 and 1,624 mi ² . L = basin length (mi), and S = basin slope (ft/mi)
$t_c = \{(0.007I + c)/S^{0.33}\} \times (IL/43,200)^{-0.67} L/60$	Izzard (1946)	Hydraulically derived formula. I = effective rainfall intensity (in/h), S = slope of overland flow path (ft/ft), L = length of overland flow path (ft), and c is a roughness coefficient ($c = 0.007$ for smooth asphalt, 0.012 for concrete pavement, 0.017 for tar and gravel pavement, and 0.060 for dense bluegrass turf)
$t_c = 0.94I^{-0.4}(Ln/S^{0.5})^{0.6}$	Henderson and Wooding (1964)	Based on kinematic wave theory for flow on an overland flow plane. I = rainfall intensity (in/h), L = length of overland flow (ft), n = Manning's roughness coefficient, S = overland flow plane slope (ft/ft)
$AV_2 = Q = A\sqrt{2g(H - H_L)}$	Federal Aviation Agency (1970)	Developed based on airfield drainage data. C = Rational Method runoff coefficient, L = overland flow length (ft), and S = slope (%)
$t_c = \frac{1}{60} \sum (L/V)_i$	Soil Conservation Service (1986)	Time of concentration is developed as a sum of individual travel times. L = length of an individual flow path (ft) and V = velocity of flow over an individual flow path (ft/s). V may be estimated using Figure 2.11 or using Manning's equation

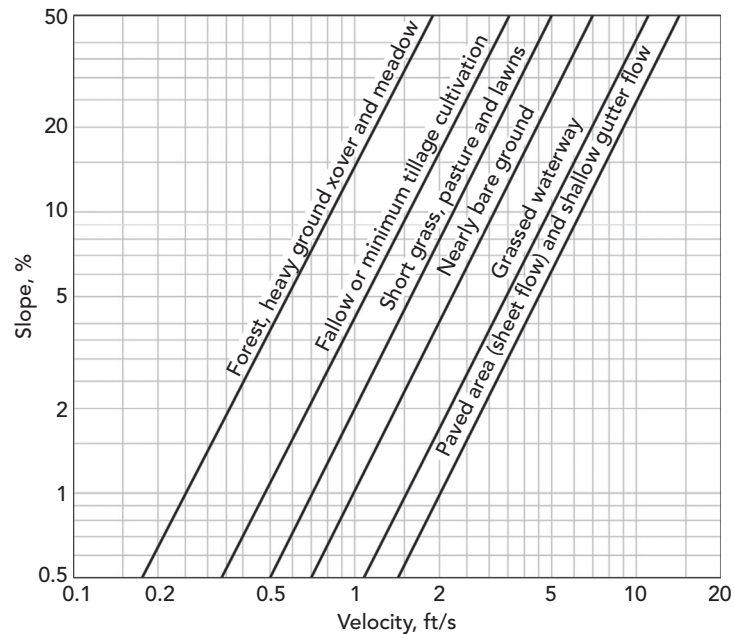


Figure 2.11: Average overland flow velocities as a function of land use characteristics and surface slope.

are channelized in gutters, open channels, or storm sewers, Manning's equation may be used to estimate the velocity of flow (see Chapter 1).

Example 2.9: Time of Concentration

An urbanized drainage basin is shown in Figure 2.12. Three types of flow conditions exist from the furthest point of the drainage basin to its outlet. Estimate the time of concentration based on the following data:

Reach	Flow Description	Slope (%)	Length (ft)
A–B	Overland (forest)	7	500
B–C	Overland (shallow gutter)	2	900
C–D	Storm sewer with manholes, inlets, etc. ($n = 0.015$, dia. = 3 ft)	1.5	2,000
D–E	Open channel, gunite-lined, trapezoidal ($B = 5$ ft, $y = 3$ ft, $z = 1:1$, $n = 0.019$)	0.5	3,000

Solution

For the reach from A to B, the average flow velocity is $V = 0.7$ ft/s. The travel time for that reach is therefore

$$t_{AB} = L/V = 500/0.7 = 714 \text{ s}$$

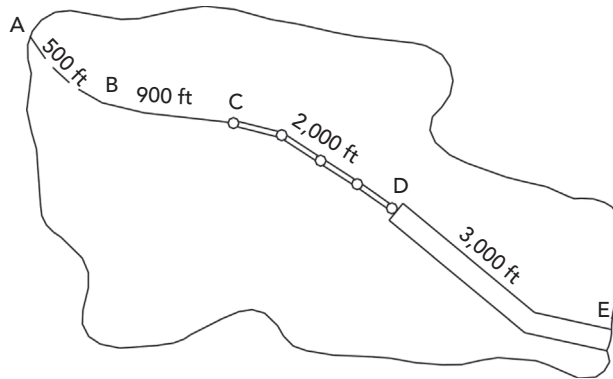


Figure 2.12: Flow paths in a drainage basin for calculation of t_c .

Similarly, for the reach from B to C, the average flow velocity is $V = 2.8 \text{ ft/s}$. The travel time for that reach is therefore

$$t_{BC} = L/V = 900/2.8 = 321 \text{ s}$$

To compute the travel time in the storm sewer from C to D, Manning's equation is employed to compute the pipe-full velocity:

$$V = \frac{1.49}{n} \left(\frac{D}{4} \right)^{2/3} S^{1/2} = \frac{1.49}{0.015} \left(\frac{3}{4} \right)^{2/3} (0.015)^{1/2} = 10 \text{ ft/s}$$

The travel time for that reach is therefore

$$t_{CD} = L/V = 2,000/10 = 200 \text{ s}$$

Travel time in the open channel from D to E is computed using the bank-full velocity, again found via Manning's equation:

$$V = \frac{1.49}{n} R^{2/3} S^{1/2} = \frac{1.49}{0.019} (1.78)^{2/3} (0.005)^{1/2} = 8.2 \text{ ft/s}$$

The travel time for that reach is therefore

$$t_{DE} = L/V = 3,000/8.2 = 366 \text{ s}$$

The time of concentration is the sum of the four individual travel times and is

$$t_c = 1,601 \text{ s} = 0.44 \text{ h}$$

The Rational Method

The Rational Method is an equilibrium-based approach to peak flow estimation that uses rainfall intensity data and watershed characteristics to predict peak flows for a rainfall event. The Rational Method is a popular choice for storm sewer design because this type of design usually considers only peak flows, and because of the simplicity of the calculations involved.

At the most fundamental level, the Rational Method assumes that a steady state is attained such that the rainfall inflow rate of water onto a drainage basin is equal to the outflow rate of water from the basin. If one expresses the volumetric inflow rate as the product of the basin area A and the effective rainfall intensity i_e , the outflow rate Q is obtained as $Q = i_e A$. Further, if one accounts for abstractions, the effective intensity is a product of the actual rainfall intensity and a runoff coefficient, resulting in

$$Q = CiA$$

where

Q = runoff rate (ac.in/h, ha mm/h),

C = runoff (abstractions) coefficient,

i = rainfall intensity (in/h, mm/h), and

A = drainage area (ac, ha).

Because $1 \text{ ac.in/hr} = 1.008 \text{ cfs} \approx 1 \text{ cfs}$, engineers performing calculations by hand typically ignore the conversion factor and simply assume that the discharge Q is in units of cfs. In SI units, a conversion factor of 0.278 will yield Q in units of m^3/s .

The time of concentration is the smallest time for which the entire basin is contributing runoff to the basin outlet; therefore the storm duration must be at least as long as the time of concentration if a steady-state condition is to be achieved. Steady-state conditions also dictate that the storm intensity be spatially and temporally uniform. It is not reasonable to expect that rainfall will be spatially uniform over a large drainage basin, or that it will be temporally uniform over a duration at least as long as the time of concentration when t_c (and hence A) is large. Therefore these conditions limit the applicability of the Rational Method to small drainage basins. An upper limit of 200 ac has been suggested by some, but the limit should really depend on the storm characteristics of the particular locale. These local characteristics may limit the applicability of the Rational Method to basins smaller than 10 ac in some cases.

When several drainage basins (or subbasins) discharge to a common facility such as a storm sewer or culvert, the time of concentration should be taken as the longest of all the individual times of concentration and should include

pipe travel times when appropriate. Further, the total drainage area served (the sum of the individual basin areas) should be no larger than the 200-ac limit (or smaller where applicable) of the Rational Method.

The basic steps for applying the Rational Method are as follows:

Step 1: Apply IDF Data

Develop or obtain a set of IDF curves for the locale in which the drainage basin resides. Assume that the storm duration is equal to the time of concentration and determine the corresponding intensity for the recurrence interval of interest. Note that the assumption that the storm duration and time of concentration are equal is conservative in that it represents the highest intensity for which the entire drainage area can contribute.

Step 2: Compute Watershed Area

The basin area A can be estimated using topographic maps, computer tools such as CAD or GIS software, or by field reconnaissance. The time of concentration may be estimated using the procedures discussed in the preceding subsection.

Step 3: Choose C Coefficients

The runoff coefficient C may be estimated using Table 2.5 if the land use is homogeneous in the basin, or a composite C value may be estimated if the land use is heterogeneous (see Example 2.7).

Step 4: Solve Peak Flow

Finally, the peak runoff rate from the basin can be computed using the equation $Q = CiA$.

The following example illustrates the use of the Rational Method for several subbasins draining into a common storm sewer system.

Example 2.10: Computing Flows for Multiple Subbasins with the Rational Method

Figure 2.13 is a plan view of a storm sewer system draining three subbasins. Use the Rational Method to determine the peak discharge in each pipe and size each pipe assuming the pipes flow full. Assume also that the pipes will be concrete with $n = 0.013$. Perform the calculations for a storm recurrence interval of 25 years. Subbasin and pipe characteristics and IDF data for the 25-year event are tabulated as follows:

Subbasin	A (ac)	C	t_c (min)
A	6.0	0.6	20
B	4.0	0.8	10
C	4.5	0.8	15

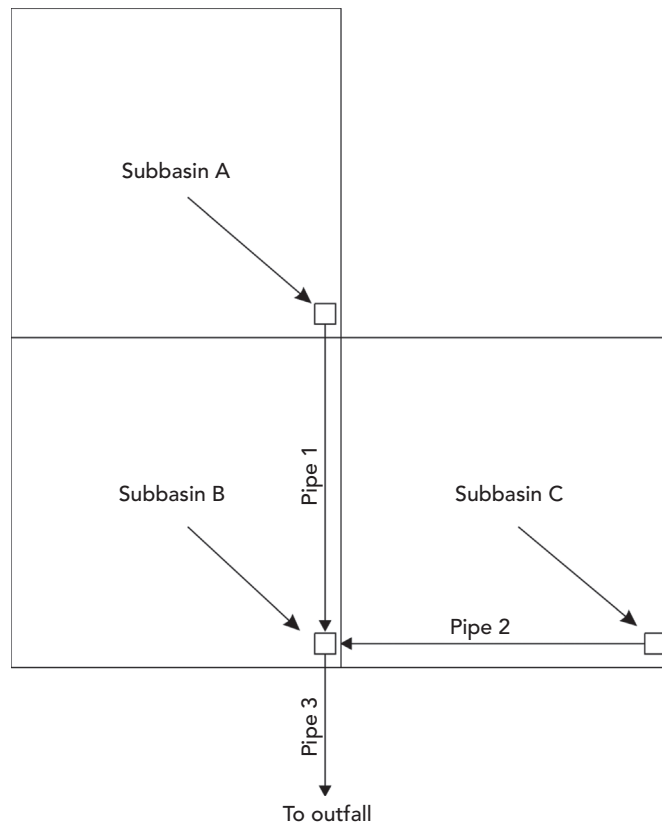


Figure 2.13: System for Example 2.10.

Pipe	Length (ft)	Slope (%)
1	500	1.0
2	400	1.2
3	500	0.9

Duration (min)	Intensity (in/h)
5	8.40
10	7.02
15	5.96
20	5.26
30	4.42
60	2.97

Solution

Flow into Pipe 1 occurs from Subbasin A only. Using the time of concentration as the storm duration, the 25-year rainfall intensity is 5.26 in/h. The peak discharge used in sizing Pipe 1 is therefore

$$Q = 0.6(5.26)(6.0) = 19 \text{ cfs}$$

Assuming that Pipe 1 is flowing full, its required diameter D may be found using Manning's equation as follows:

$$D = \left(\frac{Qn}{0.464S^{1/2}} \right)^{3/8} = \left(\frac{19(0.013)}{0.464(0.01)^{1/2}} \right)^{3/8} = 1.87 \text{ ft}$$

Rounding up to the next commercially available size, Pipe 1 should have a diameter of 24 in. Use FlowMaster to determine that the depth of flow is 1.40 ft and the area of flow is 2.36 ft².

Because the cross-sectional area of Pipe 1 is 2.36 ft², the average velocity in Pipe 1 is

$$V = Q/A = 19/2.36 = 8.05 \text{ ft/s}$$

The travel time in Pipe 1 is

$$t = L/V = 500/8.05 = 62 \text{ s} = 1.04 \text{ min}$$

Pipe 2 is treated the same way as Pipe 1, recognizing that runoff from Subbasin C only enters Pipe 2. The peak discharge from Subbasin C is $Q = 22$ cfs, and the required diameter of Pipe 2 is $D = 24$ in. The travel time in Pipe 2 is $t = 45 \text{ s} = 0.75 \text{ min}$.

Pipe 3 must be sized to handle the runoff from all three of the subbasins, which have a total area of $A = 14.5$ ac. The runoff coefficient for the combined areas is computed as a composite value and is

$$C = \frac{6(0.6) + 4(0.8) + 4.5(0.8)}{14.5} = 0.72$$

The time of concentration is computed as the longest of the travel times to the upstream end of Pipe 3. These travel times are (1) the time of concentration of Subbasin B (10 min), (2) the time of concentration of Subbasin A plus the travel time in Pipe 1 ($20 + 1.0 = 21$ min), and (3) the time of concentration of Subbasin C plus the travel time in Pipe 2 ($15 + 0.75 = 15.75$ min). Thus the time of concentration for Pipe 3 is 21 min, and the corresponding rainfall intensity (by interpolation) is 5.17 in/h.

The peak discharge for Pipe 3 is

$$Q = 0.72(5.17)(14.5) = 54 \text{ cfs}$$

The required diameter of Pipe 3 (rounded to the nearest standard size) is 36 in.

NRCS (SCS) Peak Flow Estimation

The NRCS (SCS) has developed a simple procedure for estimation of peak runoff rates for small drainage basins. Graphical and tabular solutions to this procedure are presented in Technical Release 55 (TR-55) (SCS, 1986), and tabular and regression-based solutions are presented in the U.S. Department of Transportation Federal Highway Administration's Hydrologic Engineering Circular No. 22 (HEC-22) (Brown et al., 2009). The HEC-22 regression procedure is described in this subsection.

For use with this method, the drainage basin should have a fairly homogeneous distribution of CN , and the composite CN should be at least 40. The basin should also have one main channel, although branches are acceptable if they have approximately equal times of concentration. Further, this method was developed for use with 24-h storm rainfall depths. Its use with other storm durations is not advised.

This method is implemented by first applying the following equation to estimate the depth of effective precipitation during a storm. The peak discharge is estimated as

$$Q_p = q_u A P_e$$

where

Q_p = the peak flow rate (cfs),

q_u = the unit peak flow rate (cfs/mi²/in),

A = the drainage basin area (mi²), and

P_e = the depth of effective precipitation (in).

The unit peak flow rate is estimated as

$$q_u = 10^K$$

where

$$K = C_0 + C_1 \log_{10} t_c + C_2 (\log_{10} t_c)^2$$

C_0 , C_1 , and C_2 are coefficients listed in Table 2.9 as a function of the SCS 24-h design storm distribution type and the ratio I_a/P ; and t_c is the time of concentration (h). P is the 24-h rainfall depth, and I_a is the depth of initial abstractions. For convenience, the ratio I_a/P is tabulated in Table 2.10 as a function of the

Table 2.9: Coefficients for SCS (NRCS) Peak Discharge Method (SCS, 1996)

Storm Type	I_a/P	C_0	C_1	C_2
I	0.10	2.30550	-0.51429	-0.11750
	0.15	2.27044	-0.50908	-0.10339
	0.20	2.23537	-0.50387	-0.08929
	0.25	2.18219	-0.48488	-0.06589
	0.30	2.10624	-0.45696	-0.02835
	0.35	2.00303	-0.40769	-0.01983
	0.40	1.87733	-0.32274	0.05754
	0.45	1.76312	-0.15644	0.00453
	0.50	1.67889	-0.06930	0
IA	0.10	2.03250	-0.31583	-0.13748
	0.15	1.97614	-0.29899	-0.10384
	0.20	1.91978	-0.28215	-0.07020
	0.25	1.83842	-0.25543	-0.02597
	0.30	1.72657	-0.19826	0.02633
	0.35	1.70347	-0.17145	0.01975
	0.40	1.68037	-0.14463	0.01317
	0.45	1.65727	-0.11782	0.00658
	0.50	1.63417	-0.09100	0
II	0.10	2.55323	-0.61512	-0.16403
	0.15	2.53125	-0.61698	-0.15217
	0.20	2.50928	-0.61885	-0.14030
	0.25	2.48730	-0.62071	-0.12844
	0.30	2.46532	-0.62257	-0.11657
	0.35	2.41896	-0.61594	-0.08820
	0.40	2.36409	-0.59857	-0.05621
	0.45	2.29238	-0.57005	-0.02281
	0.50	2.20282	-0.51599	-0.01259
III	0.10	2.47317	-0.51848	-0.17083
	0.15	2.45395	-0.51687	-0.16124
	0.20	2.43473	-0.51525	-0.15164
	0.25	2.41550	-0.51364	-0.14205
	0.30	2.39628	-0.51202	-0.13245
	0.35	2.35477	-0.49735	-0.11985
	0.40	2.30726	-0.46541	-0.11094
	0.45	2.24876	-0.41314	-0.11508
	0.50	2.17772	-0.36803	-0.09525

Table 2.10: I_a/P for Selected Rainfall Depths and Curve Numbers (Adapted from SCS, 1996)

P (in)	Curve Number, CN											
	40	45	50	55	60	65	70	75	80	85	90	95
0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.44	0.21
1.00	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.35	0.22	0.11
1.50	0.50	0.50	0.50	0.50	0.50	0.50	0.50	0.44	0.33	0.24	0.15	0.10
2.00	0.50	0.50	0.50	0.50	0.50	0.50	0.43	0.33	0.25	0.18	0.11	0.10
2.50	0.50	0.50	0.50	0.50	0.50	0.43	0.34	0.27	0.20	0.14	0.10	0.10
3.00	0.50	0.50	0.50	0.50	0.44	0.36	0.29	0.22	0.17	0.12	0.10	0.10
3.50	0.50	0.50	0.50	0.47	0.38	0.31	0.24	0.19	0.14	0.10	0.10	0.10
4.00	0.50	0.50	0.50	0.41	0.33	0.27	0.21	0.17	0.13	0.10	0.10	0.10
4.50	0.50	0.50	0.44	0.36	0.30	0.24	0.19	0.15	0.11	0.10	0.10	0.10
5.00	0.50	0.49	0.40	0.33	0.27	0.22	0.17	0.13	0.10	0.10	0.10	0.10
5.50	0.50	0.44	0.36	0.30	0.24	0.20	0.16	0.12	0.10	0.10	0.10	0.10
6.00	0.50	0.41	0.33	0.27	0.22	0.18	0.14	0.11	0.10	0.10	0.10	0.10
6.50	0.46	0.38	0.31	0.25	0.21	0.17	0.13	0.10	0.10	0.10	0.10	0.10
7.00	0.43	0.35	0.29	0.23	0.19	0.15	0.12	0.10	0.10	0.10	0.10	0.10
7.50	0.40	0.33	0.27	0.22	0.18	0.14	0.11	0.10	0.10	0.10	0.10	0.10
8.00	0.38	0.31	0.25	0.20	0.17	0.13	0.11	0.10	0.10	0.10	0.10	0.10
8.50	0.35	0.29	0.24	0.19	0.16	0.13	0.10	0.10	0.10	0.10	0.10	0.10
9.00	0.33	0.27	0.22	0.18	0.15	0.12	0.10	0.10	0.10	0.10	0.10	0.10
9.50	0.32	0.26	0.21	0.17	0.14	0.11	0.10	0.10	0.10	0.10	0.10	0.10
10.00	0.30	0.24	0.20	0.16	0.13	0.11	0.10	0.10	0.10	0.10	0.10	0.10
10.50	0.29	0.23	0.19	0.16	0.13	0.10	0.10	0.10	0.10	0.10	0.10	0.10
11.00	0.27	0.22	0.18	0.15	0.12	0.10	0.10	0.10	0.10	0.10	0.10	0.10
11.50	0.26	0.21	0.17	0.14	0.12	0.10	0.10	0.10	0.10	0.10	0.10	0.10
12.00	0.25	0.20	0.17	0.14	0.11	0.10	0.10	0.10	0.10	0.10	0.10	0.10
12.50	0.24	0.20	0.16	0.13	0.11	0.10	0.10	0.10	0.10	0.10	0.10	0.10
13.00	0.23	0.19	0.15	0.13	0.10	0.10	0.10	0.10	0.10	0.10	0.10	0.10
13.50	0.22	0.18	0.15	0.12	0.10	0.10	0.10	0.10	0.10	0.10	0.10	0.10
14.00	0.21	0.17	0.14	0.12	0.10	0.10	0.10	0.10	0.10	0.10	0.10	0.10
14.50	0.21	0.17	0.14	0.11	0.10	0.10	0.10	0.10	0.10	0.10	0.10	0.10
15.00	0.20	0.16	0.13	0.11	0.10	0.10	0.10	0.10	0.10	0.10	0.10	0.10

Table 2.11: Adjustment Factor, F_p , for Ponds and Swampy Areas (SCS, 1996)

Pond and Swamp Area (%)	F_p
0	1.00
0.2	0.97
1.0	0.87
3.0	0.75
5.0	0.72

curve number. The ratio I_a/P should not be less than 0.1, nor larger than 0.5. The time of concentration should be in the range of 0.1 to 10 hours.

When ponding or swampy areas exist in a drainage basin, the peak discharge should be reduced to account for the temporary storage of runoff. In this case, an adjusted peak flow Q_{pa} should be computed as

$$Q_{pa} = F_p Q_p$$

where F_p is an adjustment factor obtained from Table 2.11. Ponds and swampy areas lying along the flow path used for computation of the time of concentration should not be included in the area percentage in Table 2.11. Neither channel nor reservoir routing can be accommodated with this method.

Example 2.11: Computing Peak Discharge Using the NRCS Method

Compute the peak discharge from the 1,000-ac (1.56 mi²) drainage basin described in Example 2.8, but using AMC-II soil moisture conditions. The rainfall distribution is an SCS Type II distribution, and the time of concentration of the drainage basin is 0.9 h. There are no ponds or swampy areas in the basin.

Solution

From Example 2.8, $P = 5.0$ in and $CN = 86$ for AMC-II. For this condition, the maximum possible retention $S = 1.63$ in, and the effective precipitation is $P_e = 3.46$ in.

From Table 2.10, the ratio I_a/P is 0.10; and from Table 2.9, the coefficients C_0 , C_1 , and C_2 are $C_0 = 2.55323$, $C_1 = -0.61512$, and $C_2 = -0.16403$, respectively. Therefore

$$K = 2.55323 - 0.61512 \log_{10}(0.9) - 0.16403 [\log_{10}(0.9)]^2 = 2.58104$$

The unit peak flow rate is

$$q_u = 10^{2.58104} = 381 \text{ cfs/mi}^2/\text{in}$$

The peak discharge from the drainage basin is computed as

$$Q_p = 381(1.56)(3.46) = 2,060 \text{ cfs}$$

2.4 Computing Hydrographs

A **hydrograph** represents the flow rate as it varies over time at a particular location within a watershed. The integrated area under the hydrograph represents the total hydrograph volume of water. Base flow represents subsurface flow from groundwater that discharges into the conveyance channel. Many storm conveyance channels are dry at the beginning of a rainfall event, which equates to a base flow of zero as long as the water table does not rise into the channel during the storm. Figure 2.14 displays the various components of a surface runoff hydrograph, with a base flow of zero.

Estimation of a complete runoff hydrograph, as opposed to merely the peak rate of runoff, is necessary to account for the effects of storage in a drainage basin. Because a hydrograph accounts for volume and flow variations over an entire rainfall event, it is useful for analyzing complex watersheds and designing detention ponds. Hydrographs enable you to assess the effects of storage associated with natural ponds and lakes, and with artificial stormwater detention and/or retention facilities. Hydrograph estimation is also necessary in order to assess the impacts of storm duration and/or hyetograph shape on runoff production, and in cases when two or more adjacent drainage basins (or subbasins) are discharging to a common stream.

Procedures for estimating unit hydrographs and performing discrete convolution are presented in this and the following section. **Unit hydrograph estimation methods** involve several basic steps:

1. Establish a design storm rainfall hyetograph.
2. Convert the design storm rainfall hyetograph into a runoff (effective rainfall) hyetograph by subtraction of rainfall abstractions.

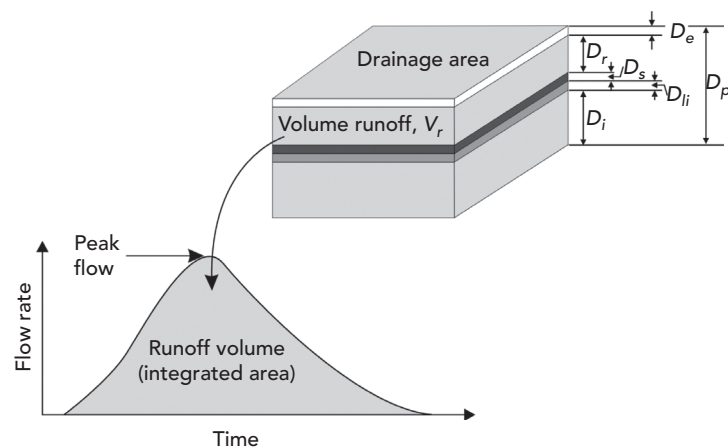


Figure 2.14: Hydrograph definition sketch.

3. Estimate a unit hydrograph for the drainage basin of interest.
4. Merge the runoff (effective rainfall hyetograph) and the unit hydrograph using discrete convolution to produce what is called a **direct runoff hydrograph**, which represents the surface runoff hydrograph for the complex storm pattern represented by the rainfall hyetograph.
5. If appropriate, add **base flow** to the direct runoff hydrograph to obtain the **total hydrograph** representing both surface runoff and base flow.

Creating Runoff (Effective Rainfall) Hyetographs

The first step in calculating a runoff hydrograph from a given rainfall hyetograph is to calculate a runoff (effective rainfall) hyetograph. Provided that the data is available, the most accurate way to do this is to develop a runoff hyetograph by modeling abstractions and infiltration within each time step of a rainfall event. To do this, subtract the interception from the beginning of a gross rainfall hyetograph, subtract infiltration from what remains after interception has been accounted for, and subtract depression storage from what remains after both interception and infiltration have been accounted for. This approach is physically based and is not limited to use with any particular storm duration or rainfall hyetograph shape.

Example 2.12: Computing an Effective Rainfall Hyetograph

Develop a runoff (effective rainfall) hyetograph for a watershed with an initial loss (interception capacity) of 0.3 in, a depression storage capacity of 0.2 in, and Horton infiltration parameters of $f_0 = 1.5$ in/h, $f_c = 0.3$ in/h, and $k = 0.04$ min⁻¹. The rainfall hyetograph is tabulated as follows:

t (min)	P (in)
0–10	0.24
10–20	0.46
20–30	1.17
30–40	0.58
40–50	0.35
50–60	0.17

Solution

The interception capacity of 0.3 in is subtracted first. Because 0.24 in of rainfall occurs during the first 10 min of the storm, all of that rainfall plus an additional 0.06 in of rainfall occurring in the second 10 min of the storm

is lost to interception. Thus, after accounting for interception, the rainfall hyetograph is

t (min)	P (in)
0–10	0
10–20	0.4
20–30	1.17
30–40	0.58
40–50	0.35
50–60	0.17

The infiltration rate, $f(t)$, can be calculated and tabulated as a function of t using the Horton equation (see Section 2.2), where t_0 is the time at which rainwater first begins to infiltrate ($t_0 = 10$ min in this example, because rainfall prior to that time is lost to interception and hence is not available for infiltration). The first column in the following table is the time since the beginning of rainfall, and the second is the time since the beginning of infiltration. The third column is the computed infiltration rate.

The fourth column contains incremental infiltration depths for each 10-min period during the storm. For example, the first value of $F = 0.22$ in is computed as the average of the current and preceding infiltration rates, multiplied by the time interval of $\Delta t = 10$ min = $1/6$ h (for example, $0.22 = [(1.50 + 1.10)/2]/(6)$).

t (min)	$t-t_0$ (min)	$f(t)$ (in/h)	Incr. F (in)
0			
10	0	1.5	
20	10	1.1	0.22
30	20	0.84	0.16
40	30	0.66	0.13
50	40	0.54	0.1
60	50	0.46	0.08

Subtraction of the infiltration depth in each time interval from the corresponding rainfall depth remaining after interception leads to the following hyetograph (any negative values produced should be set equal to zero):

t (min)	P (in)
0–10	0
10–20	0.18
20–30	1.01
30–40	0.45
40–50	0.25
50–60	0.09

Finally, subtraction of the depression storage capacity of 0.2 in leads to the following effective rainfall hyetograph:

t (min)	P_e (in)
10–20	0
10–20	0
20–30	0.99
30–40	0.45
40–50	0.25
50–60	0.09

The rainfall hyetograph and the effective rainfall hyetograph are illustrated in Figures 2.15 and 2.16. Note that the effective rainfall intensities are less than the actual rainfall intensities. Also note that effective rainfall, and hence surface runoff, does not begin (in this example) until 20 min after the beginning of the storm.

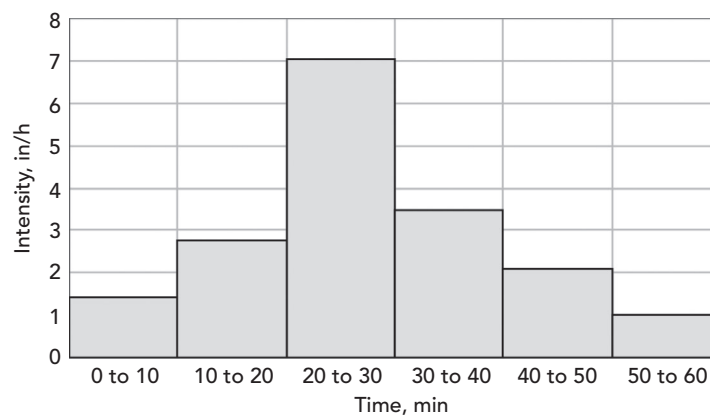


Figure 2.15: Rainfall hyetograph for Example 2.12.

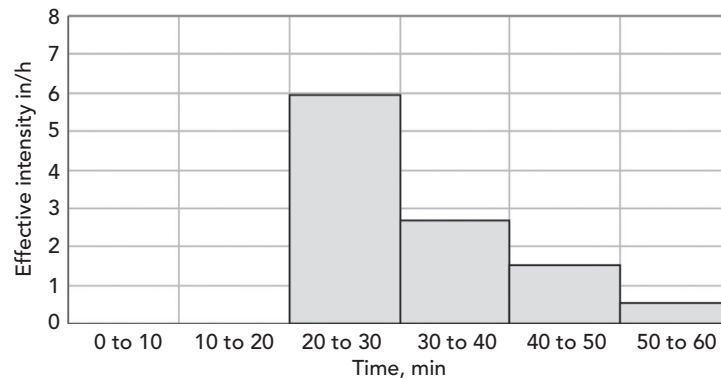


Figure 2.16: Effective rainfall (runoff) hyetograph for Example 2.12.

Unit Hydrographs

A **unit hydrograph** is an important intermediate step in computing a full runoff hydrograph for design or analysis purposes. After a unit hydrograph is established for a particular basin, the data can be used to compute the complete runoff hydrographs resulting from various rainfall events. The basic theory rests on the assumption that the runoff response of a drainage basin to an effective rainfall input is linear. Practically speaking, this assumption means that the concepts of proportionality and superposition can be applied. For example, it means that the direct runoff caused by 2 in. of effective rainfall in a given time interval is four times as great as that caused by 0.5 in. of effective rainfall in the same amount of time. It also means that the total amount of time for the basin to respond to each of these rainfall depths is the same. Consequently, the base length, in hours, of each of the direct runoff hydrographs is the same.

Visualizing and Developing Unit Hydrographs

By definition, the Δt -h (or Δt -min) unit hydrograph of a drainage basin is the direct runoff hydrograph produced by 1 in. of effective rainfall falling uniformly in time and space over the drainage basin during a duration Δt time units. By conservation of mass, the volume of runoff produced (represented by the area under a graph of the unit hydrograph) must be equal to 1 in. of runoff (effective rainfall) times the drainage basin area (see Figure 2.17). It may be observed from this definition that the unit hydrographs for two drainage basins must be different from one another, because the basin areas (as well as other factors) are different.

Figure 2.18 provides a metaphor to help demonstrate the abstract principals of a unit hydrograph. A sprinkler system is uniformly distributed over an entire drainage area. Constant rainfall is modeled by using the sprinklers to apply a constant rate of water until exactly 1 in of runoff is generated, at which time

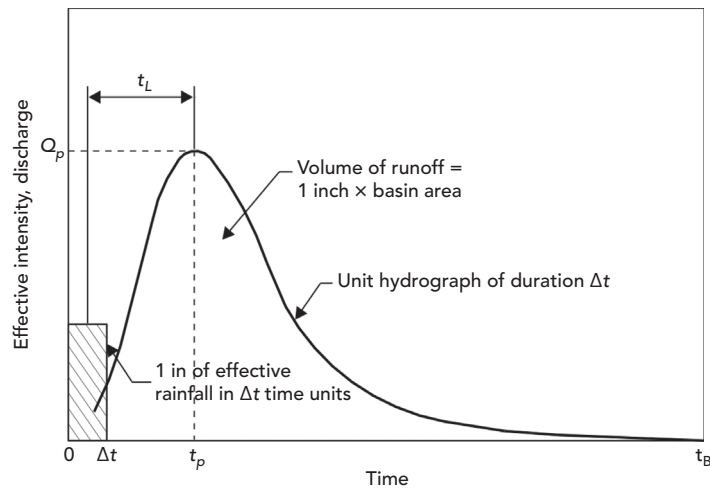


Figure 2.17: Unit hydrograph resulting from 1 in. of effective rainfall (runoff) over time (Δt).

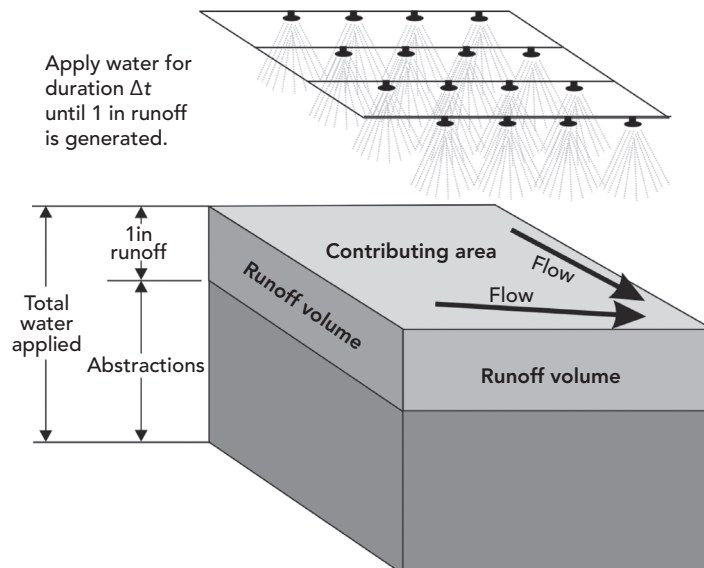


Figure 2.18: Visualizing the unit hydrograph concept.

the sprinklers are turned off. Δt represents the duration of time that the sprinklers are on. A stream gauge is set on the downstream end of the contributing drainage area to record the runoff hydrograph resulting from the 1 in of runoff. The resulting hydrograph represents the *unit hydrograph* generated for the drainage area.

The time t_L in Figure 2.17, which is the length of time from the center of Δt to the peak discharge on the unit hydrograph, is the **basin lag time**. This response time can be thought of as an approximate average of all possible travel times for runoff in a drainage basin. In practice (and as shown in the figure), it is usually assumed to be the amount of time between the center of mass of a pulse of effective rainfall and the peak of the resultant hydrograph. The basin lag time is often used instead of time of concentration when estimating the complete runoff hydrograph.

Two basic categories of unit hydrographs exist: (1) unit hydrographs for gauged watersheds and (2) **synthetic unit hydrographs**. Most commonly, rainfall and runoff records for a drainage basin do not exist, and one must resort to synthesis of a unit hydrograph based on information that can be gathered about the basin. Extensive literature exists on various ways to calculate synthetic unit hydrographs, including procedures proposed by Clark, Snyder, and Singh. This text introduces the development of synthetic unit hydrographs by applying the NRCS (SCS) method. Further details on these and other synthetic unit hydrographs methods can be found in *Stormwater Conveyance Modeling and Design* (Haestad Methods and R. Durrans, 2003)

NRCS Synthetic Unit Hydrographs

The NRCS analyzed a large number of unit hydrographs derived from rainfall and runoff records for a wide range of basins and basin locations and developed an average **dimensionless unit hydrograph** (SCS, 1969). Times on the horizontal axis of the dimensionless unit hydrograph are expressed in terms of the ratio of time to time of peak discharge (t/t_p), and discharges on the vertical axis are expressed in terms of the ratio of discharge to peak discharge (Q/Q_p). Figure 2.19 shows the NRCS dimensionless unit hydrograph, and Table 2.12 lists its ordinates.

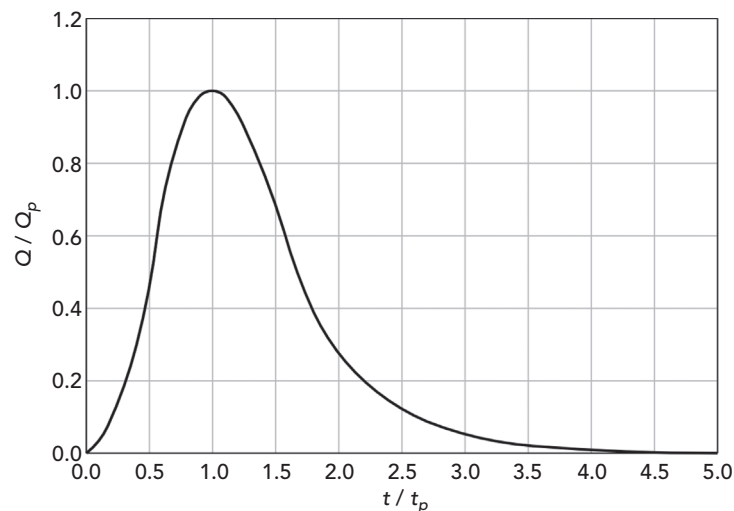


Figure 2.19: The NRCS (SCS) dimensionless unit hydrograph.

Table 2.12: Ordinates of the SCS (NRCS)
Dimensionless Unit Hydrograph (SCS, 1969)

t/t_p	Q/Q_p	t/t_p	Q/Q_p
0	0.000	1.7	0.460
0.1	0.030	1.8	0.390
0.2	0.100	1.9	0.330
0.3	0.190	2.0	0.280
0.4	0.310	2.2	0.207
0.5	0.470	2.4	0.147
0.6	0.660	2.6	0.107
0.7	0.820	2.8	0.077
0.8	0.930	3.0	0.055
0.9	0.990	3.2	0.040
1.0	1.000	3.4	0.029
1.1	0.990	3.6	0.021
1.2	0.930	3.8	0.015
1.3	0.860	4.0	0.011
1.4	0.780	4.5	0.005
1.5	0.680	5.0	0.000
1.6	0.560		

Application of the dimensionless unit hydrograph method involves estimation of the lag time t_L of the drainage basin. Lag is typically approximated as $0.6t_c$ in NRCS (SCS) unit hydrograph computations. The time to peak of the synthetic unit hydrograph of duration Δt is then computed as

$$t_p = \frac{\Delta t}{2} + t_L$$

The NRCS recommends that Δt should be equal to $0.133t_c$, or equal to $0.222t_L$ (SCS, 1969). A small variation from this is acceptable. The Δt chosen for development of the synthetic unit hydrograph must be consistent with the Δt chosen for development of the design storm and effective rainfall hyetographs.

The peak discharge Q_p (in cfs) of the synthetic unit hydrograph is calculated as

$$Q_p = \frac{645.33KAQ}{t_p}$$

where

645.33 = conversion factor = $(1 \text{ h}/3,600 \text{ s}) (1 \text{ ft}/12 \text{ in}) [(5,280 \text{ ft})^2/1 \text{ mi}^2]$;

$K = 0.75$ is constant based on geometric shape of dimensionless unit hydrograph (NEH-4);

$Q = 1 \text{ in.}$ is the runoff depth for unit hydrograph calculation;

A = the drainage basin area (mi^2); and

t_p = the time to peak (h).

Simplifying yields

$$Q_p = \frac{484A}{t_p}$$

The coefficient 484 appearing in the numerator of this equation includes a unit conversion factor and is an average value for many drainage basins. It may be reduced to about 300 for flat or swampy basins, or increased to about 600 for steep or mountainous basins. Care needs to be taken when the factor 484 is changed, as the base length and/or shape of the synthetic unit hydrograph must also be changed to ensure that it represents a volume of water equivalent to one inch of effective rainfall over the drainage basin area.

Once t_p and Q_p have been estimated using the preceding equations, the desired synthetic unit hydrograph can be graphed and/or tabulated using the dimensionless unit hydrograph shown in Figure 2.19 and Table 2.12.

Example 2.13: Developing an NRCS (SCS) Synthetic Unit Hydrograph

Develop a synthetic unit hydrograph for a 1.5-mi^2 drainage basin in Memphis, Tennessee having a time of concentration of 90 min. Assume that the basin slopes are moderate so that the factor 484 can be applied in computing Q_p .

Solution

Employing the guidelines for estimation of Δt , its value should be no larger than

$$0.133t_c = 0.133(90) = 12 \text{ min}$$

A duration of $\Delta t = 10 \text{ min}$ is selected, and the synthetic unit hydrograph will be a 10-min unit hydrograph. The basin lag is estimated as

$$t_L = 0.6t_c = 54 \text{ min}$$

Table 2.13: SCS (NRCS) Synthetic Unit Hydrograph Computations for Example 2.12

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)
t (min)	t/t_p	Q/Q_p	Q (cfs)	t (min)	t/t_p	Q/Q_p	Q (cfs)
0	0.00	0.00	0	140	2.37	0.16	118
10	0.17	0.08	59	150	2.54	0.12	89
20	0.34	0.24	178	160	2.71	0.09	67
30	0.51	0.49	363	170	2.88	0.07	52
40	0.68	0.79	585	180	3.05	0.05	37
50	0.85	0.96	710	190	3.22	0.04	30
60	1.02	1.00	740	200	3.39	0.03	22
70	1.19	0.93	688	210	3.56	0.02	15
80	1.36	0.81	599	220	3.73	0.02	15
90	1.53	0.64	474	230	3.90	0.01	7
100	1.69	0.47	348	240	4.07	0.01	7
110	1.86	0.35	259	250	4.24	0.01	7
120	2.03	0.26	192	260	4.41	0.01	7
130	2.20	0.21	155	270	4.58	0.00	0

The time to peak for the synthetic unit hydrograph is

$$t_p = (10/2) + 54 = 59 \text{ min} = 0.98 \text{ h}$$

The peak discharge is estimated as

$$Q_p = 484(1.5)/0.98 = 740 \text{ cfs.}$$

Ordinates of the synthetic unit hydrograph are given in Table 2.13. The first column of the table is the time t , in minutes, and is tabulated in $\Delta t = 10$ min intervals. The second column is the dimensionless time ratio t/t_p , where $t_p = 59$ min. The third column is the dimensionless discharge ratio and is determined using the dimensionless time ratio and interpolation from Table 2.12. The fourth and last column contains the ordinates of the 10-min unit hydrograph, which are computed as the products of the dimensionless discharge ratios and $Q_p = 740$ cfs (Figure 2.20).

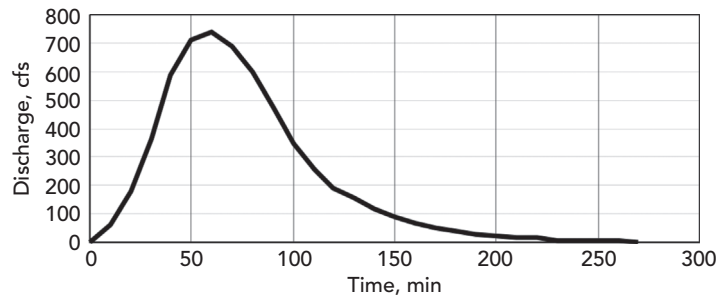


Figure 2.20: Synthetic unit hydrograph for Example 2.13.

Discrete Convolution

A unit hydrograph represents the runoff hydrograph from a drainage basin subjected to 1 in. of runoff (effective precipitation) applied over a duration of Δt . However, a runoff (effective rainfall) hyetograph typically contains many Δt time periods, each of which has its own associated runoff depth. In **discrete convolution**, the direct runoff hydrograph resulting from a complete rainfall hyetograph is computed by applying a unit hydrograph to each discrete time step within the hyetograph.

To conceptualize discrete convolution, refer to Figures 2.21 and 2.22. The first figure illustrates a runoff (effective rainfall) hyetograph consisting of $n_p = 3$ rainfall pulses, each of duration $\Delta t = 10$ min, with the depth of each pulse denoted by P_i ($i = 1, 2, \dots, n_p$). The second figure shows a unit hydrograph with $n_u = 11$ nonzero ordinates shown at Δt time intervals, with the nonzero ordinates denoted by U_j ($j = 1, 2, \dots, n_u$). (Note that the ordinates of the unit hydrograph are in units of cfs per inch of runoff, and thus the notation of this subsection is different from the previous subsection in which unit hydrograph ordinates were denoted by discharge Q .) The time increment Δt used for development of the runoff (effective rainfall) hyetograph must be the same as the Δt duration of excess rainfall used to create the unit hydrograph.

Because unit hydrograph theory assumes that drainage basins behave linearly, the ordinates Q_j of the runoff hydrograph produced by P_1 inches of effective rainfall must be equal to $Q_j = P_1 U_j$. These ordinates are shown in the third column of Table 2.14 (the first and second columns of the table reproduce the information in Figure 2.22). These ordinates were calculated by multiplying each unit hydrograph ordinate by the runoff depth for pulse $P_1 = 0.2$ in. The values in column three are shifted down such that the time at which the runoff hydrograph in column three begins is the same as the time at which the first pulse of effective rainfall begins.

The fourth and fifth columns in Table 2.14 are computed by multiplying the unit hydrograph ordinates in the second column by the effective rainfall depths

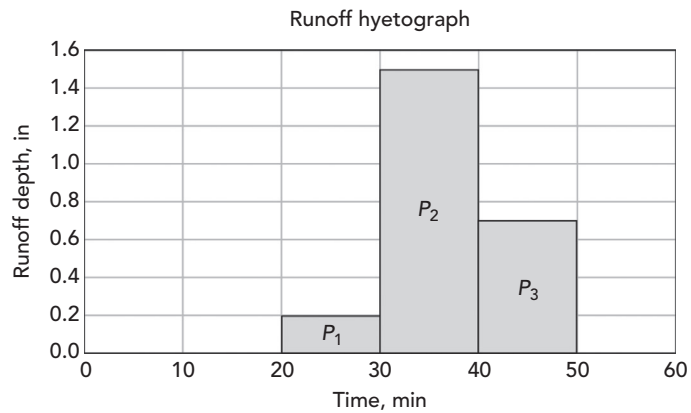


Figure 2.21: Effective rainfall (runoff) hyetograph.

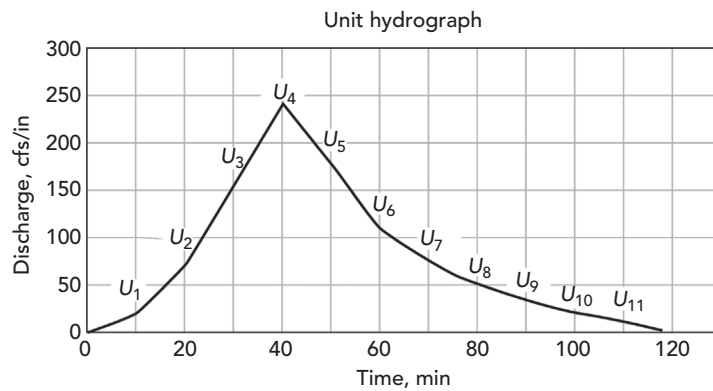


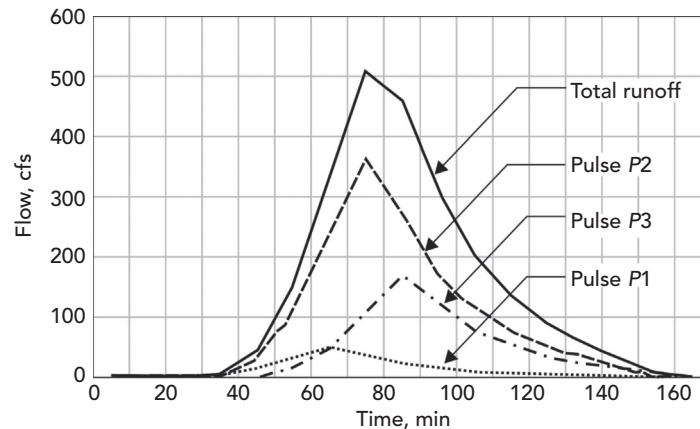
Figure 2.22: Unit hydrograph ordinates.

Table 2.14: Discrete Convolution

(1)	(2)	(3)	(4)	(5)	(6)
t (min)	U (cfs/in)	$P_1 U$ (cfs)	$P_2 U$ (cfs)	$P_3 U$ (cfs)	Q (cfs)
0	0				0
10	20				0
20	70	0			0
30	160	4	0		4
40	240	14	30	0	44
50	180	32	105	14	151
60	110	48	240	49	337

Table 2.14: (Continued)

(1)	(2)	(3)	(4)	(5)	(6)
t (min)	U (cfs/in)	$P_1 U$ (cfs)	$P_2 U$ (cfs)	$P_3 U$ (cfs)	Q (cfs)
70	75	36	360	112	508
80	50	22	270	168	460
90	33	15	165	126	306
100	21	10	113	77	200
110	10	7	75	53	134
120	0	4	50	35	89
130		2	32	23	57
140		0	15	15	30
150			0	7	7
160				0	0


Figure 2.23: Summing of hydrographs from individual rainfall pulses.

$P_2 = 1.5$ in and $P_3 = 0.7$ in, respectively. Again, the time at which each runoff pulse's hydrograph begins corresponds to the time at which each runoff (effective rainfall) pulse begins. The sum of the runoff hydrograph ordinates in each row of the table (shown in the sixth column) is an ordinate of the direct runoff hydrograph caused by the complete rainfall event.

Figure 2.23 shows the runoff hydrograph resulting from each rainfall pulse and the total runoff hydrograph for the entire rainfall event.

Example 2.14: Computing a Runoff Hydrograph

Compute the direct runoff hydrograph for the Memphis, Tennessee drainage basin described in Example 2.13. Use the unit hydrograph developed for

the basin in that example, and use the runoff (effective rainfall) hyetograph developed in Example 2.12.

Solution

There are a total of $n_p = 4$ runoff (effective rainfall) pulses, and a total of $n_u = 26$ nonzero unit hydrograph ordinates. Thus there will be a total of $n_q = 29$ nonzero direct runoff hydrograph ordinates.

Table 2.15 illustrates the tabular calculation (discrete convolution). The first column is the time since the beginning of rainfall in $\Delta t = 10$ -min increments

Table 2.15: Discrete Convolution to Obtain Direct Runoff Hydrograph

(1)	(2)	(3)	(4)	(5)	(6)	(7)
t (min)	U (cfs/in)	$0.99U$	$0.45U$	$0.25U$	$0.09U$	Q (cfs)
0	0					0
10	59					0
20	178	0				0
30	363	58	0			58
40	585	176	27	0		203
50	710	359	80	15	0	454
60	740	579	163	45	5	792
70	688	703	263	91	16	1,073
80	599	733	320	146	33	1,231
90	474	681	333	178	53	1,244
100	348	593	310	185	64	1,152
110	259	469	270	172	67	977
120	192	345	213	150	62	769
130	155	256	157	119	54	585
140	118	190	117	87	43	436
150	89	153	86	65	31	336
160	67	117	70	48	23	258
170	52	88	53	39	17	197
180	37	66	40	30	14	150
190	30	51	30	22	11	115
200	22	37	23	17	8	85
210	15	30	17	13	6	65
220	15	22	14	9	5	49
230	7	15	10	8	3	36

Table 2.15: (Continued)

(1)	(2)	(3)	(4)	(5)	(6)	(7)
t (min)	U (cfs/in)	$0.99U$	$0.45U$	$0.25U$	$0.09U$	Q (cfs)
240	7	15	7	6	3	30
250	7	7	7	4	2	19
260	7	7	3	4	1	15
270	0	7	3	2	1	13
280		7	3	2	1	12
290		0	3	2	1	6
300			0	2	1	2
310				0	1	1
320					0	0

(the duration of the unit hydrograph and the duration of the effective rainfall pulses). The second column contains the ordinates of the unit hydrograph from Example 2.13. From the third to the sixth columns are the unit hydrograph ordinates multiplied by the effective rainfall depths. Note that the first entry (the first zero) in each column corresponds to the time at which the corresponding effective rainfall pulse begins ($t = 20$ min for the first pulse, $t = 30$ min for the second pulse, and $t = 40$ min for the third pulse; see Figure 2.16). The seventh column is the sum of the previous four and is the direct runoff hydrograph.

2.5 Problems

1. The following data is from Huff and Angle (1992). The numbers indicate the total rainfall (in) expected in Chicago, Illinois for storm recurrence intervals of 2, 5, 10, 25, 50, and 100 years and durations from 5 min to 24 h. Plot a series of IDF curves showing the rainfall intensities for each recurrence interval for storm durations between 5 min and 2 h.

Duration	Recurrence Interval (Years)					
	2	5	10	25	50	100
24 h	3.11	3.95	4.63	5.60	5.63	7.36
2 h	1.83	2.33	2.74	3.31	3.86	4.47
1 h	1.46	1.86	2.18	2.63	3.07	3.51
30 min	1.15	1.46	1.71	2.07	2.42	2.77
15 min	0.84	1.07	1.25	1.51	1.76	1.99
10 min	0.68	0.87	1.02	1.23	1.44	1.62
5 min	0.37	0.47	0.56	0.67	0.78	0.89

2. Use the data in Problem 1 and the SCS Type II distribution to develop a 25-year, 24-h storm for Chicago.
3. During a 40-mm rainfall event, the interception capacity of the watershed is estimated to be 11 mm, the depression storage is 9 mm, and the infiltration is 7 mm. What is the total volume of runoff from the 2.56 ha watershed?
4. A proposed development consists of the following land use areas:

Land Use	Area (ac)
Condominiums	2.75
Greenbelt	0.89
Commercial	1.37

The time of concentration is estimated to be 25 min. Using the Rational Method IDF data from Problem 1, calculate the peak runoff resulting from a 10-year storm. Use the midpoint in the range of C coefficients in Table 2.5.

5. A suburban watershed has a time of concentration of 2.5 h and the following land use characteristics:

Land Cover	Area (mi ²)	CN_{II}
Commercial and business	4	92
1/4 ac housing	14	75
Parking lots, roofs, and driveways	0.78	98

- (a) Use the NRCS (SCS) CN method to determine volume of runoff resulting from a 6-in. storm. Assume that AMC-I applies.
 - (b) Use the SCS method to determine the peak discharge from the watershed.
6. Develop a unit hydrograph for a drainage basin with an area of 3.5 mi². Use a time of concentration of 120 min. Assume that a shape factor of 484 applies and that the time of concentration is 2 h.

3

Inlets, Gravity Piping Systems, and Storm Sewer Design

3.1 Inlet Design Overview

Road and highway designs must include sufficient drainage provisions in order to minimize the danger resulting from stormwater runoff and to optimize traffic efficiency under most weather conditions. The key objective when designing inlets is to minimize the spread of water across a roadway and in the gutter. In stormwater drainage, the **gutter** is the channel on the side of the road through which stormwater runoff is conveyed to storm sewer inlets. **Spread** is the top width of the flowing water on the road, measured from the curb.

The allowable spread length, which is generally determined by local or state regulations, is based on the classification of the road. For example, a road with a higher speed limit should have a smaller allowable spread than a road designed for slower speeds because of the increased risk of hydroplaning. In addition to spread width, roadway classification also dictates the return period of the design storm to use in calculating the spread at a point. Table 3.1, from the

Table 3.1: Suggested Minimum Design Frequencies and Spreads for Gutter Sections on Grade

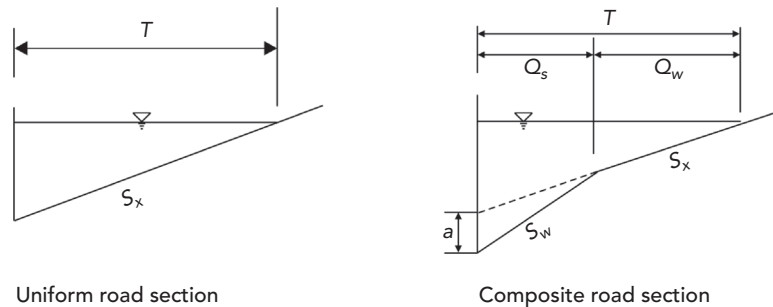
Road Classification		Design Return Period (Year)	Design Spreads
High volume or bidirectional	<70 km/h (45 mph)	10	Shoulder + 1 m (3 ft)
	>70 km/h (45 mph)	10	Shoulder
	Sag point	50	Shoulder + 1 m (3 ft)
Collector	Low volume	10	1/2 Driving lane
	High volume	10	Shoulder
	Sag point	10	1/2 Driving lane
Local streets	Low volume	5	1/2 Driving lane
	High volume	10	1/2 Driving lane
	Sag point	10	1/2 Driving lane

Federal Highway Administration's (FHWA) HEC-22 *Urban Drainage Design Manual* (Brown et al., 2009), provides an overview of different road conditions and the design criteria these conditions necessitate. We recommend you refer to the FlowMaster or StormCAD online help for further explanations, as well as the HEC-22 manual.

The incoming surface flow (and spread) observed can be controlled by the efficiency and spacing of the inlets located upstream along the road. One additional factor to consider is whether the inlet is located on grade or in a sag, as the design criteria and the equations involved differ. **Inlets on grade** are located on a slope and intercept a portion of the water as it flows past. **Inlets in sag** are located at a point where runoff from a given area will ultimately collect, and these inlets are normally designed to capture 100% of the surface flow; otherwise, flooding will occur in the surrounding area.

3.2 Gutter Sections on Grade

The main curb and gutter section types are the uniform section and the composite section, as illustrated in Figure 3.1 with their defining variables. **Uniform gutter sections** have a constant slope across the section. **Composite gutter sections** are defined by a continuous gutter depression, a , measured from the bottom of the curb to the projected road cross-slope at the curb. The frontal flow, Q_w , is normally defined as the flow in the depressed section of the gutter or the flow over the grate width in the case of a grate inlet (grate inlets are discussed later in this chapter).



- Q_w = Flow in depressed section (m^3/s , ft^3/s)
- Q_s = Side flow (m^3/s , ft^3/s)
- S_w = Gutter cross-slope (m/m , ft/ft)
- S_x = Road cross-slope (m/m , ft/ft)
- T = Total width of flow or spread (m , ft)
- a = Continuous gutter depression (mm , in)

Figure 3.1: Uniform and composite gutter sections.

Inlets, Gravity Piping Systems, and Storm Sewer Design

Gutter Sections on Grade

Runoff in gutters on grade is treated as open channel flow, and Manning's equation is applicable. Because the friction along the curb height is negligible compared to the friction against the pavement width (spread) of flow, Manning's formula was modified, becoming the following equation for calculating flow in uniform gutter sections:

$$Q = \frac{K_c}{n} S_x^{1.67} S_L^{0.5} T^{2.67}$$

where

$K_c = 0.376$ (0.56 in U.S. units),

n = Manning's coefficient, and

S_L = road longitudinal slope (m/m, ft/ft).

Using this equation, the spread, T , in a uniform gutter section can be explicitly calculated for a given flow rate, Q . However, in the case of composite gutters, T can no longer be expressed as an explicit function of Q . Rather, an iterative process is required to calculate the spread.

On a road with a grade, the spread will be at a maximum just upstream of the inlet. Note that the spread at this location is independent of the inlet's ability to capture flow and its efficiency. The spread for a specific roadway is a function of the discharge in the gutter. To decrease the flow to an inlet, reduce inlet spacing so that they serve as collection points for smaller watersheds.

For composite sections, the variable E_0 is introduced to account for the added conveyance of a depressed gutter. E_0 is the ratio of flow in the depressed section, Q_w , to the total gutter flow, and is expressed as:

$$E_0 = \left\{ 1 + \frac{S_w}{S_x} \left[\left(1 + \frac{S_w/S_x}{(T/W) - 1} \right)^{2.67} - 1 \right] \right\}^{-1}$$

In the case of a grate inlet in a uniform section, the variable E_0 is also useful, with the frontal flow, Q_w , now defined as the gutter flow contained in the width of the grate. In this case, E_0 becomes:

$$E_0 = 1 - \left(1 - \frac{W_g}{T} \right)^{2.67}$$

where W_g = width of grate inlet (m, ft).

Drainage inlets typically capture frontal flow more efficiently than side flow. In drainage inlet design, it is good practice to maximize E_0 by increasing the gutter cross-slope, or by increasing the width of the gutter depression or the grate extending into the road.

3.3 Inlets on Grade

Designs of inlets on grade are based on how much flow will be intercepted for a given total flow (gutter discharge) to the inlet. Inlet design equations solve for this efficiency.

$$E = \frac{Q_i}{Q}$$

where

E = inlet efficiency,

Q_i = intercepted flow (m³/s, cfs), and

Q = total gutter flow (m³/s, cfs).

Flow that is not intercepted by a drainage inlet on grade is bypassed and carried over to another inlet downstream or is “lost” to a stream or pond, for example.

Grate Inlets on Grade

Grate inlets, as shown in Figure 3.2, tend to be more efficient than curb inlets when on a grade. Curb inlets are discussed in the next section of this chapter.

Two concerns must be addressed in the grate inlet design process. First, the engineer needs to choose a grate type appropriate for the roadway being designed. Bicycle traffic, for example, would limit the engineer to grate types with both longitudinal and transverse bars in order to prevent bicycle accidents. Second,

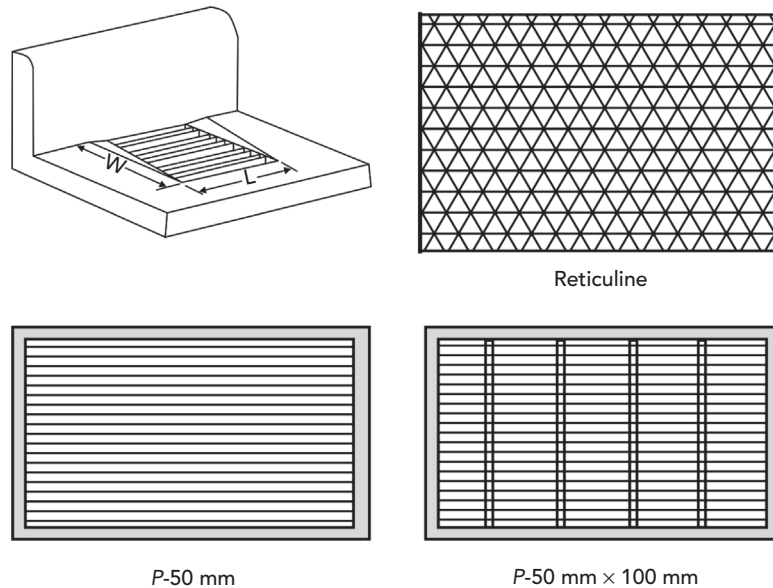


Figure 3.2: Grate inlet in a gutter and some typical grate types.

Inlets, Gravity Piping Systems, and Storm Sewer Design

Inlets on Grade

grate inlets have a higher propensity to clog than other types of inlets. If debris is prevalent in runoff at the point of design, adequate provisions must be made to account for inlet clogging, such as utilizing a combination (grate and curb) inlet at these points.

As shown in Figure 3.1, the total gutter flow is composed of frontal flow, Q_w (the flow in the depressed gutter or over the grate width), and side flow, Q_s . The total efficiency of the grate inlet is determined by calculating the grate's ability to capture frontal flow and side flow.

R_f is the ratio of intercepted frontal flow to the total frontal flow and is expressed as:

$$R_f = 1 - K_{cf}(V - V_0)$$

where

$K_{cf} = 0.295$ (0.090 in U.S. units),

V = average velocity in the gutter at the location of the inlet (m/s, ft/s), and

V_0 = splash-over velocity of the inlet (m/s, ft/s).

The **splash-over velocity** is the minimum velocity of the gutter flow capable of inducing enough momentum for some of the flow to skip over the grate opening and be carried over downstream. The splash-over velocity is a function of the grate type and the grate length and can be found in Appendix A of the HEC-22 manual (Brown et al., 2009) or obtained from the grate manufacturer. If the gutter velocity is less than the splash-over velocity, all frontal flow is intercepted and R_f equals 1.0.

The ratio, R_s , of side flow intercepted to total side flow is expressed as:

$$R_s = \left(1 + \frac{K_{cs} V^{1.8}}{S_x L^{2.3}} \right)^{-1}$$

where

$K_{cs} = 0.0828$ (0.15 in U.S. units) and

L = grate length (m, ft).

The total intercepted flow is expressed as:

$$Q_i = Q_w R_f + Q_s R_s$$

The total efficiency of the grate inlet on grade is expressed as:

$$E = R_f E_0 + R_s (1 - E_0)$$

Curb Inlets on Grade

Curb inlets are openings within the curb itself (see Figure 3.3), and they are used in areas where grate inlets are prone to clogging.

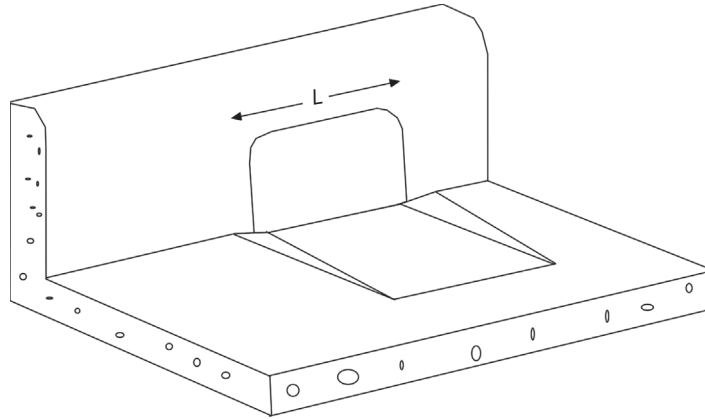


Figure 3.3: Curb inlet.

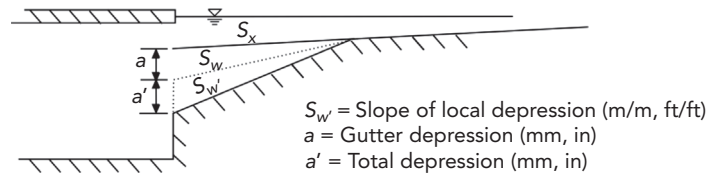


Figure 3.4: Local depression at a curb inlet.

The efficiency of a curb inlet is based on the ratio of the actual inlet length to the inlet length necessary to capture 100% of the total runoff. Curb inlets are often inset into the pavement to create a **local depression**. A local depression, as shown in Figure 3.4, is a depression of the gutter at the location of the inlet only, as opposed to a gutter depression, which is continuous along the curb.

The curb opening length, L_T , that would be required to intercept 100% of a flow, Q , on a roadway section with a uniform cross-slope is defined as:

$$L_T = K_C Q^{0.42} S_L^{0.3} \left(\frac{1}{n S_x} \right)^{0.6}$$

where $K_C = 0.817$ (0.60 in U.S. units).

To account for local depressions or gutter depressions, an additional composite or equivalent slope, S_e , is necessary:

$$S_e = S_x + S_w' E_0$$

Inlets, Gravity Piping Systems, and Storm Sewer Design

Inlets on Grade

where

S_w' = gutter cross-slope at the inlet location measured from the pavement cross-slope, S_x (m/m, ft/ft) and

E_0 = ratio of flow in the depressed section to the total gutter flow upstream of the inlet (does not account for local depression).

To calculate L_T with a composite gutter, replace the road cross-slope, S_x , with the equivalent slope, S_e , in the equation solving for L_T .

The efficiency of a curb opening on grade shorter than the required length for total interception is:

$$E = 1 - \left(1 - \frac{L}{L_T}\right)^{1.8}$$

Combination Inlets on Grade

Combination inlets, shown in Figure 3.5, consist of both grate and curb openings. The curb inlet functions as a sweep, removing debris from the runoff before it can clog the grate inlet.

If the curb inlet length is equal to the grate inlet length, the flow intercepted by the combination inlet is assumed to be equivalent to that intercepted by the grate inlet alone.

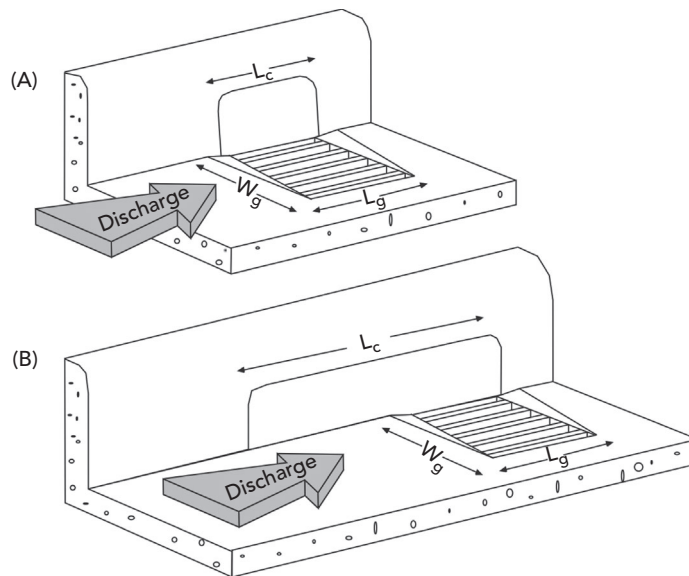


Figure 3.5: Typical combination inlets.

The curb inlet is often extended upstream of the grate for more efficient removal of debris than when the curb opening length equals the grate length. The total flow intercepted by this configuration is calculated as the sum of

- the flow intercepted by the portion of the curb opening located upstream of the grate and
- the flow that bypassed the upstream curb opening and is intercepted by the grate alone. (the flow intercepted by the portion of the curb opening adjacent to the grate is neglected.)

3.4 Inlets in Sag

Inlets located in sag are assumed to capture 100% of flow because, once collected, the runoff in the sag has no other place to go. As opposed to inlets located on a grade, the size and type of inlet directly affects the spread. As shown in Table 3.1, the HEC-22 manual typically recommends employing a larger rainfall return period for designing an inlet located in sag than for designing an inlet located on grade.

The computations for calculating the amount of flow intercepted by inlets in sag are based on the principles of weir flow and orifice flow discussed in Chapter 1.

For an unsubmerged inlet operating as a weir, the flow capacity is calculated as:

$$Q_{iw} = C_w P d^{1.5}$$

where

Q_{iw} = flow intercepted by the inlet operating as a weir (m^3/s , ft^3/s);

C_w = weir coefficient, which varies depending on the flow condition and inlet structure;

P = perimeter of the inlet (m, ft); and

d = flow depth at the curb (m, ft).

Note that for a grate inlet, the perimeter does not include the length along the curb. Also, if the gutter is depressed (locally or continuously), the perimeter, P , of the grate is calculated as:

$$P = L + 1.8W$$

where

L = grate length (m, ft) and

W = grate width (m, ft).

The depth, d , for both types of inlets is measured from the projected pavement cross-slope. For a curb inlet, the perimeter is equivalent to the length of the inlet.

Inlets, Gravity Piping Systems, and Storm Sewer Design

Inlets in Sag

If the inlet is submerged and is operating as an orifice, its capacity becomes:

$$Q_{io} = C_o A (2g d_o)^{0.5}$$

where

Q_{io} = flow intercepted by the inlet operating as an orifice (m^3/s , ft^3/s),

C_o = orifice coefficient (varies based on the class of inlet and its configuration),

A = area of the opening (m^2 , ft^2),

$g = 9.81 \text{ m/s}^2$ (32.16 ft/s^2 in U.S. customary units), and

d_o = effective head at the orifice (m, ft).

Note that for a grate inlet, the effective head, d_o , is simply the water depth along the curb. For a curb inlet, the effective head is expressed as:

$$d_o = d_i - \frac{h}{2}(\sin \theta)$$

where

d_i = depth at lip of curb opening (m, ft),

h = curb throat opening height (m, ft) (see Figure 3.6), and

θ = inclination of the curb throat measured from the vertical direction as shown in Figure 3.6.

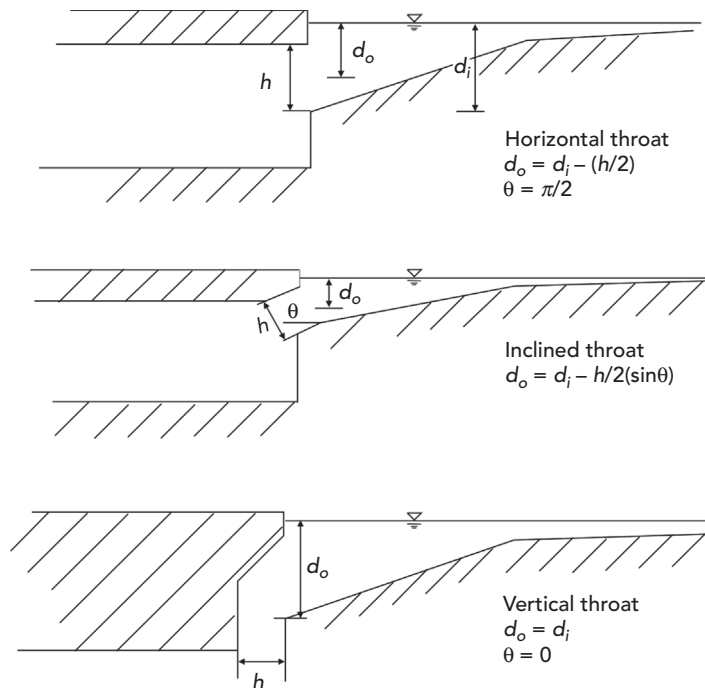


Figure 3.6: Different curb inlet throat types.

Grates alone are not typically recommended for installation in sags because of their propensity to clog and exacerbate ponding during severe weather. A combination inlet may be a better alternative. At low flow depths, the capacity of a combination inlet for which the grate inlet length equals the curb inlet length is equivalent to the capacity of the grate inlet alone. At higher flow depths for the same type of inlet, both the curb inlet and grate inlet act as orifices. The total intercepted flow is then calculated as the sum of the flows intercepted by the grate and curb openings.

3.5 Inlet Design Overview

This section provides a quick overview of features necessary to solve inlet problems using software from Bentley Systems. FlowMaster can be used to analyze the hydraulic performance of individual inlets. StormCAD is used to model inlets in storm sewer networks.

Inlet and Gutter Problems Using FlowMaster

FlowMaster can compute the capacities of gutters and drainage inlets using the methodology put forth in the FHWA's HEC-22 *Urban Drainage Design Manual* (Brown et al., 2009). Given a type of inlet and design constraints, it can also compute the inlet size required. With FlowMaster you can solve for different variables associated with the following structures by selecting them from the list of available worksheets:

- gutter sections on grade
- grate inlets
- curb inlets
- combination inlets
- slot inlets
- ditch inlets

For a more in-depth tutorial on the use of FlowMaster, see Chapter 1.

When creating a new inlet worksheet, you will be required to name the worksheet by providing a label. Additionally, you will need to specify whether the inlet is on grade or in sag by selecting the appropriate radio button. When editing an inlet's worksheet, remember that there are multiple tabs on which you have to enter data. For example, the combination inlet worksheet has a **Gutter** tab, an **Inlet** tab, a **Grate** tab, and a **Curb** tab.

In the worksheet for a gutter section, you can provide a discharge and solve for the resulting spread, or you can enter a spread and have FlowMaster solve for the allowable discharge. In the worksheet for an inlet, you can either solve for an inlet's efficiency based upon the entered parameters, or you can size the inlet to meet specified design criteria.

Inlets, Gravity Piping Systems, and Storm Sewer Design

Gradually Varied Flow

Set the variable you want to solve for by selecting it from the pull-down menu in the **Solve For:** field. Then, click **Solve**. The calculated values will appear in the fields highlighted in yellow.

Inlet and Gutter Network Problems Using StormCAD

StormCAD, like FlowMaster, calculates and designs drainage inlets by applying the HEC-22 methodology. Unlike FlowMaster, StormCAD will design an entire gutter network as opposed to just one inlet at a time. The gutter network dictates what happens to flow on the surface when it bypasses an inlet and is routed downstream to another inlet.

To edit an inlet's properties, first double-click on that inlet in the drawing pane and select the **Inlet** tab at the top of the dialog. In the **Inlet** field, select an inlet from the pull-down menu, which lists all the inlets currently defined in the **Inlet Library**. To examine the characteristics of the inlet selected, click the ellipsis (...) button next to this field. The **Inlet Library** dialog will appear. The Inlet Library is a separately saved file and editable library allowing customization for local regulations. Select the inlet of interest and click **Edit...** to examine or change its properties, or click the **Insert** button to create a new inlet. To exit an inlet's properties dialog in the library, click **OK** or **Cancel**. To exit the **Inlet Library**, click **Close**.

In the **Inlet** tab of an **Inlet Editor** dialog, the **Inlet Opening** section displays variables for the type of inlet chosen. Gutter characteristics can be entered under **Inlet Section**.

Finally, you have to specify an inlet location by clicking either the **On Grade** or **In Sag** radio button. If the inlet is on grade, fill in the **Longitudinal Slope** of the road and **Manning's n** -value for the pavement. You must also select a **Bypass Target** to establish the connectivity of the gutter network. The **bypass target** is another inlet or outlet in the same project (the target element does not have to be an element in the same pipe network) where the flows bypassing the current inlet will be carried. If no bypass target is chosen, all bypass flow will automatically carry over to the outlet of the sewer network that the inlet is part of (in this case, the bypassed flow is not accounted for in the sewer network). Click **OK** to exit the dialog. To view a graphical representation of the gutter network in plan view, select **Gutter Network** from the **Tools** pull-down menu.

3.6 Gradually Varied Flow

When performing an open channel hydraulic analysis, the change in water depth along the length of a section may be so gradual that the entire section can be assumed to have a constant water depth (such as normal depth) without any loss of accuracy. In other systems, however, such as storm sewers, there may be some sort of restriction that prevents the water depth from equaling normal

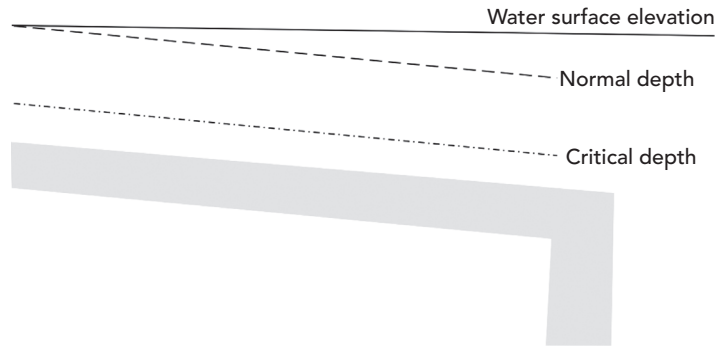


Figure 3.7: Nonuniform flow in an open channel.

depth throughout the length. For example, a high tailwater elevation may force the depth to be above normal depth at the downstream end of a pipe, as shown in Figure 3.7.

When the difference in water depth from one end of a conduit to the other is significant, a gradually varied flow profile analysis is required. Gradually varied flow profile analysis is the process of splitting a channel length into segments and analyzing each segment separately. It is based on the following several assumptions:

- The head loss within any given calculation segment is the same as for uniform flow conditions.
- The velocity is the same across the entire cross section.
- The slope of the conduit is less than 10%.
- The roughness coefficient is constant throughout the reach under consideration and is independent of the depth of flow.
- The depth of flow changes gradually along the length of the conduit, starting from some controlling boundary condition (usually the tailwater elevation); there are no sudden increases or decreases in depth.
- If the pipe is sufficiently long, the depth of flow will approach normal depth.

Flow Classification

The first step in performing a gradually varied flow analysis is to identify the flow classification that is expected to occur in the conduit based on the slope of the channel, the normal depth, the critical depth, and the controlling boundary condition.

Slope Classification

Once the normal depth and critical depth have been computed for the section, the conduit slope can be determined. If the normal depth is above critical

Inlets, Gravity Piping Systems, and Storm Sewer Design

Gradually Varied Flow

depth, the slope is said to be **mild**. If the normal depth is equal to the critical depth, the slope is said to be **critical**. If normal depth is below critical depth, the slope is said to be **steep**.

There are two other slope types to consider: a **horizontal** channel and a channel on an **adverse** slope (an “uphill”-sloping channel). For these slope types, normal depth is undefined.

For a gradually varied flow profile, the first letter of the slope type is used in the identification of the profile. For example, a channel with a hydraulically steep slope is a type S, a channel with a hydraulically mild slope is a type M, and so forth.

Flow Zone Classification

In a gradually varied flow analysis, the flow is assumed to stay within the same zone throughout the length of the conduit. For example, the flow will not jump from subcritical to supercritical within the same profile type. There are three zones where flow profiles can occur:

- Zone 1: Flow depth is above both normal and critical depths.
- Zone 2: Flow depth is between normal depth and critical depth.
- Zone 3: Flow depth is below both normal and critical depths.

A given profile will exist in only one of these zones. Because normal depth is undefined for horizontal slopes and adverse slopes, Zone 1 flow does not exist for these profile types.

Profile Classifications

Once the slope classification and flow zone have been determined, the profile type can be defined, and the engineer can determine how to proceed with the hydraulic grade computations. Figure 3.8 shows the basic profile types.

To perform the computations, the engineer must determine from the profile type whether the flow is subcritical or supercritical (based on the location of actual depths compared to critical depth). In order to prevent excessive velocities that could cause pipe scour or channel erosion, most storm sewers are designed with mild slopes to carry subcritical flows. For this type of flow, the hydraulic control is at the downstream end of the section, and the profile calculation proceeds toward the upstream end. When the flow depth is above normal depth (as in an M1 profile), this type of analysis is called a **backwater** analysis. When the flow depth is between critical depth and normal depth, it is called a **drawdown** analysis.

When supercritical flows are encountered, the controlling section is at the upstream end of the conduit, and the computations proceed from upstream to downstream. This calculation type is a **frontwater** analysis.

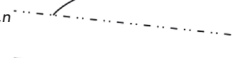
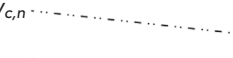
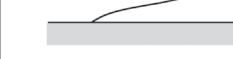
	Zone 1 profiles $y > y_n; y > y_c$	Zone 2 profiles $y_n \geq y \geq y_c$ or $y_n \leq y \leq y_c$	Zone 3 profiles $y < y_n; y < y_c$
Mild slope $y_n > y_c$	M1 	M2 	M3 
Critical slope $y_n = y_c$	C1 	C2 unstable 	C3 
Steep slope $y_n < y_c$	S1 	S2 	S3 
Horizontal slope	H1 none 	H2 	H3 
Adverse slope	A1 none 	A2 	A3 

Figure 3.8: Gradually varied flow classifications.

Energy Balance

Even for gradually varied flow, the solution is still a matter of balancing the total energy between the two ends of each segment. The energy equation as it relates to each end of a segment can be written as (note that the pressures for both ends are zero, because it is free surface flow):

$$Z_1 + \frac{V_1^2}{2g} = Z_2 + \frac{V_2^2}{2g} + H_L$$

where

Z_1 = hydraulic grade at the upstream end of the segment (m, ft),

V_1 = velocity at the upstream end (m/s, ft/s),

Z_2 = hydraulic grade at the downstream end of the segment (m, ft),

V_2 = velocity at the downstream end (m/s, ft/s),

H_L = head loss due to friction (other losses are assumed to be zero) (m, ft), and

g = gravitational acceleration constant (m/s², ft/s²).

The friction loss is computed based on the average rate of friction loss along the segment and the segment length. This relationship is:

$$H_L = S_{Avg} \Delta x = \frac{S_1 + S_2}{2} \Delta x$$

where

H_L = loss across the segment (m, ft),

S_{Avg} = average friction loss (m/m, ft/ft),

S_1 = friction slope at the upstream end of the segment (m/m, ft/ft),

S_2 = friction slope at the downstream end of the segment (m/m, ft/ft), and

Δx = length of the segment being analyzed (m, ft).

The conditions at one end of the segment are known (through assumption or from a previous calculation). Because the friction slope is a function of velocity, which is a function of depth, the depth at the other end of the segment can be found through iteration. There are two primary methods to obtain the solution: the standard-step method and the direct-step method.

Standard-Step Method

This method involves dividing the channel into segments of equal known lengths and solving for the unknown depth at one end of the segment (starting with a known or assumed depth at the other end). The standard-step method is the most popular method of determining the flow profile because it can be applied to any channel, not just prismatic channels.

Direct-Step Method

The direct-step method is based on the same basic energy principles as the standard-step method, but it takes a slightly different approach to the solution. Instead of assuming a segment length and solving for the depth at the end of the segment, the direct-step method assumes a depth and then solves for the segment length.

3.7 Mixed Flow Profiles

So far, only conditions for which the entire channel has the same profile type, resulting in the smooth curves of a gradually varied flow analysis, have been

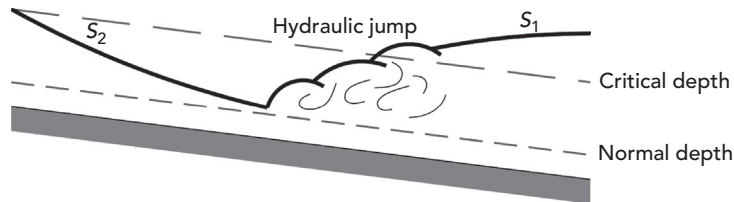


Figure 3.9: Plot of hydraulic jump.

discussed. This section explores those cases in which profile types are mixed within the same section and the steps that can be taken to analyze these occurrences.

Sealing Conditions

There may be conditions whereby part of a pipe or culvert section is flowing full, while part of the flow remains open. These conditions are called *sealing* conditions and are analyzed in separate parts. The portion of the section flowing full is analyzed as pressure flow, and the remaining portion is analyzed as gradually varied flow.

Rapidly Varied Flow

Rapidly varied flow is turbulent flow resulting from the abrupt and pronounced curvature of flow streamlines into or out of a hydraulic control. Examples of rapidly varied flow include hydraulic jumps, bends, and bridge contractions.

Hydraulic Jumps

When flow passes rapidly from supercritical to subcritical flow, a phenomenon called a *hydraulic jump* occurs. In storm sewer networks, this often occurs when a steep pipe discharges into a particularly high tailwater, as shown in Figure 3.9. There are significant losses associated with hydraulic jumps due to the amount of mixing and turbulence that occur. These forces are also highly erosive, so engineers typically try to prevent jumps from occurring in storm sewer systems, or at least predict the location of jumps in order to provide adequate channel, pipe, or structure protection.

3.8 Storm Sewer Applications

Storm sewer analysis occurs in two basic calculation sequences:

- **Hydrology:** Watersheds are analyzed and flows are accumulated from upstream inlets toward the system outlet.
- **Hydraulics:** A tailwater condition is assumed at the outlet, and the flow values (from the hydrology calculations) are used to compute hydraulic grades from the outlet toward the upstream inlets.

Hydrology Model

As the runoff from a storm event travels through a storm sewer, it combines with other flows, and the resulting flows are based on the overall watershed characteristics. As with a single watershed, the peak flow at any location within the storm sewer is assumed to occur when all parts of the watershed are contributing to the flow. Therefore the rainfall intensity that produces the largest peak flow at a given location is based on the controlling system time at that same location.

The controlling system time is the larger of the *local time of concentration* (to a single inlet) and the *total upstream system time* (including pipe travel times). The controlling time is used for computing the intensity (and therefore the flow) in the combined system.

Example 3.1: Flow Accumulation

A storm sewer inlet has a local time of concentration of 8 min for a watershed with a weighted *CA* (i.e., the weighted runoff coefficient times the total drainage area) of 1.23 ac. This inlet discharges through a pipe to another storm sewer inlet with a weighted *CA* of 0.84 ac and a local time of concentration of 9 min. If the travel time in the pipe is 2 min, what is the overall system *CA* and corresponding storm duration?

Solution

The total *CA* can be found by simply summing the *CA* values from the two inlets. The storm duration, however, must be found by comparing the local time of concentration at the second inlet to the total time for flow from the upstream inlet to reach the downstream inlet.

$$\text{Total } CA = 1.23 \text{ ac} + 0.84 \text{ ac} = 2.07 \text{ ac}$$

$$\text{Upstream time} = 8 \text{ min} + 2 \text{ min} = 10 \text{ min}$$

The total upstream flow time of 10 min is greater than the local time of concentration at the downstream inlet (9 min). The 10-min value is therefore the controlling time and should be used as the duration of the storm event.

Other Sources of Water

There may be a number of flow sources for a storm sewer system other than direct watershed inflow. Flow may be piped into an inlet from an external connection, or there could be flow entering an inlet that is the carryover (bypass) flow from another storm sewer inlet.

Design practices for handling carryover flows vary from jurisdiction to jurisdiction. Some local regulations may require that pipes be sized to include all flow that arrives at an inlet, whereas other locales may specify that pipes be sized to accommodate only the flow that is actually intercepted by the upstream inlets.

It is the responsibility of the design engineer to ensure that the design is in agreement with the local criteria and policies.

The hydraulics of a storm sewer are often computed as described previously in this chapter. Normal depth may occur in portions of the system, whereas other areas may experience pressure (submerged) conditions. Gradually varied flow and rapidly varied flow may also occur.

Computations start at the system outlet, where a tailwater condition is assumed. There are four basic assumptions for tailwater conditions:

- **Normal depth:** The depth at the outfall of the farthest downstream pipe is assumed to be equal to normal depth, as in a sufficiently long S2 profile.
- **Critical depth:** The depth at the pipe outfall is assumed to be critical depth, as in subcritical flow to a free discharge.
- **Crown elevation:** The depth is set to the crown (top) of the pipe for free outfall.
- **User-specified tailwater:** A fixed tailwater depth can also be used, as when there is a known pond or river water surface elevation at the outfall of the storm sewer.

Care should be taken to choose an accurate tailwater condition, because this value can affect the hydraulics of much of the system. The designer of a storm sewer system should consider the tailwater depth during storm conditions. An outlet may be above the receiving stream during dry weather but may be submerged during the design storm event.

3.9 StormCAD, SewerGEMS and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software. Information about SewerGEMS can be found in Chapter 9.

What Does StormCAD Do?

StormCAD is an extremely powerful, easy-to-use program that helps civil engineers design and analyze storm sewer systems. Just draw your network on the screen by using the tool palette, double-click any element to enter data, and click the **GO** button to calculate the network.

Rainfall information is calculated using rainfall tables, equations, or the National Weather Service's HYDRO-35 data. StormCAD also plots intensity–duration–frequency (IDF) curves. You have a choice of conveyance elements

Inlets, Gravity Piping Systems, and Storm Sewer Design

StormCAD, SewerGEMS and Tutorials

that include circular pipes, pipe arches, boxes, and more. Flow calculations handle pressure and varied flow situations, including hydraulic jumps, backwater, and drawdown curves. StormCAD's flexible reporting features allow you to customize and print the design and analysis results in report format or as a graphical plot.

How Can You Use StormCAD?

StormCAD is so flexible that you can use it for all phases of your project, from the feasibility report to the final design drawings and analysis of existing networks. During the feasibility phase, you can use StormCAD to create several different system layouts with an AutoCAD® or MicroStation™ drawing as the background. For the final design, StormCAD lets you complete detailed drawings with notes that can be used to develop construction plans. In summary, you can use StormCAD to:

- design multiple storm sewer systems with constraint-based design;
- design/analyze inlets based on HEC-22 methodology;
- use AASHTO, HEC-22 Energy, standard, absolute, or user-specified (generic) methods to compute structure losses;
- analyze various design scenarios for storm sewer systems;
- import and export AutoCAD® and MicroStation™ DXF files;
- predict rainfall runoff rates;
- generate professional-looking reports for clients and review agencies; and
- generate plan and profile plots of the network.

The theory and background used in StormCAD are presented in more detail in the StormCAD online help system.

Analysis and Design

StormCAD's automatic design feature allows you to design whole or part of a storm sewer system based on a set of user-defined design constraints. These constraints include minimum/maximum velocity, slope, and cover; choice of pipe invert or crown matching at structures; inlet efficiency; and gutter spread and depth. StormCAD will automatically design the invert elevations and diameters of pipes, as well as the size of a drainage inlet necessary to maintain a given spread (for inlets in sag) or capture efficiency (for inlets on grade).

Profiles

StormCAD also includes an option to automatically generate storm sewer **profiles**—longitudinal plots of the storm sewer. Profiles allow the design engineer, the reviewing agency, the contractor, and others to visualize the storm system. They

are useful for viewing the hydraulic grade line and determining if the proposed storm sewer is in conflict with other existing or proposed underground utilities.

3.10 Problems

Solve Problems 1–3 using FlowMaster and Problems 4–11 using StormCAD.

1. The spread on a proposed road paved with smooth asphalt and having a 1.0% longitudinal slope is limited by local regulations to 3.0 m for the sake of automobile safety. The road cross-slope is 2.5%.
 - (a) What is the maximum allowable discharge on the road if the gutter section is uniform? What is the depth of the flow measured at the curb for this rate of discharge?
 - (b) An alternative design proposes the addition of a gutter depression with a width of 1.5 m and a gutter cross-slope of 4.0%. What is the maximum allowable discharge in the gutter under these conditions? What is the depth of the flow at this rate of discharge?
 - (c) Does the addition of the gutter depression increase or decrease the capacity of the gutter? Why?
2. A proposed gutter section has a road cross-slope of 0.015 m/m and a gutter cross-slope of 0.045 m/m. The gutter is 1.20-m wide and is on a longitudinal slope of 0.02 m/m. The Manning's coefficient for the road is 0.013. The predetermined runoff in the gutter is $0.05 \text{ m}^3/\text{s}$.
 - (a) What is the efficiency of a P-50 mm $\times$ 100 mm grate inlet that is 0.70-m long and 0.5-m wide? Assume no clogging. What is the intercepted flow? What is the bypassed flow?
 - (b) What is the efficiency of a 0.70-m-long curb inlet on the same gutter section with no local depression? How does this efficiency compare to the efficiency of the grate inlet in part (a)?
 - (c) For the curb inlet in part (b), add a local depression that is 20-mm deep and 0.7-m wide. What is the efficiency of the inlet? How does this efficiency compare to the grate inlet in part (a) and the curb inlet in part (b)?
3. An inlet in sag must collect the 50-year peak runoff of 8.0 cfs. The gutter section has a road cross-slope of 2.0% and a gutter cross-slope of 4.0%. The gutter width is 3.0 ft. The local depression is 2.0-in. deep and 3.0-ft wide.
 - (a) What length combination inlet is necessary to maintain a spread of 8.0 ft at the section? The grate is a 3.0-ft wide, P-50-mm type, and the curb inlet has an opening height of 0.7 ft and a vertical throat configuration. Assume the length of the grate will equal the length of the curb opening.

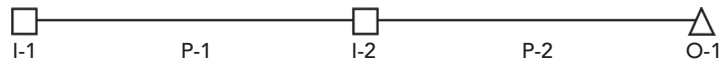
Inlets, Gravity Piping Systems, and Storm Sewer Design

Problems

- (b) How big would the curb inlet from part (a), acting alone, have to be to maintain the spread width of 8.0 ft under the same conditions? How long would the grate inlet in part (a) have to be to maintain the same spread? Which inlet type is the best solution?

Solve the following problems using StormCAD.

4. Lay out the storm sewer system shown below in StormCAD and enter the data for the network from the tables below. Calculate the results using Manning's equation and the 10-year storm event data tables that follow. Inlet I-1 is on grade with a longitudinal slope of 3%, whereas inlet I-2 is in sag. The tailwater condition at the outlet is free outfall. Assume that there is no clogging of the inlets.



Schematic for Problem 4

Pipe Data for Problem 4

Pipe	Circular Section Size (mm)	Upstream Invert Elevation (m)	Downstream Invert Elevation (m)	Length (m)	Pipe Material (n-value)
P-1	300	115.40	115.10	46	Concrete (0.013)
P-2	300	115.05	114.60	61	Concrete (0.013)

Node Data for Problem 4

Nodes	Ground Elevation (m)	Sump Elevation (m)	Head Loss Method	Head Loss Coefficient
I-1	117.40	115.40	Standard	0.5
I-2	117.15	115.05	Standard	0.5
O-1	118.00	114.60	—	—

Inlet Catchment Data for Problem 4

Inlet	Time of Concentration (min)	Additional Carryover (m ³ /s)	Area (ha)	Inlet C
I-1	6.3	0	0.45	0.75
I-2	5.2	0	0.22	0.80

Inlet Data for Problem 4

Inlet	Inlet Type	Grate Length (m)	Road Cross-Slope (m/m)	Bypass Target	Manning's n
I-1	Grate DI-1	1.1	0.02	I-2	0.012
I-2	Grate DI-1	1.1	0.02	–	–

Grate DI-1

(Provided in case missing from inlet library)

Structure width = 0.67 m

Structure length = 0.67 m

Grate Type = P-50 mm $\times$ 100 mm

Width = 0.76 m Standard Length = 0.76 m

Rainfall Data for Problem 4

Duration (min)	Rainfall Intensities (mm/h)		
	5 Years	10 Years	50 Years
5	69.8	78.7	99.6
10	54.6	61.0	77.5
20	40.6	45.2	56.6
30	31.7	36.2	45.7

Fill out an answer table like the one below for each of the following situations:

- Assume uniform gutters with a slope of 0.02 m/m.
- Assume continuously depressed gutters with a road cross-slope of 0.02 m/m, a gutter cross-slope of 0.04 m/m, and a gutter width of 0.8 m.
- Assume continuously depressed gutters as described in (b), as well as inlet lengths increased from 1.1 to 1.6 m.
- Explain the reasons for the differences between the three resulting tables.

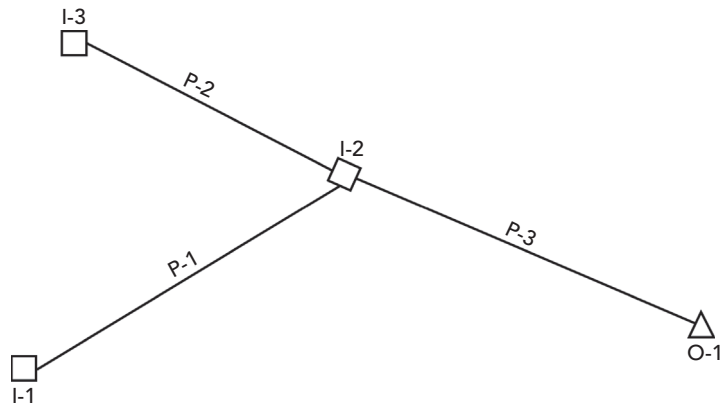
Answer Table for Problem 4

Inlet	Gutter Spread (m)	Total Flow to Inlet (m^3/s)	Intercepted Inlet Flow (m^3/s)	Bypassed Inlet Flow (m^3/s)	Efficiency (%)
I-1					
I-2					

- Enter and calculate the storm sewer network shown below using Manning's equation and the rainfall data used in Problem 4. The tailwater condition is free outfall. Assume no clogging of the inlets. Answer the questions that follow.

Inlets, Gravity Piping Systems, and Storm Sewer Design

Problems



Schematic for Problem 5

Pipe Data for Problem 5

Pipe	Circular Section Size (mm)	Upstream Invert Elevation (m)	Downstream Invert Elevation (m)	Length (m)	Pipe Material (<i>n</i> -value)
P-1	300	425.40	424.70	45	Concrete (0.013)
P-2	450	424.70	424.00	40	Concrete (0.013)
P-3	300	425.00	424.70	54	Concrete (0.013)

Node Data for Problem 5

Node	Ground Elevation (m)	Sump Elevation (m)
I-1	428.00	425.40
I-2	427.50	424.70
I-3	428.60	425.00
O-1	427.10	424.00

Inlet Data for Problem 5

Inlet	Inlet Type	Inlet Length (m)	Road Cross-Slope (m/m)	Inlet Location	Bypass Target	Long. Slope (m/m)	Manning's <i>n</i>
I-1	Curb DI-3A	1	0.02	On Grade	I-2	0.01	0.012
I-2	Curb DI-3A	1	0.02	In Sag	—	—	—
I-3	CurbDI-3A	1	0.02	In Sag	—	—	—

Inlet Catchment Data for Problem 5

Inlet	Time of Concentration (min)	Additional Carryover (m ³ /s)	Area (ha)	Inlet <i>C</i>
I-1	5.5	0	0.67	0.90
I-2	5.0	0	0.70	0.80
I-3	6.0	0	0.80	0.90

- (a) Given the design criteria shown in Table 3.1, what minimum gutter cross-slope should the 3-m-wide shoulders have (within 1%) on this high-volume, bidirectional road if the speed limit is 90 km/h?
- (b) As an alternative, set all gutter cross-slopes to 4% and replace the curb inlet at I-2 with a grate inlet called "Test." You will need to add this new inlet to the program's Inlet Library. The inlet has the characteristics given below. What is the minimum grate length for I-2 that would satisfy the 50-year design criteria shown in Table 3.1, assuming no clogging?

Test Inlet Data

Structure width and length	1.2 m
Grate type	P-50 mm
Grate width	1 m

Grate lengths start at 1 m and are available in increments of 0.5 m.

- (c) Using the same data and inlet designed in part (b), calculate the spread at I-2 assuming 50% clogging.

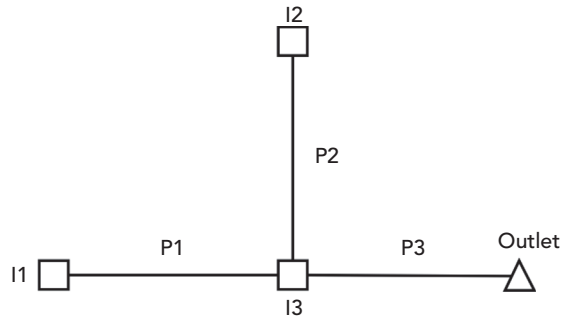
Solve the following problems using StormCAD. Assume all inlets are **Generic Default 100%**, which means that they are assumed to capture 100% of the surface flow.

6. The data that follows describes the existing storm sewer system shown below. For runoff calculations, assume $C = 0.3$ for pervious land cover and $C = 0.9$ for impervious cover. The ground elevation at the system discharge point is 17.0 m. All pipes are concrete ($n = 0.013$) and circular. Apply a standard head loss coefficient of 0.5 to inlet I-3.
 - (a) Analyze the system for a design return period of 10 years. Assume a free outfall condition. Provide output tables summarizing pipe flow conditions and hydraulic grade lines at the inlets. How is this system performing?
 - (b) Increase the size of pipe P-3 to 450 mm. Rerun the analysis and present the results. How does the system perform with this improvement?
 - (c) Local design regulations require that storm sewer systems handle 25-year return periods without flooding. Rerun the analysis for the improved system in part (b). Does the system meet this performance requirement?

Inlets, Gravity Piping Systems, and Storm Sewer Design

Problems

- (d) The above analyses are run using a default Manning's n of 0.013. Many drainage design manuals propose a less conservative design roughness of 0.012. Reanalyze the improved system under 25-year flows using $n = 0.012$. How does this change influence the predicted performance of the system?



Schematic for Problem 6

Rainfall Data for Problem 6

Duration (min)	Rainfall Intensity (mm/h)		
	5 Years	10 Years	25 Years
5	165	181	205
10	142	156	178
15	123	135	154
30	91	103	120
60	61	70	80

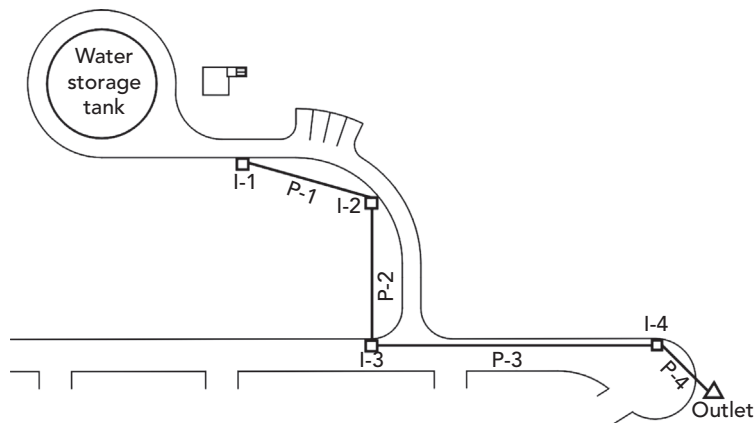
Inlet Information for Problem 6

Inlet	Ground Elevation (m)	Impervious Area (ha)	Pervious Area (ha)	Time of Concentration (min)
I-1	17.9	0.13	0.32	6.0
I-2	18.0	0.15	0.58	5.0
I-3	17.6	0.08	0.36	5.0

Pipe Information for Problem 6

Pipe	Upstream Invert (m)	Downstream Invert (m)	Diameter (mm)	Length (m)
P-1	16.7	16.15	300	56
P-2	16.8	16.1	375	46
P-3	16.1	15.3	375	54

7. You have been asked by the lead project engineer for a water supply utility to design the stormwater collection system for the proposed ground storage tank and pump station facility shown in the layout. Pipe lengths for P-1, P-2, P-3, and P-4 are 88, 92, 185, and 46 ft, respectively. See the CAD drawing and data for the system layout. Assume $C = 0.3$ for pervious areas and $C = 0.9$ for impervious areas.
- Using the StormCAD program's Automatic Design feature, size the system using the following design data. Use concrete pipe ($n = 0.013$) and the 25-year intensity–duration–frequency data provided in Problem 6 (Hint: StormCAD can mix SI and U.S. customary units). The top of bank elevation at the outfall ditch is 846.1 ft. The outfall pipe invert must be located at or above elevation 838.0 ft. Assume that the water surface elevation at the outfall is 842.0 ft, and that the pipes should have matching soffit (crown) elevations at every structure. Present your design in tabular form and provide a profile plot of your design.
 - During agency review, the county engineer requests that the water utility and the county work cooperatively to accommodate the planned construction of an elementary school nearby by increasing the size of the proposed storm system so that it can handle the design runoff from the school. The county engineer performs his own calculations and asks that you increase the size of pipes P-3 and P-4 to handle an external CA of 9.5 acres with a time of concentration of 12 min. Using StormCAD, introduce the additional flow at inlet I-3 and revise the facility design using the Automatic Design functionality of the program. Are all the design constraints met? What can you say about the flow conditions in pipe P-3 and pipe P-4? In the rest of the pipes?
 - If necessary, manually fine-tune and revise the design to meet all design criteria. Document your design as in part (a).



CAD Drawing for Problem 7

Inlet Information for Problem 7

Inlet	Ground Elevation (ft)	Impervious Area (ac)	Pervious Area (ac)	Time of Concentration (min)
I-1	848.9	0.25	0.25	6.0
I-2	848.3	0.09	–	5.0
I-3	847.0	0.20	0.49	5.0
I-4	846.5	0.18	0.5	7.5

Design Constraints for Problem 7

Constraint	Minimum	Maximum
Velocity	2 ft/s	15 ft/s
Cover	4 ft*	–
Pipe Slope	0.005 ft/ft	0.100 ft/ft

**Cover constraints may be relaxed at the outfall, where invert elevation requirements will typically govern.*

8. An inlet with a ground elevation of 260 m is connected to an outlet with a ground elevation of 259 m and a tailwater elevation of 256 m. The pipe connecting the inlet and outlet is a 20-m-long, concrete, circular pipe with a diameter of 525 mm and an upstream invert/sump elevation of 257 m. Print and compare pipe slope, average velocity, pipe flow time, capacity, energy slope, hydraulic grade line upstream, and flow profiles for each of the following conditions.

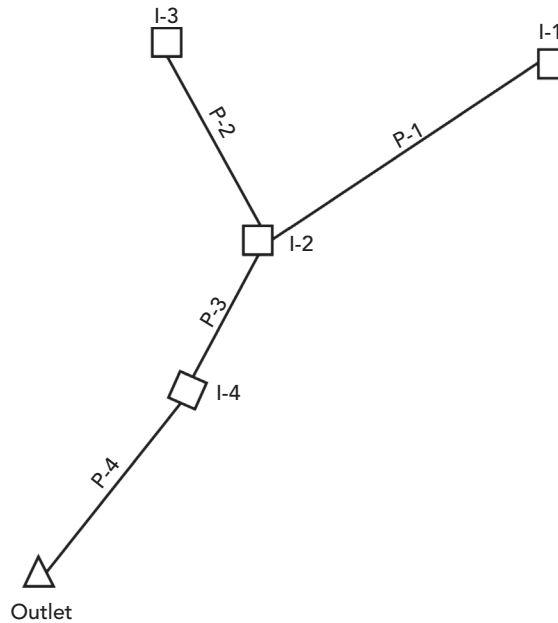
- (a) Scenario 1:
Inlet inflow = $0.3 \text{ m}^3/\text{s}$
Downstream invert/sump elevation = 256.9 m
- (b) Scenario 2:
Inlet inflow = $0.5 \text{ m}^3/\text{s}$
Downstream invert/sump elevation = 256.9 m
- (c) Scenario 3:
Inlet inflow = $0.5 \text{ m}^3/\text{s}$
Downstream invert/sump elevation = 256.7 m
- (d) Scenario 4:
Inlet inflow = $0.5 \text{ m}^3/\text{s}$
Downstream invert/sump elevation = 256.2 m

What are the differences between the four scenarios, and why do they occur?

9. The system described below drains into a river. Because the water surface elevation of the river varies, analyze the following system with different tailwater elevations. Use the Darcy–Weisbach equation because it is more appropriate for pressure flow than Manning’s equation. All pipes are concrete (roughness = 0.122 mm), circular pipes. The pipe and inlet data follow. The ground elevation at the outlet is 312.0 m.

What is the hydraulic grade at the entrance of inlet I-2 under the following tailwater conditions for the 10-year storm event?

- (a) Tailwater elevation = 310.0 m
- (b) Tailwater elevation = 311.0 m
- (c) Tailwater elevation = 313.0 m



Schematic for Problem 9

Rainfall Data for Problem 9

Duration (min)	Rainfall Intensity (mm/h)		
	2 years	10 years	100 years
5	80	130	210
15	65	97	145
30	45	68	100
60	25	37.5	55

Inlets, Gravity Piping Systems, and Storm Sewer Design

Problems

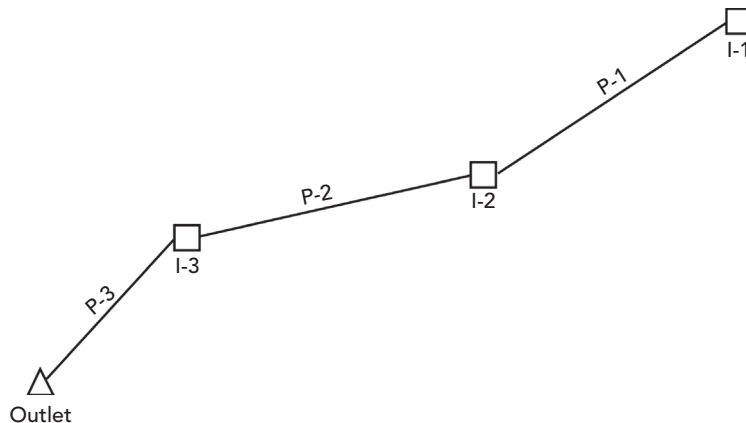
Pipe Information for Problem 9

Pipe	Length (m)	Diameter (mm)	Upstream Invert Elevation (m)	Downstream Invert Elevation (m)
P-1	23	300	312.4	311.5
P-2	25	300	311.5	311.0
P-3	32	525	311.0	310.5
P-4	17	525	310.5	310.4

Inlet Information for Problem 9

Inlet	Area ₁ (ha)	C ₁	Area ₂ (ha)	C ₂	t_c (min)	Ground Elev. (m)	Head Loss Coefficient (Standard Method)	Sump Elev. (m)
I-1	0.2	0.7	0.35	0.6	5.0	315.0	0.5	312.4
I-2	0.3	0.8	0.18	0.6	5.0	314.5	0.7	311.0
I-3	0.8	0.4	0.3	0.7	6.0	314.3	0.5	311.5
I-4	0.9	0.4	0.3	0.5	8.0	313.0	0.5	310.5

10. A schematic of an existing storm system for a residential subdivision is shown below. The rainfall, pipe, inlet, and hydrologic information are provided in the data tables. All pipes are circular, PVC pipes with Manning's $n = 0.010$.



Schematic for Problem 10

- (a) Analyze the system for a 25-year storm event. Assume a free outfall (critical depth) condition. Provide output tables summarizing pipe flow, velocity, and hydraulic grade at the upstream end of each pipe. How is this system performing?

- (b) If the existing system were constructed using CMP ($n = 0.024$) instead of PVC, how would the system perform in a 25-year storm event?
- (c) If the outfall were discharging to a pond that had a water surface elevation equal to that of the ground (using the original PVC pipes), how would the system perform in a 25-year storm? A 50-year storm?
- (d) Using the tailwater conditions specified in part (a), if the park were paved (with a $C = 0.9$ and time of concentration = 8 min), how would the system perform in a 25-year storm? In a 50-year storm?

Rainfall Data for Problem 10

Duration (min)	Rainfall Intensity (in./h)	
	25 years	50 years
5	4.00	4.60
15	2.80	3.20
30	1.80	2.00
60	0.75	0.90
100	0.55	0.65

Pipe Information for Problem 10

Pipe	Length (ft)	Diameter (in)	Upstream Invert Elevation (ft)	Downstream Invert Elevation (ft)
Pipe-1	70	18	166.70	165.30
Pipe-2	40	18	165.20	164.80
Pipe-3	53	24	164.80	164.40

Inlet Information for Problem 10

Inlet	Ground Elevation (ft)	Rim Elevation (ft)	Sump Elevation (ft)	Head Loss Coefficient (Standard Method)
1	172.00	172.00	166.50	0.5
2	172.00	172.00	165.00	0.5
3	168.00	168.00	164.50	0.5
Outfall	166.50	166.50	164.40	—

Inlets, Gravity Piping Systems, and Storm Sewer Design

Problems

Inlet Hydrologic Information for Problem 10

Inlet	Area (ac)	Area Type	Description	C	t_c (min)
1	8	Residential	Single Family	0.5	12
	1	Pavement	Asphalt/Concrete	0.7	
	0.04	Misc.	Playground	0.2	
2	1.2	Pavement	Asphalt/Concrete	0.7	5
3	10	Misc.	Park	0.2	15

11. Use the same network and data provided in part (a) of Problem 10. During a 25-year storm, it was determined that inlet 1 is not capable of capturing all of the flow produced by the runoff for its catch basin. A flow of 5 cfs is bypassed to inlet 2, which is in sag (all gutter flow is captured). Hint: You can apply negative and positive additional carryovers to inlets I-1 and I-2 under the **Catchment** tab.
- What is the gutter flow captured by inlet 1?
 - What is the gutter flow captured by inlet 2?
 - How is the system performing?
 - A flow monitoring study was performed and the following data was collected:

Pipe	25-Year Storm (cfs)	50-Year Storm (cfs)
Pipe-1	10	11.5
Pipe-2	17.3	19.9
Pipe-3	21.0	25.0

Analyze the system using these known flow values for both the 25- and 50-year storms. How is the system performing? Report the flows, upstream velocities, and hydraulic grade lines for each pipe.

4

Culvert Hydraulics

4.1 Culvert Systems

Culverts are commonplace in practical hydraulic design, including applications such as roadway crossings and detention pond outlets. A roadway cross-drainage culvert is typically designed to carry flows from one side of the road the other, without allowing the *headwater* (HW) (water surface elevation just upstream of the culvert) to exceed safe levels. When engineers analyze these culvert systems, they are usually trying to solve for one or more of the following:

- the size, shape, and number of new or additional culverts required to pass a design discharge;
- the hydraulic capacity of an existing culvert system under an allowable HW constraint;
- the upstream flood level at an existing culvert system resulting from a discharge rate of special interest; and
- hydraulic performance curves for a culvert system, which are used to assess hydraulic risk at a crossing or as input for another hydraulic or hydrologic model.

Similar to a storm sewer system, a culvert system consists of a hydrologic component, a culvert component, and a tailwater component. Although the watershed hydrology and tailwater conditions are almost identical to those that you would normally use in analyzing a storm sewer or other open channel transport system, there are additional hydraulic computations typically reserved for culvert analysis only.

In this chapter, procedures for designing culverts using CulvertMaster are presented. The use of nomographs as aids to hand calculations is described in detail in the Hydraulic Design of Highway Culverts manual (Normann et al., 2001, and Schall et al., 2012). Further discussion on hydraulic theory, end treatments, alignments, and roadway overtopping is presented by Haestad Methods and R. Durrans (2003)

Culvert Hydraulics

Obtaining accurate solutions in culvert hydraulics can be a formidable computational task. Culverts act as a significant constriction to flow and are subject to a range of flow types, including both gradually varied and rapidly varied flows.

It is this mix of flow conditions and the highly transitional nature of culvert hydraulics that makes the hydraulic solutions so difficult. For this reason, the documented approach is to simplify the hydraulics problem and analyze the culvert system using two different assumptions of flow control:

- **Outlet control** assumption—Computes the upstream HW depth using conventional hydraulic methodologies that consider the predominant losses due to the culvert barrel friction, as well as the minor entrance and exit losses. The tailwater condition during the design storm has an important effect on the culvert system.
- **Inlet control** assumption—Computes the upstream HW depth resulting from the constriction at the culvert entrance, while neglecting the culvert barrel friction, tailwater elevation, and other minor losses.

The controlling HW depth is the larger value of the computed inlet control and outlet control HW depths. Because the culvert system may operate under inlet control conditions for a range of flow rates and under outlet control conditions for another range, calculations must be performed for both control conditions.

4.2 Outlet Control Hydraulics

Calculations for outlet control HW depths are similar to those performed for storm sewer analysis. The HW depth is found by summing the tailwater depth, entrance minor loss, exit minor loss, and friction losses along the culvert barrel. The energy basis for solving the outlet control HW for a culvert is given by the basic energy equation and is presented graphically in Figure 4.1.

Full flow conditions (Cases A, B, and C in Figure 4.1) result in the same velocity (and therefore the same velocity head) throughout the length of the culvert; therefore the energy grade line and hydraulic grade line are parallel throughout the length of the barrel. At the culvert entrance, there is a slight increase in velocity and dip in the Hydraulic Grade Line (HGL) caused by the flow contraction that occurs there.

The energy equation can be rewritten specifically for culvert terms, which results in the following form:

$$HW_O + \frac{V_u^2}{2g} = TW + \frac{V_d^2}{2g} + H_L$$

where

HW_O = headwater depth above the outlet invert (m, ft);

V_u = approach velocity (m/s, ft/s);

TW = tailwater depth above the outlet invert (m, ft);

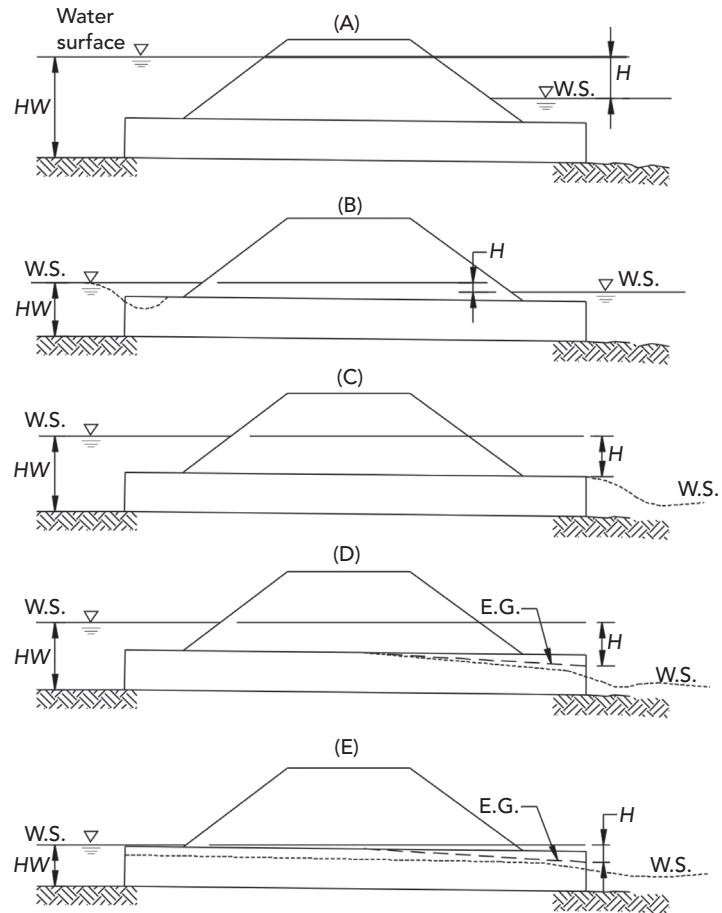


Figure 4.1: Outlet control flow conditions.

V_d = exit velocity (m/s, ft/s); and

H_L = sum of all losses, including the entrance minor loss (H_E), barrel friction losses (H_F), the exit loss (H_O), and other losses (m, ft).

Culverts often connect ponds or other bodies of water with negligible velocities, so the approach velocity and the velocity downstream of the culvert are often neglected, resulting in the following equation:

$$HW_O = TW + H_L$$

where

HW_O = headwater depth above the outlet invert (m, ft),

TW = tailwater depth above the outlet invert (m, ft), and

H_L = sum of all losses as listed above (m, ft).

Friction Losses

Culverts are frequently hydraulically short (entrance and exit losses exceed pipe friction losses), and uniform flow depths are not always achieved. For this reason, gradually varied flow methods are well suited to the analysis. For a more detailed description of gradually varied flow, see Chapters 3.6 and 9.4

Entrance Minor Loss

The entrance loss is caused by the contraction of flow as it enters the culvert and is a function of the barrel velocity head just inside the entrance. It is expressed by the following equation:

$$H_e = k_e \left(\frac{V^2}{2g} \right)$$

where

H_e = entrance loss (m, ft),

k_e = entrance loss coefficient,

V = velocity just inside the barrel entrance (m/s, ft/s), and

g = gravitational acceleration (m/s², ft/s²).

The entrance loss coefficient varies depending on the type of inlet that is present. The smoother the transition from the channel or pond into the culvert, the lower the loss of coefficient is. Values for the coefficients are presented in Table 4.1.

Table 4.1: Entrance Loss Coefficients (Normann et al., 2001)

Culvert Type	Entrance Type and Description	Entrance Loss Coefficient, k_e
Pipe, concrete	Projecting from fill, socket end (groove end)	0.2
	Projecting from fill, square cut end	0.5
	Headwall or headwall with wingwalls	
	Socket end of pipe (groove end)	0.2
	Square edge	0.5
	Rounded (radius = 1/12D)	0.2
	Mitered to conform to fill slope	0.7
	End-section conforming to fill slope*	0.5
	Beveled edges, 33.7° or 45° bevels	0.2
	Side- or slope-tapered inlet	0.2

Culvert Type	Entrance Type and Description	Entrance Loss Coefficient, k_e
Pipe or Pipe Arch, Corrugated Metal	Projecting from fill (no headwall)	0.9
	Headwall or headwall and wingwalls square edge	0.5
	Mitered to conform to fill slope, paved or unpaved slope	0.7
	End-section conforming to fill slope*	0.5
	Beveled edges, 33.7° or 45° bevels	0.2
	Side- or slope-tapered inlet	0.2
Box, Reinforced Concrete	Headwall parallel to embankment (no wingwalls)	
	Square-edged on 3 edges	0.5
	Rounded on 3 edges to radius of 1/12 bbl dimension	0.2
	Or beveled edges on 3 sides	
	Wingwalls at 30°–75° to barrel	
	Square-edged at crown	0.4
	Crown edge rounded to radius of 1/12 bbl dimension	0.2
	Or beveled top edge	
	Wingwalls at 10°–25° to barrel	
	Square-edged at crown	0.5
	Wingwalls parallel (extension of sides)	
	Square-edged at crown	0.7
	Side- or slope-tapered inlet	0.2

*Note: “End-section conforming to fill slope,” made of either metal or concrete, are the sections commonly available from manufacturers. From limited hydraulic tests, they are equivalent in operation to a headwall in both inlet and outlet control. Some end sections with a closed taper in their design have superior hydraulic performance.

Exit Minor Loss

The exit loss is an expansion loss, which is a function of the change in velocity head that occurs at the discharge end of the culvert. In culvert hydraulics, this sudden expansion loss is expressed as:

$$H_o = 1.0 \left[\frac{V^2}{2g} - \frac{V_d^2}{2g} \right]$$

where

V_d = velocity of the outfall channel (m/s, ft/s),

V = velocity just inside the end of the culvert barrel (m/s, ft/s), and

g = gravitational acceleration (m/s², ft/s²).

When the discharge velocity is negligible (as for a pond or slow-moving channel), the exit loss is equal to the barrel velocity head.

Gradually Varied Flow Analysis

Culverts operating under outlet control with unsubmerged outlets (Cases D and E in Figure 4.1) require an analysis of the hydraulic profile in order to determine the HW elevation. In Case D, the water surface reaches the crown of the culvert at some point upstream of the barrel. In both cases, the flow passes through critical depth at the outlet end of the barrel. The flow is represented by the M2 profile (see Figure 3.8).

4.3 Inlet Control Hydraulics

When a culvert is operating under inlet control conditions (as in Figure 4.2), the hydraulic control section is the culvert entrance itself. This means that the friction and minor losses within the culvert are not as significant as the losses caused by the entrance constriction.

Because the control section for culverts operating under inlet control conditions is at the upstream end of the culvert barrel, critical depth generally occurs at or near the inlet, and flows downstream of the inlet are supercritical. The hydraulic profile and outlet velocities are determined using frontwater gradually varied flow techniques.

Three types of inlet control hydraulics are in effect over a range of culvert discharges:

- **Unsubmerged (A and B)**—For low discharge conditions, the culvert entrance acts as a weir. The hydraulics of weir flow are governed by empirical working equations developed as a result of model tests.
- **Submerged (C)**—When the culvert entrance is fully submerged, it is assumed to be operating as an orifice.
- **Transitional (D)**—This flow type occurs in the poorly defined region just above the unsubmerged zone and below the fully submerged zone.

Unsubmerged Flow

There are two equations for unsubmerged (weir) flow. The first is based on the specific head at critical depth (with correction factors), whereas the second equation is more closely related to a weir equation. Although either equation will produce adequate results, the second equation is more commonly used during hand calculations because it is easier to apply.

Note that these equations were developed by the U.S. FHWA and as such are only intended for use in U.S. customary units. SI units should be converted before applying these equations.

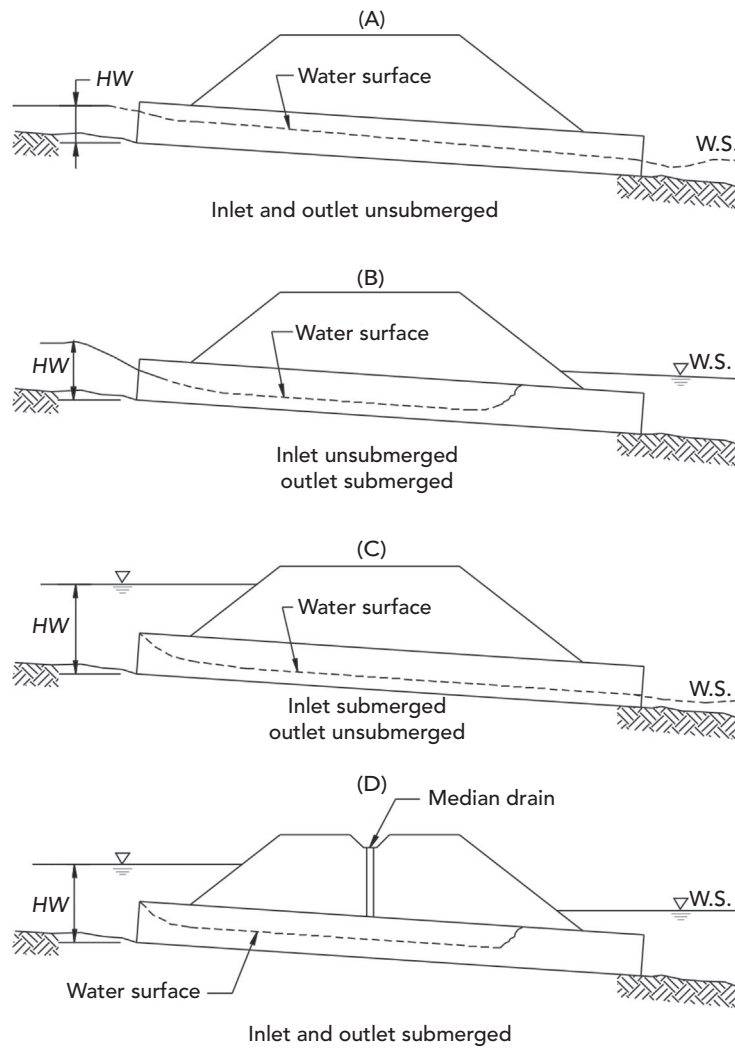


Figure 4.2: Inlet control flow conditions.

Unsubmerged, Form 1

$$\frac{HW_i}{D} = \frac{H_c}{D} + K \left[\frac{Q}{AD^{0.5}} \right]^M - 0.5S$$

Unsubmerged, Form 2

$$\frac{HW_i}{D} = K \left[\frac{Q}{AD^{0.5}} \right]^M$$

where

HW_i = headwater depth above the control section invert (ft);

D = interior height of the culvert barrel (ft);

H_c = specific head at critical depth, $y_c + V_c^2/2g$ (ft), where y_c = critical depth (ft) and V_c = velocity at critical depth (ft/s);

Q = culvert discharge (ft³/s);

A = full cross-sectional area of the culvert barrel (ft²);

S = culvert barrel slope (ft/ft); and

K, M = constants from Table 4.2.

These equations are applicable up to approximately $Q/AD^{0.5} = 3.5$. When using the first equation with mitered inlets, use a slope correction factor (last term of Form 1 equation) of $+0.7S$ instead of $-0.5S$.

Submerged Flow

The equation for submerged (orifice) flow is:

$$\frac{HW_i}{D} = c \left[\frac{Q}{AD^{0.5}} \right]^2 + Y - 0.5S$$

where

HW_i = headwater depth above the control section invert (ft),

D = interior height of the culvert barrel (ft),

H_c = specific head at critical depth, $y_c + V_c^2/2g$ (ft),

Q = culvert discharge (ft³/s),

A = full cross-sectional area of the culvert barrel (ft²),

S = culvert barrel slope (ft/ft), and

c, Y = constants from Table 4.2.

This equation for submerged flow is applicable above approximately $Q/AD^{0.5} = 4.0$. When using this equation with mitered inlets, use a slope correction factor (last term of equation) of $+0.7S$ instead of $-0.5S$.

Table 4.2: Coefficients for Inlet Control Design Equations

Shape and Material	Inlet-Edge Description	Equation Form	Unsubmerged		Submerged	
			K	M	c	Y
Circular Concrete	Square-edge w/headwall	1	0.0098	2.0	0.0398	0.67
	Groove end w/headwall		0.0018	2.0	0.0292	0.74
	Groove end projecting		0.0045	2.0	0.0317	0.69

Shape and Material	Inlet-Edge Description	Equation Form	Unsubmerged		Submerged	
			K	M	c	Y
Circular CMP	Headwall	1	0.0078	2.0	0.0379	0.69
	Mitered to slope		0.0210	1.33	0.0463	0.75
	Projecting		0.0340	1.50	0.0553	0.54
Circular	Beveled ring, 45° bevels	1	0.0018	2.50	0.0300	0.74
	Beveled ring, 33.7° bevels		0.0018	2.50	0.0243	0.83
Rectangular Box	30°–75° wingwall flares	1	0.026	1.0	0.0347	0.81
	90° and 15° wingwall flares		0.061	0.75	0.0400	0.80
	0° wingwall flares		0.061	0.75	0.0423	0.82
Rectangular Box	45° wingwall flare $d = 0.043D$	2	0.510	0.667	0.0309	0.80
	18°–33.7° wingwall flare $d = 0.083D$		0.486	0.667	0.0249	0.83
Rectangular Box	90° headwall w/3/4" chamfers	2	0.515	0.667	0.0375	0.79
	90° headwall w/45° bevels		0.495	0.667	0.0314	0.82
	90° headwall w/33.7° bevels		0.486	0.667	0.0252	0.865
Rectangular Box	3/4" chamfers; 45° skewed headwall	2	0.545	0.667	0.0505	0.73
	3/4" chamfers; 30° skewed headwall		0.533	0.667	0.0425	0.705
	3/4" chamfers; 15° skewed headwall		0.522	0.667	0.0402	0.68
	45° bevels; 10°–45° skewed headwall		0.498	0.667	0.0327	0.75
Rectangular Box 3/4" Chamfers	45° nonoffset wingwall flares	2	0.497	0.667	0.0339	0.803
	18.4° nonoffset wingwall flares		0.493	0.667	0.0361	0.806
	18.4° nonoffset wingwall flares					
	30° skewed barrel		0.495	0.667	0.0386	0.71
Rectangular Box Top Bevels	45° wingwall flares—offset	2	0.497	0.667	0.0302	0.835
	33.7° wingwall flares—offset		0.495	0.667	0.0252	0.881
	18.4° wingwall flares—offset		0.493	0.667	0.0227	0.887
CM Boxes	90° headwall	1	0.0083	2.0	0.0379	0.69
	Thick wall projecting		0.0145	1.75	0.0419	0.64
	Thin wall projecting		0.0340	1.5	0.0496	0.57
Horizontal Ellipse Concrete	Square edge w/headwall	1	0.0100	2.0	0.0398	0.67
	Groove end w/headwall		0.0018	2.5	0.0292	0.74
	Groove end projecting		0.0045	2.0	0.0317	0.69

Shape and Material	Inlet-Edge Description	Equation Form	Unsubmerged		Submerged	
			K	M	c	Y
Vertical	Square edge w/headwall	1	0.0100	2.0	0.0398	0.67
Ellipse	Groove end w/headwall		0.0018	2.5	0.0292	0.74
Concrete	Groove end projecting		0.0095	2.0	0.0317	0.69
Pipe Arch	90° headwall	1	0.0083	2.0	0.0379	0.69
18" Corner	Mitered to slope		0.0300	1.0	0.0463	0.75
Radius CM	Projecting		0.0340	1.5	0.0496	0.57
Pipe Arch	Projecting	1	0.0300	1.5	0.0496	0.57
18" Corner	No bevels		0.0088	2.0	0.0368	0.68
Radius CM	33.7° bevels		0.0030	2.0	0.0269	0.77
Pipe Arch	Projecting	1	0.0300	1.5	0.0496	0.57
31" Corner	No bevels		0.0088	2.0	0.0368	0.68
Radius CM	33.7° bevels		0.0030	2.0	0.0269	0.77
Arch CM	90° headwall	1	0.0083	2.0	0.0379	0.69
	Mitered to slope		0.0300	1.0	0.0463	0.75
	Thin wall projecting		0.0340	1.5	0.0496	0.57
Circular	Smooth tapered inlet throat	2	0.534	0.555	0.0196	0.90
	Rough tapered inlet throat		0.519	0.64	0.0210	0.90
Elliptical Inlet Face	Tapered inlet-beveled edges	2	0.536	0.622	0.0368	0.83
	Tapered inlet-square edges		0.5035	0.719	0.0478	0.80
	Tapered inlet-thin edge projecting		0.547	0.80	0.0598	0.75
Rectangular	Tapered inlet throat	2	0.475	0.667	0.0179	0.97
Rectangular	Side tapered—less favorable edges	2	0.56	0.667	0.0446	0.85
Concrete	Side tapered—more favorable edges		0.56	0.667	0.0378	0.87
Rectangular	Slope tapered—less favorable edges	2	0.50	0.667	0.0446	0.65
Concrete	Slope tapered—more favorable edges		0.50	0.667	0.0378	0.71

CMP is Corrugated Metal Pipe; CM is Corrugated Metal.

4.4 CulvertMaster and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of

hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software.

What Does CulvertMaster Do?

CulvertMaster is a program that helps civil engineers design and analyze culvert hydraulics. Just click a button to create a new worksheet, enter data in the clearly labeled fields, and click to calculate. You can solve for most hydraulic variables, including culvert size, flow, and HW. It also allows you to generate and plot rating tables and graphical output showing computed flow characteristics.

CulvertMaster contains options to automatically generate peak discharges, including the Rational Method and SCS Graphical Peak Discharge Method. You have a choice of culvert barrel shapes, including circular pipes, arches, boxes, and more. Calculations handle free surface, pressure, and varied flow situations, including backwater and drawdown curves. CulvertMaster's flexible reporting features allow you to print the results in report format or as a graphical plot.

The theory and background used in CulvertMaster are described in detail in this chapter, as well as in the CulvertMaster online help system.

How Can You Use CulvertMaster?

CulvertMaster can design or analyze culverts and compute HW for a range of flow rates. For a typical CulvertMaster project, an engineer may be interested in several culvert locations and try several designs for each location. You can use CulvertMaster to

- size culverts,
- compute and plot rating tables and curves,
- view output in both English and SI units, and
- generate professional-looking reports and graphs.

4.5 Problems

Solve the following problems using CulvertMaster's Quick Culvert Calculator.

1. A culvert is 11-m long and has upstream and downstream inverts of 263.4 and 263.1 m, respectively. The downstream tailwater elevation is below the downstream pipe invert.

For a K_c of 0.5 and a Manning's n value of 0.013, what minimum diameter concrete circular culvert (in mm) is required to pass $1.4 \text{ m}^3/\text{s}$ under a roadway with a maximum allowable headwater elevation of 265.2 m?

What is the headwater elevation for the selected culvert?

2. An existing 9.73-m-long, 560×420-mm steel and aluminum var CR arch ($n = 0.025$) has a 90-degree headwall. The inverts are 33.11 and 33.09 m. Assuming no tailwater effects, what is the maximum discharge that can pass through this culvert before the maximum allowable water surface elevation of 34.25 m is exceeded?
3. Twin 1,220×910-mm box culverts ($n = 0.013$, entrance has 90-degree and 15-degree wingwall flares) carry 8.5 m³/s along a 31-m length of pipe constructed at a 1.0% slope. The tailwater depth is 0.61 m.
 - (a) What is the headwater depth?
 - (b) Are the culverts flowing under inlet or outlet control conditions?
 - (c) What would the headwater depth and flow regime for a flow rate of 380 cfs?
4. A 100-ft horizontal concrete ellipse pipe ($n = 0.013$, $K_e = 0.5$) on a 3.5% slope is required to carry 65 cfs. Assume there is a free outfall. The maximum allowable headwater is 4.7 ft.
 - (a) What is the minimum pipe size (in in.) required?
 - (b) What would be the minimum size for a vertical concrete ellipse pipe?
5. A 12.2-m-long, 920 × 570-mm concrete arch pipe ($n = 0.013$, groove end with headwall entrance) constructed at a 0.8% slope carries 1.84 m³/s.
 - (a) If there is a constant tailwater depth of 0.3 m, what is the headwater depth for both inlet and outlet control conditions?
 - (b) Is the culvert flowing under inlet or outlet control conditions?
 - (c) What would be the result if the tailwater were 0.5 m deeper?
6. Triple 3,050 × 1,830-mm, 200-m-long, concrete box culverts ($n = 0.013$, $K_e = 0.5$) carry 110 m³/min. The culverts are constructed on a 1.4% slope and discharge into a pond with a depth 0.8 m above the downstream culvert invert. What is the exit velocity of the culverts?
7. Twin culverts are proposed to discharge 6.5 m³/s. The culverts will be 36.6-m long and have inverts of 20.1 and 19.8 m. The design engineer analyzed the three culvert systems described below. The tailwater elevation is free outfall. Which of the following proposed culverts will result in the highest headwater elevation? The lowest?
 - (a) 1,200-mm circular concrete pipes ($n = 0.013$ and $K_e = 0.5$)
 - (b) 1,220 × 910-mm concrete box culverts ($n = 0.013$, 90° and 15° wingwall flares at entrance)
 - (c) 1,630 × 1,120-mm steel and aluminum var CR arches ($n = 0.025$ and $K_e = 0.5$)

8. A 40-ft-long elliptical pipe ($n = 0.013$ and $K_e = 0.5$) will be constructed to carry 80 cfs with inverts of 22.6 and 22.1 ft. The tailwater is constant at an elevation of 24.0 ft. Which pipe will provide a lower headwater elevation: a 38×60 -in. horizontal ellipse or a 60×38 -in. vertical ellipse?
9. A circular concrete culvert has a free outfall. The culvert is 60-ft long on a 2% slope and is 30 in. in diameter. The culvert entrance will project from the embankment. Create a rating table that correlates the culvert discharge to the depth of the headwater, from 0 to 6 ft in 0.5-ft increments.
10. A detention pond drains through a circular concrete culvert that has a square-edge inlet with a headwall ($n = 0.013$, $K_e = 0.5$). The peak discharge from the pond is $4 \text{ m}^3/\text{s}$, and the headwater elevation is 15.5 m. The culvert is 10-m long on a 5% slope. The depth of the headwater is 4 m, and the tailwater elevation is below the elevation of the downstream invert. What is the elevation of the upstream invert? What size culvert is required?
11. A stream flows under a road through a 48-in. concrete circular culvert that has a square-edge inlet with a headwall ($n = 0.013$, $K_e = 0.5$). The culvert is on a 5% slope with a length of 31.2 ft, a downstream invert elevation of 92.5 ft, and a free tailwater outfall. The surface of the road is at an elevation of 108.3 ft, and the stream has a base flow rate of 100 cfs.
 - (a) If the runoff from a rainstorm adds another 200 cfs of flow to the stream, will the road be flooded?
 - (b) What is the maximum flow rate that will not flood the road?
12. A 15-m-long culvert that has a projecting groove-end inlet is needed on a 2% slope with a free outfall. The elevation of the upstream invert is 100 m, and the maximum headwater depth is 5 m.
 - (a) What size concrete arch culvert ($n = 0.013$, $K_e = 0.5$) is needed to carry $6 \text{ m}^3/\text{s}$ of water?
 - (b) Is the culvert operating under inlet or outlet control?
 - (c) If the tailwater depth changes to 0.3 m, what is the flow control?
 - (d) If the tailwater depth changes to 3.3 m, what is the flow control?
 - (e) Why does the flow control change or not change?
13. A 15-m-long concrete vertical ellipse culvert that has a square-edge inlet with a headwall ($860 \times 550 \text{ mm}$, $n = 0.013$, $K_e = 0.5$) has upstream and downstream invert elevations of 2.00 and 1.46 m, respectively. If the tailwater elevation is 3.02 m, at what flow rate (greater than $0.1 \text{ m}^3/\text{s}$) are inlet and outlet control headwater elevations equal? (Hint: Use rating curves.) Is this free-surface flow or pressurized flow?

5

Detention Pond Design

5.1 Overview of Stormwater Detention

Stormwater detention is the temporary storage of runoff in basins, underground containers, or on rooftops. Wet or dry **detention basins** (also called **detention ponds**) are the most commonly used structures and are the subject of this chapter. Once runoff has been collected in a detention basin, it can be released to downstream properties or conveyances in a controlled manner such that downstream flooding and other adverse impacts are prevented or alleviated. The construction of stormwater detention facilities is typically associated with new developments, as these usually result in increased stormwater runoff rates and volumes.

Figure 5.1 demonstrates the basic principles of detention pond analysis. An inflow runoff hydrograph (see Chapter 2) from one or more contributing drainage areas is directed to a storage facility (in this case, a graded detention pond). Runoff is then released from the facility at a controlled rate through a properly sized outlet structure such as a culvert. The result is a pond outflow hydrograph that is substantially flatter (i.e., has a lower peak flow) than the inflow hydrograph.

This chapter provides basic information on stormwater detention design and analysis concepts. The topics covered include basic detention facility types, pond volume calculations, outlet structure types, and the storage-indication method of pond routing. For more in-depth coverage of stormwater detention, the reader is referred to *Stormwater Conveyance Modeling and Design* (Haestad Methods and R. Durrans, 2003)

5.2 Basic Design Considerations

Design criteria for detention ponds vary widely. In the United States, detention regulations are typically adopted at the municipal or county level, so local regulations and guidelines must be consulted before the stormwater detention

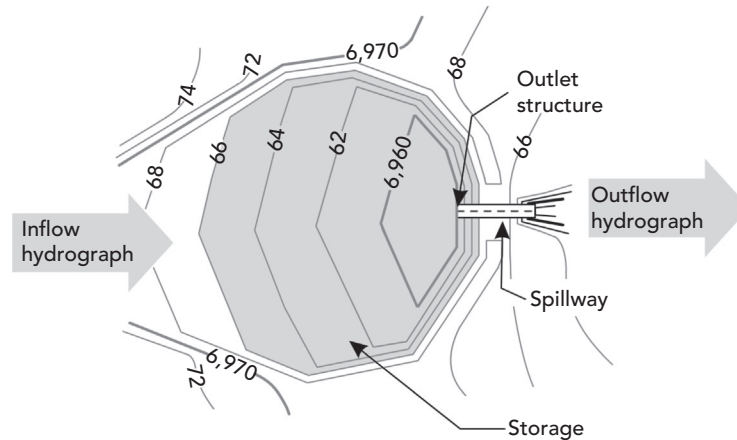


Figure 5.1: Conceptual drawing of a detention pond.

facility can be designed. Despite the wide variation in design practices, the underlying philosophy of most regulations is that peak runoff rates discharged from the undeveloped site for one or more design storm events should not be exceeded after the site development.

Several basic categories of detention facilities exist. Ponds may be wet or dry between storm events; detention may occur on the surface or underground; and a facility may lie in a main drainage path or be removed from it. The type of facility may be dictated by regulations or by the needs and limitations of the individual site or drainage area.

Predevelopment versus Postdevelopment Criteria and Recurrence Frequency

As a watershed is developed, the peak discharge rate and the amount of runoff will tend to increase, whereas the time to peak will typically decrease. The intent of stormwater detention is typically to limit postdevelopment peak discharges from a site to discharge rates no greater than those that occurred prior to development. Thus the conveyance capacities of existing downstream structures will still be sufficient if these structures were adequately sized prior to development.

Sometimes allowable detention facility discharges may be based on criteria other than predevelopment flows. For instance, if a structure downstream of the property is undersized, the developer may be required to actually lower peak discharge rates below predevelopment values so as not to exceed its capacity. It is important to note, however, that even if peak discharge objectives are met, detention ponds typically do not mitigate the increase in runoff *volume*

caused by land development. Although the peak flow rate from a site for a given storm event may be the same as it was prior to development, the duration of the discharge can be substantially longer.

The criteria described previously must be met for one or more specified design storm events. For most localities, these design events will be specified in their regulations. For example, the regulations may specify that postdevelopment peak flows be less than predevelopment peaks for 24-h, synthetic storms having return frequencies of 2, 5, and 10 years. (It is advisable, however, to check the design for storms of different durations.) The regulations may further state that an emergency overflow structure capable of handling flows from the 100-year storm be constructed. The magnitude of the largest storm that the facility must be designed to accommodate sometimes depends on factors such as site area and land use.

Types and Configurations of Stormwater Detention Facilities

A ***dry pond*** has an outlet that is positioned at or below the lowest elevation in the pond, such that the pond drains completely between storm events. A ***wet pond***, on the other hand, is one, the outlet of which is at an elevation above the pond bottom. Water remains in the pond during dry-weather periods between storm events and this water can only be depleted by infiltration into the soils and by evaporation from the water surface. A ***retention pond*** is a type of wet pond in which no outlet structure exists; infiltration and evaporation are the only means by which water can escape.

Detention ponds can be designed in an almost unlimited number of shapes and sizes. An ***aboveground pond*** usually consists of a depressed or excavated area with an earthen berm. In cases where there is not ample surface area available to meet storage requirements, ***underground detention*** may be necessary. Underground detention may consist of a series of large pipes or prefabricated custom chambers manufactured specifically for underground detention.

Online and off-line ponds are distinguished from one another based on whether they are positioned along the alignment of the main stormwater drainage path. An ***online pond***, as its name suggests, is positioned along the pathway and all runoff must pass through it. An ***off-line pond*** is located outside of the main drainage pathway, and only a portion of the total contributing stormwater runoff is diverted through it.

Multiple ponds may be necessary to achieve detention requirements under certain site conditions. Ponds can be connected in series such that the outflow hydrograph from one pond is also the inflow hydrograph for the next pond downstream. In some cases, particularly where the topography is flatter, the ponds may be close enough in elevation that the downstream pond stage has tailwater effects on the rating curve for the upstream pond. This type of

calculation is called interconnected pond modeling and is beyond the scope of this chapter.

5.3 Detention Pond Modeling Concepts

Stormwater detention design requires estimation of a complete runoff hydrograph, as opposed to the single peak discharge typically used in storm sewer design. Procedures for inflow hydrograph estimation were presented in Chapter 2. This section provides a conceptual review of stormwater runoff hydrographs and discusses how they are shaped and altered by added detention facilities.

Figure 5.2 serves as a definition sketch for the basic hydrograph concepts associated with detention. The ***inflow hydrograph*** represents the runoff from a watershed. Its principal attributes are the peak discharge rate (in ft^3/s or m^3/s) and the time from the beginning of the storm to the peak discharge. The area under this curve represents the total volume of runoff resulting from this storm event.

The ***outflow hydrograph*** represents the discharge from a detention facility. The outflow hydrograph is obtained by performing a ***routing analysis*** (discussed later in this chapter) for a given detention pond, outlet configuration, and inflow hydrograph. The reduction in the peak discharge seen in the outflow hydrograph is referred to as ***attenuation***, and the increase in the time to peak discharge is referred to as ***lag***.

If the infiltration and evaporation occurring in the pond during the storm event are negligible, the areas under the two curves (i.e., the runoff volumes) must be equal. The area between the inflow and outflow hydrographs represents the maximum volume of water present in the pond during the storm

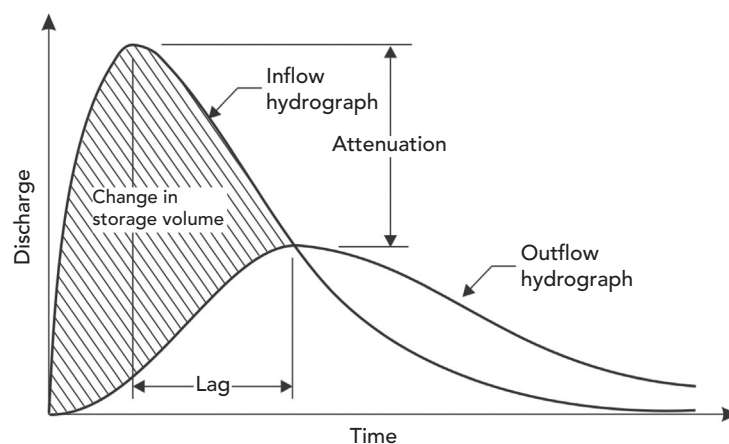


Figure 5.2: Detention pond inflow and outflow hydrographs.

event. If this volume is greater than the storage capacity of the pond, the pond will overflow.

The following three scenarios summarize the relationship of a pond's inflow rate, outflow rate, storage, and water surface elevation for any given time interval within a stormwater runoff event:

- If the average inflow rate is greater than average outflow rate during a time interval, the volume of stored water in the pond increases during the time interval, and the water surface elevation in the pond increases.
- If the average inflow rate is equal to the average outflow rate during a time interval, the volume of stored water does not change during the interval, and the water surface elevation remains constant.
- If the average outflow rate is greater than the average inflow rate during a time interval, the volume of stored water decreases during the time interval, and the water surface elevation in the pond decreases.

Most detention ponds have *uncontrolled outlets* (i.e., their outlets have no valves or gates that can be opened or closed). For a pond with an uncontrolled outlet into a free outfall, the peak of the outflow hydrograph from the pond will occur at the point where the outflow hydrograph intersects the receding limb of the inflow hydrograph (see Figure 5.2).

This concept can be used as a check on the validity of a set of routing calculations and can be reasoned as follows. Prior to the time of the intersection point, the inflow rate is larger than the outflow rate and the volume of water in the pond is increasing. After that intersection time, the volume is decreasing because the outflow rate is greater than the inflow rate. Thus the maximum stored volume of water in the pond occurs at the time where the two hydrographs intersect. Because the outflow rate from a pond increases as the headwater depth in the pond increases, it follows that the maximum outflow rate must occur at the maximum depth, which also corresponds to the maximum storage during the event. Finally, at the point of intersection, the outflow is equal to the inflow rate, resulting in a zero rate of change of stored volume at the moment in time when the water surface crests in the pond.

5.4 Components of Detention Facilities

Detention facilities usually consist of an aboveground pond that is excavated and graded into the land surface, or they may consist of shallow collection areas in parking lots. In areas where available space is at a premium, it may be economically advantageous to use underground storage consisting of pipes and/or vaults. Thus the physical components of individual detention facilities can be quite variable, but some general observations can be made. The following subsections provide descriptions of the most common elements of detention facilities.

Dam Embankments

In many cases, detention facilities are formed by excavating and grading a depression in the land surface, and the resulting pond has no dam embankment per se. In other cases, a stream or localized depression is dammed to create a detention pond. Most typically, a combination of these two approaches is used in which an existing depression is enlarged and dammed. The design of the dam should conform to accepted geotechnical engineering practices. Governmental dam safety and permitting requirements may be applicable if the embankment height or storage exceeds a regulatory limit.

Freeboard

Freeboard for a storm event is defined as the vertical distance from the maximum water surface elevation (stage) to the top of the dam. The storage available within the freeboard elevation range adds a “safety” volume to the required detention storage volumes that exist below this elevation. Some jurisdictions and reviewing agencies mandate freeboard requirements for the maximum design event.

Pond Bottoms and Side Slopes

Side slopes of ponds, especially for wet ponds that become attractions for people, should be gentle. Barrier vegetation, such as cattails, can also be installed along the perimeters of wet ponds to limit access to the water’s edge.

Detention pond bottoms should have a slope sufficient to ensure complete drainage between storm events in dry ponds, and to facilitate pond drainage for maintenance in the case of wet ponds. For dry ponds, the engineer should consider providing a riprap or concrete low-flow channel from the inlet to the outlet structure to assist in draining the pond bottom.

Outlet Structures

The detention pond outlet structure allows flow to discharge from the pond at a controlled rate. Early detention ponds were usually designed to control only a single runoff event having a specified recurrence interval; thus these ponds had very simple outlet structures. This practice provided little to no benefit with smaller or larger storm events.

Modern detention design typically focuses on control of multiple storm events and often requires several outlet openings at different pond stages. These multiple openings can consist of multiple pipe outlets with differing diameters or invert elevations, orifices at various levels, overflow weirs, or any combination of these elements. Figure 5.3 depicts a profile view of an outlet structure configuration consisting of several structural components.

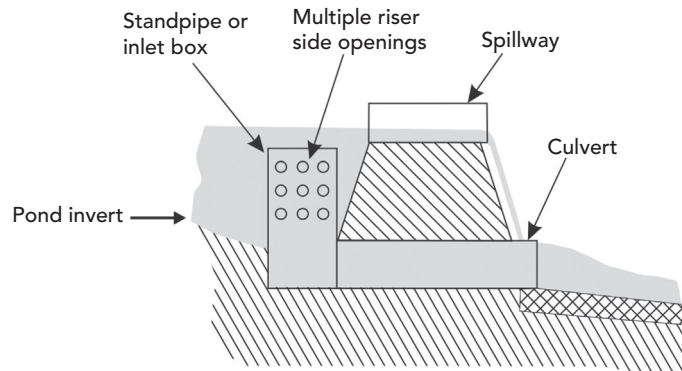


Figure 5.3: Detention pond inflow and outflow hydrographs.

Overflow Spillways

All detention ponds, regardless of the type, should be fitted with **overflow spillway** facilities to convey flows in excess of those for which the detention facility was designed. The overflow spillway may be an extension of a normal weir outlet, or it may be completely separate. To prevent undue damages downstream, consideration must be given to the paths that pond overflows will take. These outflow pathways are part of the major drainage system referred to in previous chapters.

5.5 Routing Data: Storage and Hydraulic Relationships

Hydrograph routing through a detention facility is the process of computing an outflow hydrograph and water surface elevation data based on a specified inflow hydrograph. Pond routing requires two key types of data in order to describe the pond's storage and hydraulic characteristics:

- **stage versus storage volume data** to describe the pond's shape and size
- **stage versus discharge data** to describe the pond outlet hydraulics.

These relationships are discussed in the following subsections.

Stage versus Storage Volume

The **stage versus storage relationship** for a pond relates pond water surface elevation (the **stage**) or water depth (h) to the volume of water stored (S). The relationship may be given in a table or graphed as in Figure 5.4. Precision of the curve or table increases as the incremental elevation between the points decreases. The designer must judge the appropriate degree of storage approximation when setting the elevation increment size for the rating curve.

Figure 5.4 demonstrates how stage-storage data is used in routing to determine the relative rise or fall in water surface for a given change in volume. In this example, h_1 and S_1 are, respectively, the elevation and volume at the beginning of the time step, and ΔS is the change in volume during the time step calculated from the difference between the integrated inflow and outflow hydrographs. At the end of the time step, volume $S_2 = S_1 + \Delta S$. Elevation h_2 at the end of the time step can be found from the curve based on S_2 . In this example, the net change in storage is positive; thus the change in water surface elevation is also positive (the water is rising).

If a detention facility has a regular geometric shape (e.g., it is a cylindrical tank), a geometric equation can be applied to describe the stage-storage volume relationship. In most cases, however, the engineer must first determine a detention basin's surface area (A) at incremental elevations or depths and then use this information to develop stage-storage data. Two methods for determining the stage-storage relationship—the *average-end-area method* and the *conic method*—are presented in this section. Numerical integration may also be used in some cases, although this technique is beyond the scope of this text.

Average-End-Area Method

The average-end-area method uses the same concept frequently seen in computations for estimating earthwork volumes. In this method, the incremental storage volume available between two water surface elevations is computed as:

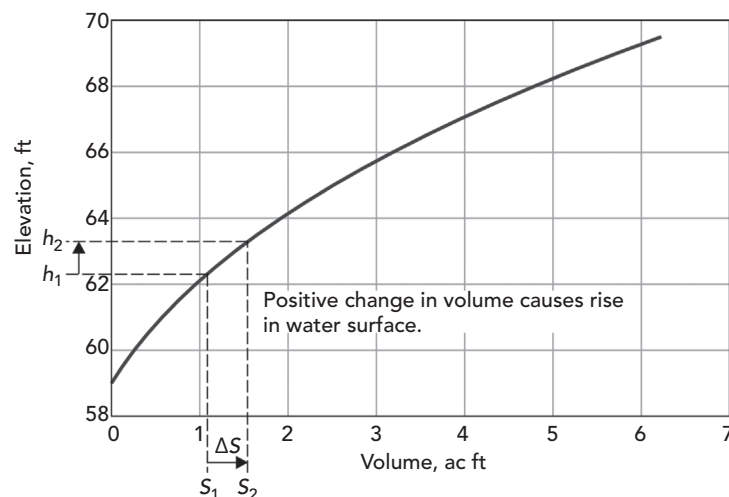


Figure 5.4: Relationship of stage-storage data to pond routing.

$$\Delta S = \frac{A_1 + A_2}{2} (h_2 - h_1)$$

where

h_1 = water surface elevation 1

h_2 = water surface elevation 2 ($h_2 > h_1$)

A_1 = available water surface area corresponding to stage h_1

A_2 = available water surface area corresponding to stage h_2

ΔS = incremental storage volume available between h_1 and h_2 .

Computation of incremental storage amounts for a series of known stages and surface areas, followed by summing of the incremental storages, yields the desired relationship between h and S .

One limitation of the average-end-area method is that it applies a linear averaging technique to describe area, whereas the change in area is actually a second-order function of height for a pond with sloped sides. However, the error associated with numerical volume integration is well within the error bounds of other key design assumptions, such as the selected design storm and watershed homogeneity.

Example 5.1: Computing Storage Volume Using the Average-End-Area Method

An underground storage vault is to be constructed to serve as a detention pond for a heavily urbanized area. The walls of the vault will be vertical, but the floor will be sloped along its length to promote drainage. In plan view, the vault is rectangular with a length of 75 ft and a width of 40 ft. The slope of the floor, along the 75-ft dimension, is 2%. The invert of the lower end of the vault is at an elevation of 123.00 ft.

Determine the elevation versus surface area relationship for the vault and use this information to calculate and graph the elevation-storage relationship. Use the average-end-area method.

Solution

The difference in elevation along the length of the floor of the vault is

$$75(0.02) = 1.50 \text{ ft}$$

Because the floor elevation at its lower end is at an elevation (stage) of 123.00 ft, the elevation at its high end is 124.50 ft. For any stage between these elevations, the rectangular water surface area is equal to the width of the vault ($W = 40$ ft) times the length of the water surface area. That length depends on the depth of water in the vault and can be expressed in terms of the elevation, h , as

$$L = \frac{(b - 123.00)}{0.02}$$

The corresponding water surface area is

$$A_s = WL = \frac{40(b - 123.00)}{0.02}$$

These expressions apply to values of b ranging from 123.00 to 124.50 ft. For stages above 124.50 ft, the surface area is constant because of the vertical walls and is equal to

$$A_s = 40(75) = 3,000 \text{ ft}^2$$

The first two columns of Table 5.1 give the elevation versus surface area relationship for elevations from 123.00 to 130.00 ft.

To compute the elevation–storage relationship using the average-end-area method, adjacent area values in Column 2 are added and then divided by the stage increment for those areas. The steps are described next, and results are reported in Table 5.1. The relationship is plotted in Figure 5.5.

- Column 3 = [Column 2 (current row)] + [Column 2 (previous row)]
- Column 4 = [Column 1 (current row)] – [Column 1 (previous row)]
- Column 5 = [Column 3 (current row)/2] × [Column 4 (current row)]
- Column 6 = [Column 5 (current row)] + [Column 6 (previous row)]

Table 5.1: Calculation of Detention Volume using the Average-End-Area Method

(1)	(2)	(3)	(4)	(5)	(6)
Elevation (ft)	Area (ft²)	$A_s(b_1) + A_s(b_2)$ (ft²)	$b_1 - b_2$ (ft)	ΔS (ft³)	Cumulative S (ft³)
123.00	0	0	0.00	0.0	0.0
123.25	500	500	0.25	62.5	62.5
123.50	1,000	1,500	0.25	187.5	250.0
123.75	1,500	2,500	0.25	312.5	562.5
124.00	2,000	3,500	0.25	437.5	1,000.0
124.25	2,500	4,500	0.25	562.5	1,562.0
124.50	3,000	5,500	0.25	687.5	2,250.0
125.00	3,000	6,000	0.50	1,500	3,750.0
130.00	3,000	6,000	5.00	15,000	18,750.0

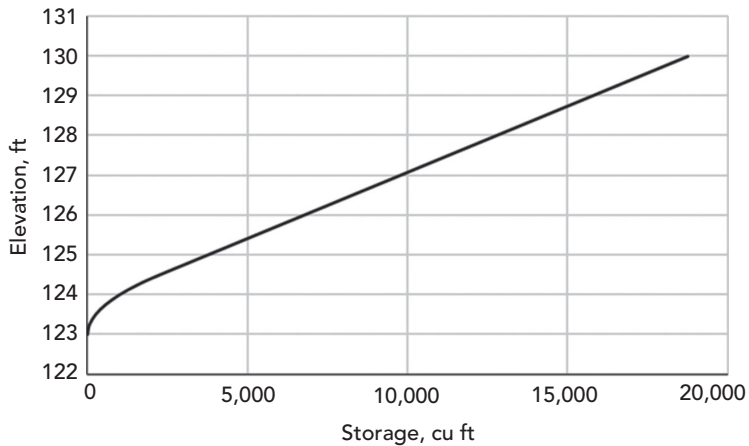


Figure 5.5: Plot of elevation–storage relationship for Example 5.1.

Conic Method

This method applies a conic geometry to each stage increment to approximate the nonlinear relationship between stage and water surface area, as illustrated in Figure 5.6. The incremental storage volume between two stages h_1 and h_2 ($h_2 > h_1$) is defined as

$$\Delta S = \left(\frac{h_2 - h_1}{3} \right) (A_1 + A_2 + \sqrt{A_1 A_2})$$

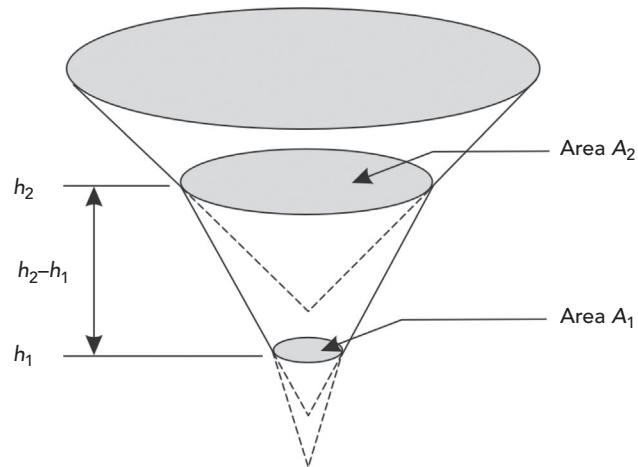
Example 5.2: Computing Storage Volume Using the Conic Method

Use the conic method to calculate and graph the elevation versus storage relationship for a pond with an elevation versus surface area relationship as given in Table 5.2.

Solution

Table 5.3 is used to determine the cumulative volume at each elevation. Each column represents a step in the calculations used for the conic method. A plot of the elevation–storage relationship is given in Figure 5.7.

- Columns 1 and 2 are the elevation versus surface area relationship and are taken from Table 5.2.
- Column 3 = {[Column 1 (current row)] – [Column 1 (previous row)]}/3.

**Figure 5.6:** Illustration of conic method for computing pond volume.**Table 5.2:** Elevation-Area Data for Example 5.2

Elevation (ft)	Area (ac)
100.00	0.158
100.50	0.170
101.00	0.182
101.50	0.194
102.00	0.207
102.50	0.221
103.00	0.234
103.50	0.248
104.00	0.263
104.50	0.278
105.00	0.293
105.50	0.309
106.00	0.325
106.50	0.342
107.00	0.359

- Column 4 = [(Column 2 (current row)) + (Column 2 (previous row))] + [(Column 2 (current row)) × (Column 2 (previous row))] ^{0.5}.
- Column 5 = (Column 3) × (Column 4).
- Column 6 = [Column 6 (previous row)] + [Column 5 (current row)].

Detention Pond Design

Routing Data: Storage and Hydraulic Relationships

Table 5.3: Calculations for Example 5.2

(1)	(2)	(3)	(4)	(5)	(6)
Elevation (ft)	Area (ac)	$(b_2 - b_1)/3$ (ft)	$A_1 + A_2 + \sqrt{A_1 A_2}$ (ac)	ΔS (ac ft)	Cumulative S (ac ft)
100.00	0.158	0.000	0.000	0.000	0.000
100.50	0.170	0.167	0.492	0.082	0.082
101.00	0.182	0.167	0.528	0.088	0.170
101.50	0.194	0.167	0.564	0.094	0.264
102.00	0.207	0.167	0.602	0.100	0.364
102.50	0.221	0.167	0.641	0.107	0.471
103.00	0.234	0.167	0.682	0.114	0.585
103.50	0.248	0.167	0.723	0.121	0.705
104.00	0.263	0.167	0.766	0.128	0.833
104.50	0.278	0.167	0.811	0.135	0.968
105.00	0.293	0.167	0.856	0.143	1.111
105.50	0.309	0.167	0.903	0.150	1.262
106.00	0.325	0.167	0.951	0.158	1.420
106.50	0.342	0.167	1.000	0.167	1.587
107.00	0.359	0.167	1.051	0.175	1.762

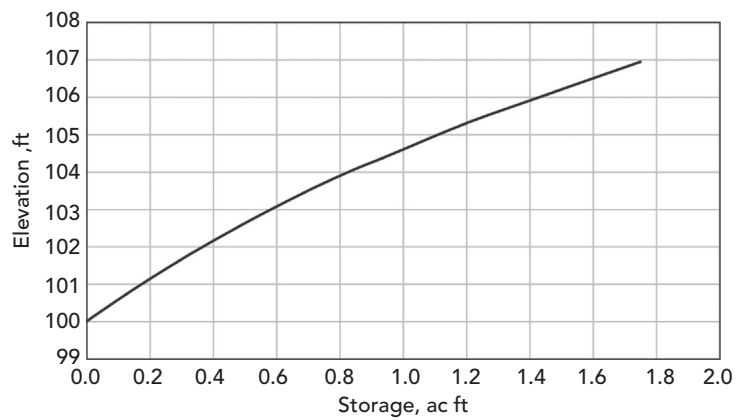


Figure 5.7: Stage-storage relationship for Example 5.2.

Stage versus Discharge

The discharge Q from a detention facility depends on the stage h of the pond, on the geometric and hydraulic characteristics of the outlet(s), and possibly on tailwater effects that may influence the outlet hydraulics. In cases where interconnected ponds are to be routed, the discharge from one pond to another depends on the stages in the ponds, and on the hydraulic characteristics of the channel or conduit connecting them.

The hydraulic characteristics for a pond outlet structure are described by a **stage** versus **discharge** rating table, which is applied during routing to determine the relative increase or decrease in outflow, given a change in the stage (water surface elevation). Figure 5.8 demonstrates how elevation versus discharge data is related to routing. The rating curve is calculated by computing the flow for various stage elevations. Precision of the curve increases as the incremental elevation between points decreases. The designer must judge the appropriate degree of flow approximation when setting the elevation increment size for the rating curve.

In Figure 5.8, h_1 and Q_1 are the elevation and outflow at the beginning of the time step, respectively. Elevation h_2 at the end of the time step is determined from the stage versus storage curve based on the change in volume. From h_2 , Q_2 can be found using the stage versus discharge curve. Thus for this example the net change in elevation is positive (rising), so the change in outflow is positive (increasing).

Most dry detention facilities have both **low-** and **high-level outlets**. The low-level outlet may consist of a pipe or culvert, that is, culvert one or more orifices

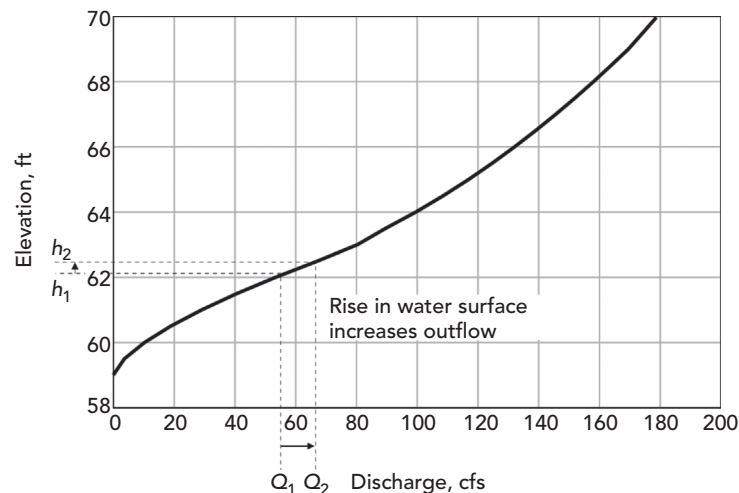


Figure 5.8: Relationship of elevation–discharge data to detention routing.

or a weir positioned at or near one or more orifices, or a weir. The high-level outlet is usually an overflow spillway and is expected to function only during extreme runoff events with recurrence intervals that exceed the pond's design constraints. Where ponds are designed to control runoff rates for several different recurrence intervals, intermediate outlets are usually also provided. They are similar to low-level outlets in practice and design but are installed at higher elevations.

The stage versus discharge relationship for a pond is developed using hydraulic relationships depending on the number and types of individual outlets. The relationships for various types of pond outlets are discussed next.

Pipe or Culvert Outlets

The stage versus discharge relationship for a pipe or culvert outlet can be developed using culvert design procedures described in Chapter 4. The outlet may perform under either inlet or outlet control, depending on downstream effects. A culvert performance curve delineating discharge as a function of the headwater depth (or stage) is the desired stage versus discharge relationship.

For small diameter pipes ($D < 12$ in.), culvert hydraulics can be adequately represented using the orifice equation (discussed in Section 1.3). This assumption holds true if the ratio of headwater depth to pipe diameter is at least 1.5, and if tailwater (outlet control) effects are negligible (Brown et al., 2009).

Orifices and Weirs

The hydraulic properties of orifices and weirs are discussed in Section 1.3.

Overflow Spillways

An overflow spillway has the purpose of safely conveying discharges when the design storage capacity of a detention pond is exceeded during large storm events. Overflow spillways are commonly referred to as *emergency spillways* because they are designed to operate under extreme rainfall conditions to prevent dam overtopping.

The storm recurrence interval for which an overflow spillway should be designed is an issue that requires exercise of professional judgment, as well as consideration of the adverse effects of downstream flooding or damages. In areas where the major drainage system is evaluated for a designated storm recurrence interval, the same recurrence interval might be used for overflow spillway design. On the other hand, if failure of a dam embankment could cause considerable damages or loss of life downstream, regulatory requirements pertaining to dams may be appropriate.

Detailed coverage of spillway hydraulics is beyond the scope of this book. For more advanced discussion on spillway and outlet design, refer to *Stormwater Conveyance Modeling and Design* (Haestad Methods and R. Durrans, 2003).

Composite Stage versus Discharge Relationships

The overall, or composite, stage versus discharge relationship for a detention pond represents the total discharge from a pond, possibly occurring through several outlets, as a function of the water surface elevation (or depth). Assembly of a composite relationship involves computation and summing of the discharges through each outlet for each stage of interest. Example 5.3 demonstrates the development of the stage–discharge relationship.

Structures in Parallel

The most basic type of composite outlet structure is one that has different outlet devices working independently of one another. In this situation, the hydraulic rating characteristics for each device are simply summed at each elevation to compute the total outflow. Figure 5.9 displays a composite set of outlet structures working in parallel, and Figure 5.10 shows the rating curve resulting from this outlet configuration.

Example 5.3: Stage versus Discharge for a Composite Structure

A simple composite outlet structure is being designed with an orifice and a weir operating in parallel. The orifice is a 6-in. diameter orifice plate with an invert elevation of 100.00 ft and an orifice coefficient of 0.60. The orifice discharges into the top section of a large culvert; under maximum head conditions in the pond, the water within the culvert will not rise above the orifice invert (i.e., the orifice controls throughout). The concrete rectangular suppressed weir has a weir coefficient approximated as 2.6 for the entire range of headwaters, a weir length of 15 ft, and a crest elevation at 105.00 ft. It operates under free outfall conditions.

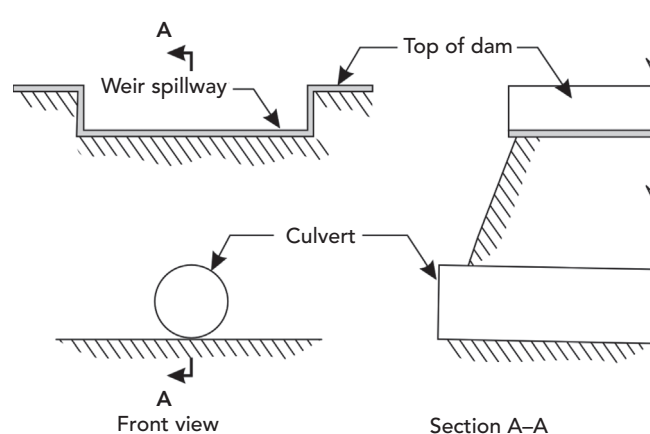


Figure 5.9: Composite outlet with outlet structures discharging in parallel.

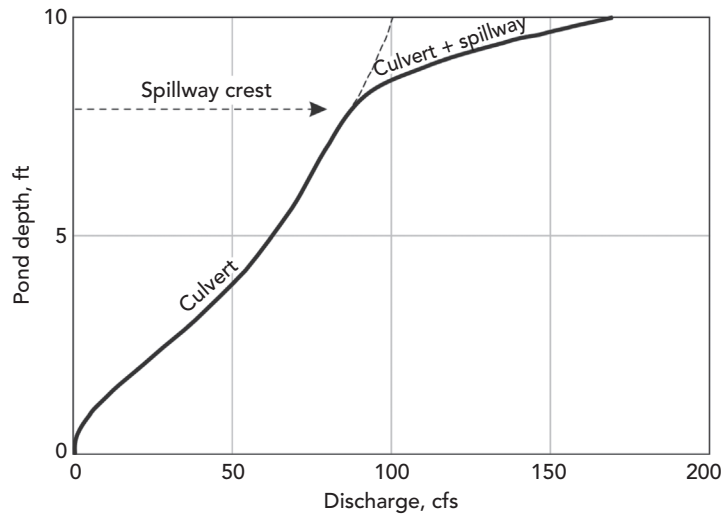


Figure 5.10: Depth versus discharge for a composite outlet with outlet structures discharging in parallel.

Calculate and graph the elevation versus discharge rating curve from elevations 100.00 to 107.00 ft for each 0.50-ft increment. (Note that a smaller increment would yield a smoother, more precise rating curve.)

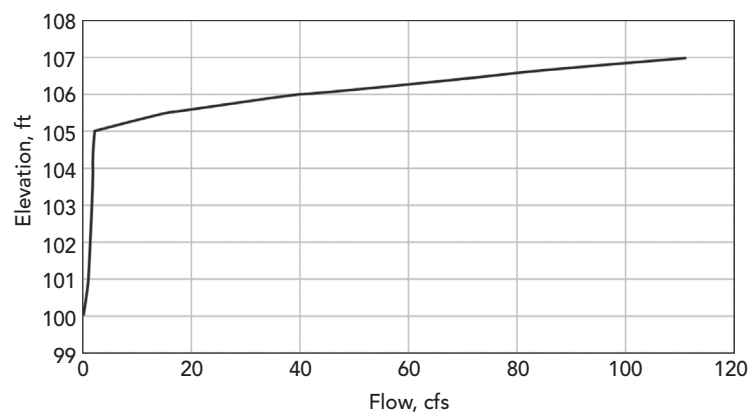
Solution

Table 5.4 shows the stage-storage calculation results described by the following steps. Figure 5.11 is a plot of the composite structure rating curve.

- Orifice centroid elevation = $100.00 \text{ ft} + (1/2 \times 0.5 \text{ ft}) = 100.25 \text{ ft}$.
- Orifice area = $\pi D^2/4 = 0.196 \text{ ft}^2$.
- Column 1 is the range of rating table elevations (100.0 – 107.0 ft, in 0.5-ft increments).
- Column 2 is the head on the orifice = (Column 1) – (orifice centroid elevation).
- Column 3 is the orifice discharge = solution to orifice equation for $C = 0.60$, $A = 0.196$, $h = \text{Column 2}$ (the orifice equation is $Q = CA(2gH)^{0.5}$, as introduced in Chapter 1.).
- Column 4 is the head on the weir = (Column 1) – (weir crest elevation).
- Column 5 is the weir discharge = solution to rectangular weir equation for $C = 2.6$, $L = 15.0$, $h = \text{Column 4}$ (the weir equation is $Q = CLH^{3/2}$, as introduced in Chapter 1.).
- Column 6 is the composite structure discharge = (Column 3) + (Column 5).

Table 5.4: Stage–Discharge Calculations for Example 5.3

(1)	(2)	(3)	(4)	(5)	(6)
Elevation (ft)	Centroid = 100.25 ft		Crest = 105.00 ft		Total Flow (cfs)
	Head (ft)	Flow (cfs)	Head (ft)	Flow (cfs)	
100.00	0.00	0.00	0.00	0.00	0.00
100.50	0.25	0.47	0.00	0.00	0.47
101.00	0.75	0.82	0.00	0.00	0.82
101.50	1.25	1.06	0.00	0.00	1.06
102.00	1.75	1.25	0.00	0.00	1.25
102.50	2.25	1.42	0.00	0.00	1.42
103.00	2.75	1.57	0.00	0.00	1.57
103.50	3.25	1.70	0.00	0.00	1.70
104.00	3.75	1.83	0.00	0.00	1.83
104.50	4.25	1.95	0.00	0.00	1.95
105.00	4.75	2.06	0.00	0.00	2.06
105.50	5.25	2.16	0.50	13.79	15.95
106.00	5.75	2.26	1.00	39.00	41.27
106.50	6.25	2.36	1.50	71.65	74.01
107.00	6.75	2.45	2.00	110.31	112.76

**Figure 5.11:** Rating curve for outlet structure in Example 5.3.

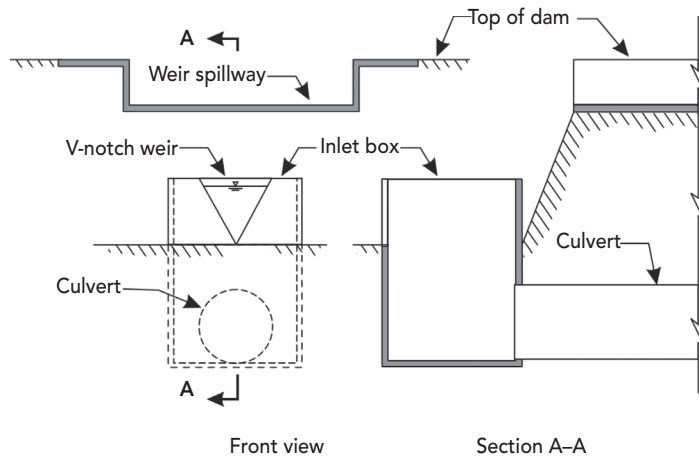


Figure 5.12: Composite outlet with outlet structures discharging in parallel and in series.

Structures in Series

When outlet structures are in series, it is possible that backwater effects caused by hydraulic controls at downstream structures will affect the discharges through connecting upstream structures. In these cases, hydraulic grade line calculations are required to determine the interactions between individual structure components.

Figure 5.12 illustrates a pond outlet with structures in series. In this example, as the stage rises, the culvert eventually throttles (controls) the combined flow through the v-notch weir and the inlet box, causing the inlet box riser to be fully submerged. This phenomenon is shown graphically near the center portion of the stage versus discharge relationship in Figure 5.13.

This text provides only an introduction to basic outlet hydraulics. For more detailed explanations on analyzing complex outlet structures such as outlets in series, refer to *Stormwater Conveyance Modeling and Design* (Haestad Methods and R. Durrans, 2003).

5.6 Storage Indication Method

Routing of a detention pond is the process by which a given inflow hydrograph to the pond is transformed into the corresponding outflow hydrograph. In addition to the inflow hydrograph, the outflow hydrograph also depends on the storage and hydraulic relationships of the pond (described in the previous

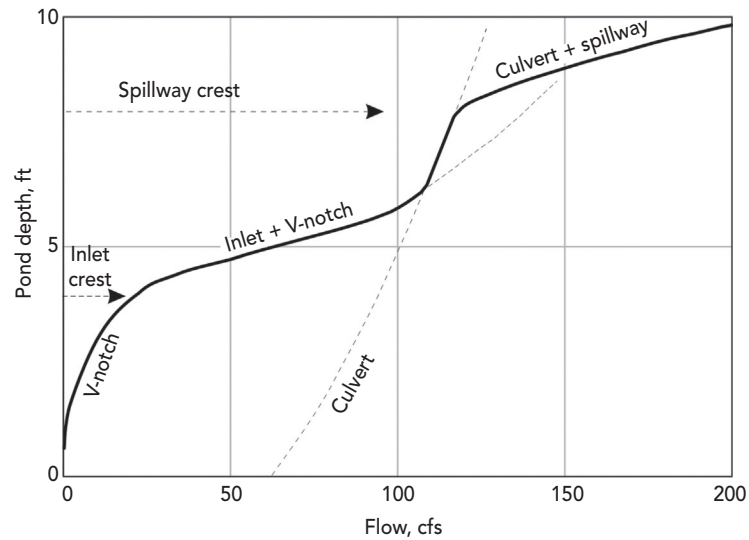


Figure 5.13: Rating curve for a composite outlet with outlet structures discharging in parallel and in series.

sections) and on the initial water surface elevation in the pond when the inflow hydrograph begins.

Fundamentally, routing of a pond amounts to a solution of the conservation of mass equation:

$$\frac{dS}{dt} = I(t) - O(t)$$

where

S = the volume of water in the pond (ft^3 , m^3);

$I(t)$ = specified inflow at time t (cfs, m^3/s); and

$O(t)$ = the outflow at time t (cfs, m^3/s).

This equation states that the change in storage during a time period is equal to the difference between the inflow and the outflow. There are two unknowns in this equation (S and O); thus information on both storage characteristics and hydraulic characteristics is required to solve it. Various routing methods are available, but the most commonly used method in simple stormwater detention is the **storage indication method**, which is based on a finite difference approximation of the conservation of mass equation.

Figure 5.14 shows the inflow and outflow hydrographs for a pond. The graph magnifies a time slice (step) for which the inflow is greater than the outflow. The x -axis of the slice has units of time (t), and the y -axis of the slice, which represents

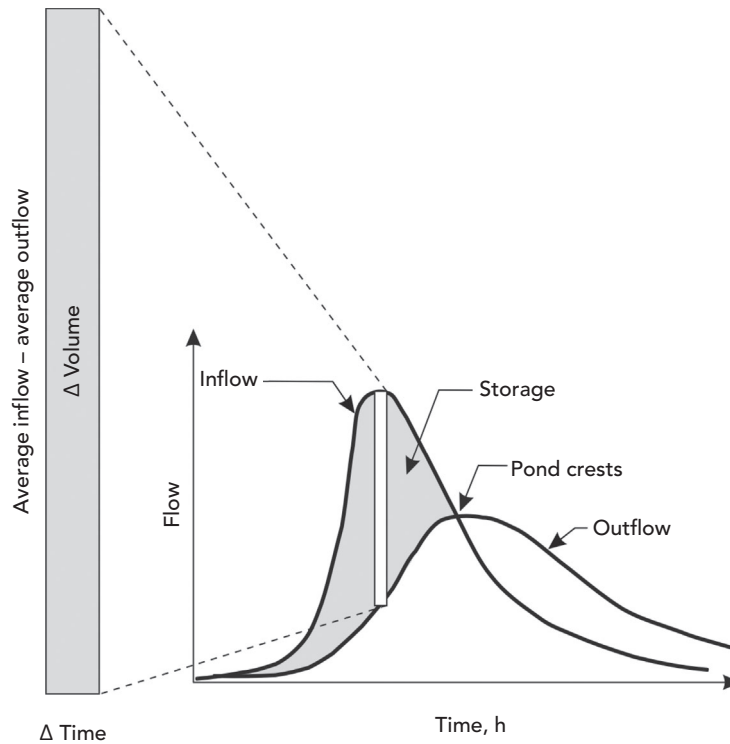


Figure 5.14: Change in pond volume for a routing time step.

flow rate, has units of volume (L^3) per time, or L^3/t . Thus the bounded area of the slice has units of $T \times (L^3/t) = L^3$, or volume. Because the inflow is greater than the outflow for this time slice, the change in volume is positive, which means the pond water surface (stage) is rising during this time step.

At the point where the inflow equals the outflow, the change in storage is zero, the water surface elevation (stage) is constant, and the pond is cresting. After the pond has crested, the outflow rate is greater than the inflow rate, resulting in a net loss in volume that causes the water surface (stage) to fall during the time step.

Because the shaded area of Figure 5.14 represents the bounded area for which inflow is greater than outflow before the pond crests, this area represents the total volume stored while the pond is filling. As stated previously, the shaded area represents the maximum storage volume.

Figure 5.15 shows a portion of the inflow and outflow hydrographs for a pond. The graph illustrates a time period in which the inflow rate is greater than the outflow rate, but the relationships that follow are applicable for any time period, regardless of the relative magnitudes of the inflow and outflow rates. Over a time interval $\Delta t = t_{n+1} - t_n$, the inflow rate varies from I_n to I_{n+1} , and

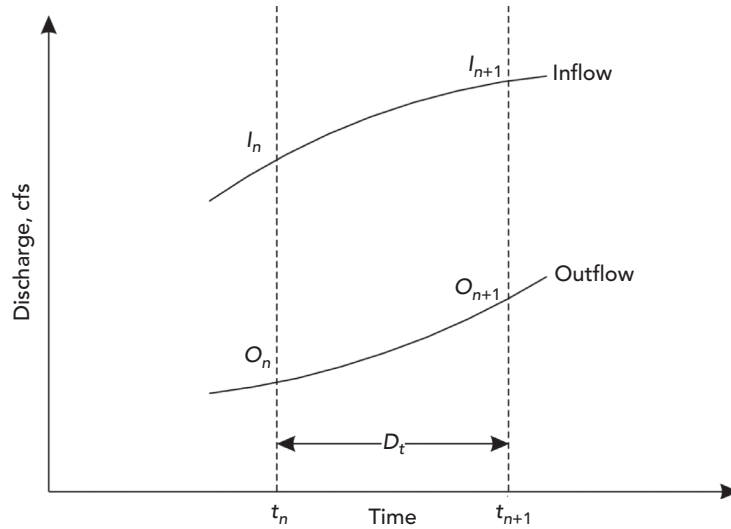


Figure 5.15: Definition sketch for terms in finite difference approximation.

therefore the average inflow rate over the time interval can be approximated as $(I_n + I_{n+1})/2$. Over the same time interval, the average outflow rate can be approximated as $(O_n + O_{n+1})/2$. The change in storage over the time interval is $\Delta S = S_{n+1} - S_n$, whereas the time rate of change of storage may be approximated as $\Delta S/\Delta t = (S_{n+1} - S_n)/\Delta t$. Substitution of the average inflow and outflow rates and the approximate rate of change of storage into the conservation of mass equation yields

$$\frac{S_{n+1} - S_n}{\Delta t} = \frac{I_n + I_{n+1}}{2} - \frac{O_n + O_{n+1}}{2}$$

Because the inflow hydrograph is a known quantity in detention pond routing, the terms I_n and I_{n+1} are known for every time interval in the routing process. For the first time interval, at the beginning of a pond inflow event, the terms O_n and S_n are also initially known. Note that O_n and S_n are related as for each pond stage h_n there is a corresponding S_n and a corresponding O_n . Rearranging the finite difference approximation with unknown quantities on the left and known quantities on the right yields the storage indication equation:

$$\left(\frac{2S_{n+1}}{\Delta t} + O_{n+1} \right) = \left(\frac{2S_n}{\Delta t} - O_n \right) + (I_n + I_{n+1})$$

Thus knowledge of the quantities S_n , O_n , and I_n at the beginning of a time interval (at time t_n), as well as knowledge of the inflow rate I_{n+1} at the end of the time interval, can be used to compute the value of the left side of the equation. However, there are two unknowns on the left side (O_{n+1} and S_{n+1}). Additional information must be established to expose the relationship between O_{n+1} and S_{n+1} so that these two values can be determined.

A **storage indication curve** is a graph delineating the relationship between O and $(2S/\Delta t + O)$. Such a graph can be constructed by realizing that for any stage h in a pond, there are corresponding values of O and S ; hence, there are corresponding values of O and $(2S/\Delta t + O)$.

In performing storage indication calculations for a specified time interval $n+1$, the storage indication equation is applied to compute the numerical value of the quantity $(2S_{n+1}/\Delta t + O_{n+1})$. Next, the value of $(2S_{n+1}/\Delta t - O_{n+1})$ for the time interval $n+1$ must be found. Because the value of $(2S_{n+1}/\Delta t + O_{n+1})$ is known, and this value can be used with the storage indication curve to find O_{n+1} , $(2S_{n+1}/\Delta t - O_{n+1})$ can be computed as

$$\left(\frac{2S_{n+1}}{\Delta t} - O_{n+1} \right) = \left(\frac{2S_{n+1}}{\Delta t} + O_{n+1} \right) - 2O_{n+1}$$

The term on the left side of the equation can now be used in the next time step ($n+2$) to compute $(2S_{n+2}/\Delta t + O_{n+2})$, and the process repeats through every time step.

Section 5.7 describes the full procedure used in storage indication routing and presents an example problem.

5.7 Stormwater Detention Analysis Procedure

This chapter introduced each of the basic components required for modeling how a pond functions during a rainfall event. In order to apply pond routing techniques to model a pond, the various components of hydrographs, storage volume, and outlet structures are used in the following steps:

1. Calculate the postdeveloped inflow hydrograph discharging into the pond. Methods for calculating hydrographs are covered in Chapter 2.
2. Calculate the storage volume rating curve (stage versus storage) for the given pond design. The rating curve should start at the dry invert of the pond and extend to the top of the dam.
3. Calculate the outflow rating curve (stage versus outflow) for the given pond outlet structure. The rating curve should match the range of elevations of the physical pond design, that is, match the storage rating curve range of elevations. This holds true even if lower elevations will have zero discharge within a wet pond.

4. Establish the relationship between stage, storage, and discharge by creating a storage indication curve for $(2S/\Delta t + O)$ versus outflow.
5. Use the hydrograph from Step 1 and the storage indication curve from Step 4 to route the inflow hydrograph through the storage and discharge relationship to obtain a pond outflow hydrograph.

Example 5.4: Detention Pond Routing Example

Route the inflow hydrograph given in Table 5.5 through the pond described by the stage versus storage data computed in Example 5.2 and the stage versus discharge data computed in Example 5.3. Determine the maximum water surface elevation for the detention basin, the maximum storage volume, maximum outflow rate, and peak lag resulting from detention.

Table 5.5: Inflow Hydrograph for Example 5.4

Time (min)	Inflow (cfs)
0	0.00
10	0.58
20	5.94
30	15.44
40	20.75
50	20.27
60	17.44
70	14.19
80	11.42
90	9.44
100	8.12
110	7.27
120	6.61
130	5.49
140	3.32
150	1.52
160	0.66
170	0.29
180	0.12
190	0.05
200	0.02
210	0.00

Solution

The results from Examples 5.2 and 5.3 are combined in Table 5.6. Note the importance of calculating the same elevations for storage and discharge in order to directly correlate flow and storage at each elevation.

Follow the five steps outlined for storage indication pond routing:

1. Compute the postdeveloped pond inflow hydrograph using the procedures presented in Chapter 2 or another method. In this example, the final inflow hydrograph is given.
2. Calculate the stage versus storage curve from the pond grading plan or storage structure geometry. This information was calculated in Example 5.2 (no calculations are necessary for this example).
3. Calculate the stage versus outflow curve for the pond outlet structure. This information was calculated in Example 5.3.
4. Calculate and graph the storage indication curve that correlates $2S/\Delta t + O$ and outflow. To calculate this curve, the rating elevations for the stage versus storage curve must be identical to those used for the stage versus outflow curve. Use a 10-min time step in this example.

Step 4 results for $2S/\Delta t + O$ are given in Table 5.7. Values given in Column 4 were computed using

Table 5.6: Elevation–Volume–Flow Data for Example 5.4

Elevation (ft)	Total Volume (ac-ft)	Total Flow (cfs)
100.00	0.000	0.00
100.50	0.082	0.47
101.00	0.170	0.82
101.50	0.264	1.06
102.00	0.364	1.25
102.50	0.471	1.42
103.00	0.585	1.57
103.50	0.705	1.70
104.00	0.833	1.83
104.50	0.968	1.95
105.00	1.111	2.06
105.50	1.262	15.95
106.00	1.420	41.27
106.50	1.587	74.01
107.00	1.762	112.76

$$\text{Column 4} = \{2.0 \times [(\text{Column 2}) \times 43,560 \text{ ft}^2/\text{ac}] / (10 \text{ min} \times 60 \text{ s/min})\} + (\text{Column 3})$$

Because the maximum inflow is about 20 cfs, we already know that the maximum outflow value should be equal to or less than 20 cfs. The storage indication curve shown in Figure 5.16 is an expanded view for the outflow range below 20 cfs, which makes it easier to read the smaller flow values during the routing tabulation.

5. Use the inflow hydrograph from Step 1 and the storage indication curve from Step 4 to route the hydrograph using a systematic worksheet approach as shown in Table 5.8.
 - Columns 1 and 2 represent the given inflow hydrograph.
 - Under initial conditions, Columns 3–6 have values of 0. Because the pond is initially empty, the initial elevation (Column 6) is 100.00 ft.
 - For subsequent time steps,
 - [Column 3 (current row)] = [Column 2 (previous row)] + [Column 2 (current row)]

Table 5.7: Relationship between Elevation and $2S/\Delta t + O$ for Example 5.4

(1)	(2)	(3)	(4)
Elevation (ft)	Volume (ac ft)	Flow (cfs)	$2S/\Delta t + O$ (cfs)
100.00	0.000	0.00	0.00
100.50	0.082	0.47	12.38
101.00	0.170	0.82	25.50
101.50	0.264	1.06	39.39
102.00	0.364	1.25	54.10
102.50	0.471	1.42	69.81
103.00	0.585	1.57	86.51
103.50	0.705	1.70	104.07
104.00	0.833	1.83	122.78
104.50	0.968	1.95	142.50
105.00	1.111	2.06	163.38
105.50	1.262	15.95	199.19
106.00	1.420	41.27	247.45
106.50	1.587	74.01	304.44
107.00	1.762	112.76	368.60

Detention Pond Design

Stormwater Detention Analysis Procedure

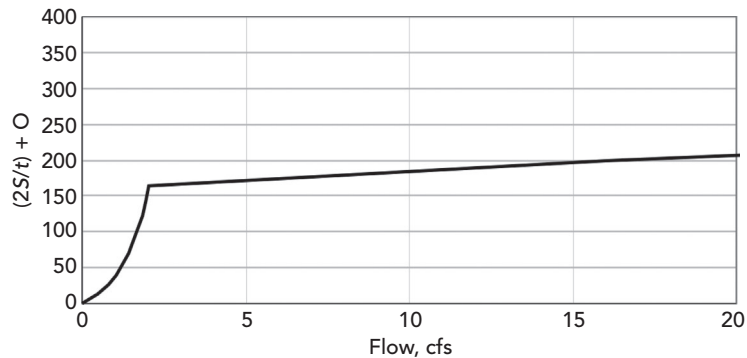


Figure 5.16: Storage indication curve for Example 5.4.

- [Column 5 (current row)] = [Column 3 (current row)] + [Column 4 (previous row)].
- Determine values in Column 6 by using the values in Column 5 and solving for the associated outflow from the storage indication curve calculated in Step 4.
- Solve Column 7 (current row) using stage versus outflow curve created in Step 3.
- Finally, to complete the current row, solve
 - [Column 4 (current row)] = [Column 5 (current row)] - 2 × [Column 6 (current row)].
- The tabulated results shown stop at $t = 300$ min into the routing. To complete the routing, the calculations need to be continued until the pond is fully drained (the routed outflow in Column 6 recedes to zero). The graph for the inflow and outflow hydrographs displays the entire routing time.

From the routing table, one can find:

- Maximum stage = 105.30 ft, which is above spillway crest.
- Maximum storage = 1.2 ac ft (determined from the stage versus storage data for an elevation of 105.30 ft).
- Maximum flow = 10.38 cfs.
- Outflow peak time = 90 min.
- Peak flow was delayed 50 min from inflow peak time of 40 min.

Table 5.8: Routing Results from Example 5.4

(1)	(2)	(3)	(4)	(5)	(6)	(7)
Time (min)	Inflow (cfs)	$I_1 + I_2$ (cfs)	$2S/\Delta t - O$ (cfs)	$2S/\Delta t + O$ (cfs)	Outflow (cfs)	Elevation (ft)
0	0	0	0.00	0.00	0.00	100.00
10	0.58	0.58	0.54	0.58	0.02	100.02
20	5.94	6.52	6.52	7.06	0.27	100.28
30	15.44	21.38	26.18	27.90	0.86	101.09
40	20.75	36.19	59.69	62.37	1.34	102.26
50	20.27	41.02	97.36	100.71	1.68	103.40
60	17.44	37.71	131.26	135.07	1.90	104.31
70	14.19	31.63	158.77	162.89	2.06	104.99
80	11.42	25.61	163.97	184.38	10.21	105.29
90	9.44	20.86	164.07	184.83	10.38	105.30
100	8.12	17.56	163.35	181.63	9.14	105.25
110	7.27	15.39	162.70	178.74	8.02	105.21
120	6.61	13.88	162.22	176.58	7.18	105.18
130	5.49	12.1	161.71	174.32	6.30	105.15
140	3.32	8.81	160.86	170.52	4.83	105.10
150	1.52	4.84	159.78	165.70	2.96	105.03
160	0.66	2.18	157.86	161.96	2.05	104.97
170	0.29	0.95	154.73	158.81	2.04	104.89
180	0.12	0.41	151.11	155.14	2.02	104.80
190	0.05	0.17	147.29	151.28	2.00	104.71
200	0.02	0.07	143.41	147.36	1.98	104.62
210	0	0.02	139.52	143.43	1.95	104.52
220	0	0	135.65	139.52	1.93	104.42
230	0	0	131.84	135.65	1.91	104.33
240	0	0	128.07	131.84	1.89	104.23
250	0	0	124.34	128.07	1.86	104.13
260	0	0	120.66	124.34	1.84	104.04
270	0	0	117.03	120.66	1.82	103.94
280	0	0	113.45	117.03	1.79	103.85
290	0	0	109.92	113.45	1.77	103.75
300	0	0	106.44	109.92	1.74	103.66

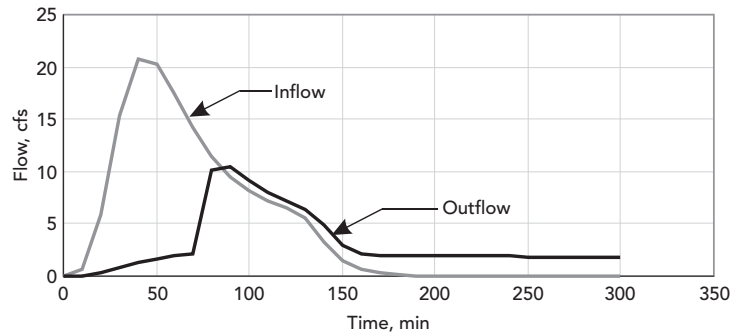


Figure 5.17: Pond inflow and outflow hydrographs.

5.8 PondPack and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software.

What Is PondPack?

PondPack is a stormwater modeling program capable of analyzing a wide range of situations, from simple sites to complex networked watersheds. The program analyzes pre- and postdeveloped watershed conditions and sizes ponds.

PondPack also computes outlet structure rating curves with tailwater effects, pond infiltration, pond detention times, and channel routing effects. It includes the capability to model interconnected ponds with divergent (multiple) outfalls.

The program can use any rainfall duration or distribution to compute hydrographs for multiple events. Hydrographs can be added at junctions and routed through multiple reaches and ponds. PondPack helps automate almost every aspect of drainage design computations.

PondPack graphically displays such items as watershed diagrams, hydrographs, rainfall curves, *I-D-F* curves, outlet rating curves, volume curves, time versus elevation curves, time versus volume curves, cross sections, channel rating curves, and a wide variety of other output diagrams.

PondPack builds customized reports organized by categories and automatically creates section and page numbers, tables of contents, and indexes. You can display a short results summary for an entire watershed or build a comprehensive drainage report showing any or all report items.

How Can You Use PondPack for Windows?

You can apply this software to all phases of a stormwater detention project, from feasibility studies to analysis of the final design. This product can be used to

- create full drainage reports with table of contents and index,
- compute pre- and postdeveloped flows using popular methods such as Technical Release-55 or Rational Method,
- estimate storage requirements,
- size ponds,
- model multistage outlets with tailwater considerations,
- analyze culverts with inlet/outlet control checking,
- model simple pond sites,
- solve complex watershed networks with interconnected ponds,
- model unlimited diversions,
- review tidal outfall effects on pond outlets,
- analyze channel capacity,
- check outflow rates against predeveloped conditions,
- compute water quality parameters such as minimum drain time and detention time,
- model any rainfall duration or time distribution, and
- simulate gauged rainfall events and synthetic rainfall distribution events.

5.9 Problems

The following problems can be solved with the PondPack computer program.

1. A detention pond has the shape of a trapezoidal basin. The specifications are:

Top elevation	120 ft
Top length	400 ft
Top width	300 ft
Bottom elevation	100 ft
Bottom length	280 ft
Bottom width	180 ft

Detention Pond Design

Problems

Construct an elevation versus volume graph for the basin. What is the volume at an elevation of 120 ft?

- Given the elevation-area data for a detention pond, construct an elevation versus volume graph for the pond. What is the volume of the pond at an elevation of 205 m?

Elevation—Volume Data for Problem 2

Elevation (m)	Area (m ²)
200.0	1,700
200.5	2,500
201.0	2,900
201.5	3,700
202.0	4,300
202.5	4,800
203.0	5,500
203.5	6,900
204.0	8,100
204.5	12,300
205.0	15,000

- An outlet structure for a pond consists of a riser pipe with two orifices. The riser is a 3-ft diameter pipe with a rim elevation of 105 ft. A 1-ft diameter orifice has an invert of 101 ft and a 0.5-ft orifice has an invert of 100 ft. Use 3.33 for the weir coefficient and 0.614 for the orifice coefficient. Construct an elevation versus flow diagram for a headwater range of 100–120 ft, with an increment of 1 ft. What is the discharge at a headwater elevation of 120 ft?
- A composite outlet has a 4-m broad crested weir at an elevation of 203.5 m and a 50-cm diameter orifice at an elevation of 200 m. The weir coefficient is 1.84 and the orifice coefficient is 0.614. Construct an elevation versus discharge curve for the structure over a headwater range of 200–205 m with a step of 1 m. What is the discharge at a headwater elevation of 205 m?
- Use the detention pond described in Problem 5.2 and the outlet structure described in Problem 5.4 to route the following hydrograph. What is the peak discharge from the pond? What is the maximum elevation in the pond?

Inflow Hydrograph Data for Problem 5

Time (h)	Flow (m ³ /s)	Time (h)	Flow (m ³ /s)
0.0	0.00	3.6	3.12
0.2	0.24	3.8	2.64
0.4	0.80	4.0	2.24
0.6	1.52	4.2	1.84
0.8	2.48	4.4	1.66
1.0	3.76	4.6	1.44
1.2	5.28	4.8	1.18
1.4	6.56	5.0	0.90
1.6	7.44	5.2	0.86
1.8	7.92	5.4	0.70
2.0	8.00	5.6	0.62
2.2	7.92	5.8	0.53
2.4	7.44	6.0	0.44
2.6	6.88	6.2	0.26
2.8	6.24	6.4	0.14
3.0	5.44	6.6	0.08
3.2	4.48	6.8	0.00
3.4	3.68	7.0	0.00

6. A proposed development will consist of single-family homes and a small park on a 25-ac site. Local regulations require that on-site detention be used to maintain peak runoff rates at predevelopment levels for a 25-year storm. Design a detention facility to meet this requirement.

Use the SCS 24-h Type II storm with a total depth of 6.5 in. and the SCS Unit Hydrograph Method to compute the runoff hydrograph. The pond is to be a trapezoidal basin with a total depth of 5 ft and 3H:1V side slopes. For the outlet structure, use a 1-ft diameter orifice with an invert at the bottom of the basin together with a rectangular weir. Size and place the weir such that the predevelopment peak flow is not exceeded and a minimum of 1 ft of freeboard is maintained. Document your design by filling in the table.

Detention Pond Design

Problems

Land Cover Data for Problem 6

Stage	Land Cover	Area (ac)	CN	t_c (min)
Predevelopment	Wooded	15	58	45
	Meadow	10	65	
Postdevelopment	0.5-ac residential lots	21	75	30
	Roads	0.3	98	
	Park	3.7	79	

Proposed Design for Problem 6

Predevelopment peak runoff flow rate/allowable outflow rate	
Estimated storage volume required	
Dimensions of bottom of pond	
Dimensions of top of pond	
Computed pond volume below freeboard elevation	
Weir length	
Height of weir above bottom of pond	
Maximum water surface elevation	
Peak pond outflow rate	

6

Pressure Piping Systems and Water Quality Analysis

6.1 Pressure Systems

Pressure piping network analysis has many applications, including well pumping systems and heating and cooling systems. This chapter deals primarily with the topic of pressure piping as it relates to potable water distribution systems.

The main purpose of a water distribution system is to meet demands for potable water. People use water for drinking, cleaning, gardening, and any number of other uses, and this water needs to be delivered in some fashion. A secondary purpose of many distribution systems is to provide water for fire protection.

If designed correctly, the network of interconnected pipes, storage tanks, pumps, and regulating valves provides adequate pressure, adequate supply, and good water quality throughout the system. If incorrectly designed, some areas may have low pressures, poor fire protection, and even present health risks.

Water Demands

Just as storm sewer analysis is driven by the watershed runoff flow rate, water distribution system analysis is driven by customer demand. Water usage rates and patterns vary greatly from system to system and are highly dependent on climate, culture, and local industry. Every system is different, so the best source of information for estimating demands is directly recorded system data.

Metered Demand

Metered demands are often a modeler's best tool and can be used to calculate average demands, minimum demands, peak demands, and so forth. This data can also be compiled into daily, weekly, monthly, and annual reports that show how the demands are influenced by weather, special events, and other factors.

Unfortunately, many systems still do not have complete system metering. For these systems, the modeler is often forced to use other estimation tools (including good engineering judgment) to obtain realistic demands.

Demand Patterns

A **pattern** is a function relating water use to time of day. Patterns allow the user to apply automatic time-variable changes within the system. Different categories of users, such as residential or industrial customers, will typically be assigned different patterns to accurately reflect their particular demand variations. A **diurnal curve** is a type of pattern that describes changes in demand over the course of a daily cycle, reflecting times when people are using more or less water than average. Most patterns are based on a multiplication factor versus time relationship, whereby a multiplication factor of 1.0 represents the base value (often the average value). In equation form, this relationship is written as:

$$Q_t = A_t \times Q_{base}$$

where

Q_t = demand at time t ,

A_t = multiplier for time t , and

Q_{base} = baseline demand.

Using a representative diurnal curve for a residence (Figure 6.1), we see that there is a peak in the diurnal curve in the morning as people take showers and prepare breakfast, another slight peak around noon, and a third peak in the evening as people arrive home from work and prepare dinner. Throughout the night, the pattern reflects the relative inactivity of the system, with very low flows compared to the average. (Note that this curve is conceptual and should not be construed as representative of any particular network.)

There are two basic forms for representing a pattern: stepwise and continuous. A **stepwise pattern** is one that assumes a constant level of usage over a period of time and then jumps instantaneously to another level where it again remains

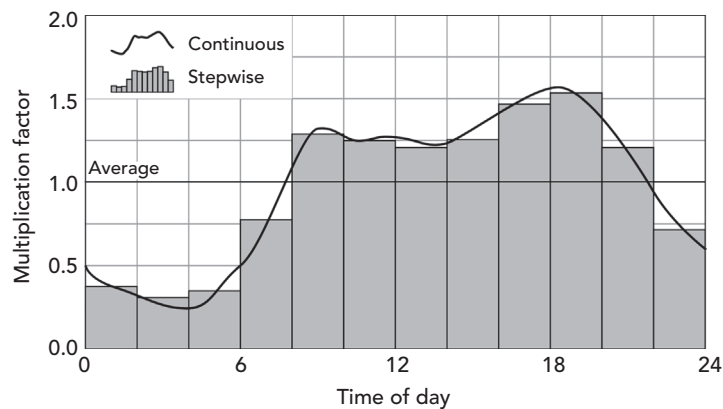


Figure 6.1: Typical Diurnal Curve.

steady until the next jump. A **continuous pattern** is one for which several points in the pattern are known and sections in between are transitional, resulting in a smoother pattern. Notice that, for the continuous pattern in Figure 6.1, the magnitude and slope of the pattern at the start and end times are the same—a continuity that is recommended for patterns that repeat.

Because of the finite time steps used in the calculations, most computer programs convert continuous patterns into stepwise patterns for use by the algorithms, with the duration of each step equal to the time step of the analysis.

Nonrevenue Water and Leaks

Not all the water that is produced by a treatment facility is delivered to the users. A portion of that water does not make it to the customers due to leaks or is used as unmetered flow. This nonrevenue water flow includes (1) what is lost in the system due to leaks and breaks in pipes and joints, (2) unmetered uses such as fire protection and maintenance and meter inaccuracy, (3) water theft, and (4) a variety of other minor water losses. Generally in all cases, more water will be produced and enter the piping network than is delivered to consumers.

Many times, the unaccounted-for water is used to gauge the performance of a pipe network. On an annual basis, water suppliers try to maintain nonrevenue water to less than 10% of the total water production. However, for older systems, the nonrevenue water can be much higher due to the aging pipe network, which can have a significant amount of leakage and other losses.

A hydraulic model can be used to evaluate leakage in a water distribution system. Pipe leakage is a pressure-dependent water loss (demand) that can be distributed throughout the pipe network, placed with a few large leaks at specific locations, or a combination of the two. Without complete knowledge of the type, size, and location of all leaks, some assumptions will need to be made when using a hydraulic model to assess and analyze leaks.

The first step is to estimate the amount of leakage. This is done by subtracting the metered flow for consumers from the total flow entering a water system. To estimate the total flow for the entire water system, the metered production water from a treatment plant can be used, or for a subsection of the water system, the metered flow at points where there is a measured flow such as a pumping station can be used. The difference between the total flow and the metered flow to consumers is the amount of unmetered flow. A portion of the unmetered flow would be due to leaks. If field tests are conducted during times when there is no unmetered flow for fire protection or maintenance, such as late at night without a fire event, it is likely that much of the calculated unmetered flow would be due to leaks.

$$Q_{\text{leaks}} = Q_{\text{total}} - Q_{\text{metered}} - Q_{\text{fire and maintenance}}$$

The calculated flow rate due to leaks is then distributed throughout the water system as many small leaks and/or a few large leaks at system nodes. Various methods are used to distribute small leaks throughout the pipe network.

Conventional method—The simplest method to distribute the leakage flow would be to divide the calculated leakage flow among the nodes in the water system as an added demand. The baseline leakage rate is typically set to be proportional to the metered demand. This method does not take into account the pressure-dependent flow rate of the leak.

Uniform emitter coefficient—If the actual locations of the leaks are unknown, a uniform emitter coefficient (K) can be used for each node to distribute the leakage flow. The emitter coefficient is estimated so that the total leakage flow rate is distributed based on the pressure head at each node. This ensures that the leakage flow rate is pressure-dependent.

$$Q_{leaks} = K \sum_{i=1}^N P_i^n$$

where

N = total number of demand nodes,

P_i = Pressure head at node i , and

n = exponent.

Proportional methods—There are other various methods that use an emitter coefficient, the node pressure head, and proportional values for the pipe length, and metered consumption. These methods use weighted factors for pipe length, and metered consumption in addition to the pressure-dependent leakage flow rate at each node.

6.2 Energy Losses

Friction Losses

The hydraulic theory behind friction losses is the same for pressure piping as it is for open-channel hydraulics. The most commonly used methods for determining head losses in pressure piping systems are the Hazen–Williams equation and the Darcy–Weisbach equation, both discussed in Chapter 1. Many of the general friction loss equations can be simplified and revised because of the following assumptions that can be made for a pressure pipe system:

- Pressure piping is almost always circular, so the flow area, wetted perimeter, and hydraulic radius can be directly related to diameter.
- Pressure systems flow full (by definition) throughout the length of a given pipe, so the friction slope is constant for a given flow rate. This means that the energy grade and hydraulic grade drop linearly in the direction of flow.
- Because the flow rate and cross-sectional area are constant, the velocity must also be constant. By definition, then, the energy grade line and hydraulic grade line (HGL) are parallel, separated by the constant velocity head.

These simplifications allow for pressure pipe networks to be analyzed much more quickly than systems of open channels or partially full gravity piping.

Several hydraulic components that are unique to pressure piping systems, such as regulating valves and pumps, add complexity to the analysis.

Minor Losses

Localized areas of increased turbulence cause energy losses within a pipe, creating a drop in the energy and hydraulic grades at that point in the system. These disruptions are often caused by valves, meters, or fittings (such as the pipe entrance in Figure 6.2), and are generally called **minor losses**. These minor losses are often negligible relative to friction losses and may be ignored during analysis.

Although the term “minor” is a reasonable generalization for most large-scale water distribution models, these losses may not always be as minor as the name implies. In piping systems that contain numerous fittings relative to the total length of pipe, such as heating or cooling systems, the minor losses may actually have a significant impact on the energy loss.

The equation most commonly used for determining the loss in a fitting, valve, meter, or other localized component is:

$$H_m = K \frac{V^2}{2g}$$

where

H_m = minor loss (m, ft),

K = minor loss coefficient for the specific fitting,

V = velocity (m/s, ft/s), and

g = gravitational acceleration (m/s², ft/s²).

Typical values for the fitting loss coefficient are included in Table 6.1. As can be seen with similar fitting types, the K -value is highly dependent on bend radius, contraction ratios, and so forth. Gradual transitions create smoother flow lines and smaller head losses than sharp transitions because of the increased turbulence and eddies that form near a sharp change in the flow pattern. Figure 6.2 shows flow lines for a pipe entrance with and without rounding.

Table 6.1: Typical Fitting K Coefficients

Fitting	K-Value	Fitting	K-Value
Pipe Entrance		90° Smooth bend	
Bellmouth	0.03–0.05	Bend radius/ $D = 4$	0.16–0.18
Rounded	0.12–0.25	Bend radius/ $D = 2$	0.19–0.25
Sharp edged	0.50	Bend radius/ $D = 1$	0.35–0.40
Projecting	0.80	Mitered bend	

Table 6.1: (Continued)

Fitting	K-Value	Fitting	K-Value
Contraction—sudden		$\theta = 15^\circ$	0.05
$D_2/D_1 = 0.80$	0.18	$\theta = 30^\circ$	0.10
$D_2/D_1 = 0.50$	0.37	$\theta = 45^\circ$	0.20
$D_2/D_1 = 0.20$	0.49	$\theta = 60^\circ$	0.35
Contraction—conical		$\theta = 90^\circ$	0.80
$D_2/D_1 = 0.80$	0.05	Tee	
$D_2/D_1 = 0.50$	0.07	Line flow	0.30–0.40
$D_2/D_1 = 0.20$	0.08	Branch flow	0.75–1.80
Expansion—sudden		Cross	
$D_2/D_1 = 0.80$	0.16	Line flow	0.50
$D_2/D_1 = 0.50$	0.57	Branch flow	0.75
$D_2/D_1 = 0.20$	0.92	45° Wye	
Expansion—conical		Line flow	0.30
$D_2/D_1 = 0.80$	0.03	Branch flow	0.50
$D_2/D_1 = 0.50$	0.08		
$D_2/D_1 = 0.20$	0.13		

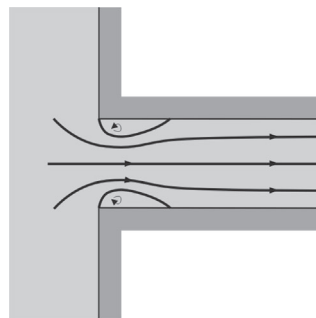
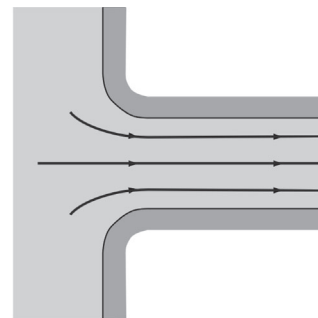
Sharp entrance: $K = 0.50$ Rounded entrance: $K = 0.03$

Figure 6.2: Flow Lines in Minor Losses.

6.3 Energy Gains—Pumps

Pumps are an integral part of many pressure systems and are an important part of modeling head change in a network. Pumps add energy (head gains) to the flow to counteract head losses and hydraulic grade differentials within the system. There are several types of pumps that are used for various purposes; pressurized water systems typically have centrifugal pumps.

A centrifugal pump is defined by its **characteristic curve**, which relates the pump head (head added to the system) to the flow rate. To model the behavior of the pump system, additional information is needed to ascertain the actual point at which the pump will be operating.

The **system operating point** is the point at which the pump curve crosses the **system curve**—the curve representing the static lift (H_s) and head losses (H_L) due to friction and minor losses. When these curves are superimposed (as in Figure 6.3), the operating point is easily located.

As water surface elevations and demands throughout the system change, the static head (H_s) and head losses (H_L) vary. These changes cause the system curve to move around, whereas the pump characteristic curve remains constant. These shifts in the system curve result in a shifting operating point over time.

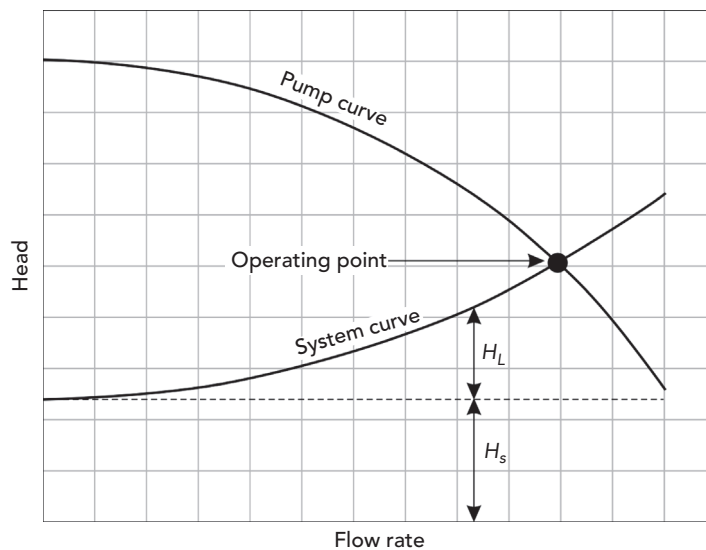


Figure 6.3: System Operating Point.

Variable-Speed Pumps

A centrifugal pump's characteristic curve is fixed for a given motor speed and impeller diameter but can be determined for any speed and any diameter by applying the affinity laws. For variable-speed pumps, these affinity laws are presented as:

$$\frac{Q_1}{Q_2} = \frac{n_1}{n_2} \quad \text{and} \quad \frac{H_1}{H_2} = \left(\frac{n_1}{n_2} \right)^2$$

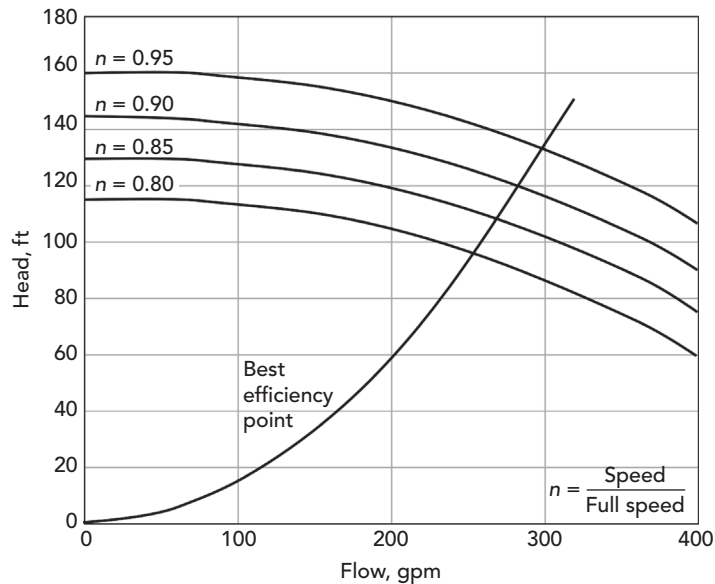


Figure 6.4: Relative Speed Factors for Variable-Speed Pumps.

where

Q = pump flow rate (m^3/s , ft^3/s),

H = pump head (m, ft), and

n = pump speed (rpm).

Thus pump discharge rate is proportional to pump speed, and the pump discharge head is proportional to the square of the speed. Using this relationship, once the pump curve is known, the curve at another speed can be predicted. Figure 6.4 illustrates the affinity laws applied to a variable-speed pump. The line labeled “Best efficiency point” indicates how the best efficiency point changes at various speeds.

Constant Horsepower Pumps

During preliminary studies, the exact characteristics of the pump may not be known. In these cases, the assumption is often made that the pump is adding energy to the water at a constant rate. Horsepower is input as the actual power added to the system, and not the rated horsepower of the motor (because there is a loss of efficiency in the motor, and motors usually run at less than their rated capacity). Specifying a pump as a constant horsepower pump means that the pump will add the same power to the water at any flow rate. Although this assumption is useful for some applications, a constant horsepower pump should only be used for preliminary studies.

6.4 Control Valves

There are several types of valves that may be present in a typical pressurized pipe system. These valves have different behaviors and different applications, but all valves are used to automatically control parts of the system, opening, closing, or throttling to achieve the desired result.

Check Valves (CVs)

Check valves (CVs) are used to maintain flow in one direction only by closing when the flow begins to reverse. When the flow is in the same direction as the specified direction of the CV, the valve is considered to be fully open.

Flow Control Valves (FCVs)

A *flow control valve (FCV)* limits the flow rate through the valve to a specified value in a specified direction. A flow rate is used to control the operation of an FCV. These valves are commonly found in areas where a water district has contracted with another district or a private developer to limit the maximum demand to a value that will not adversely affect the provider's system.

Pressure-Reducing Valves (PRVs)

Pressure-reducing valves (PRVs) are often used to separate pressure zones in water distribution networks. These valves prevent the pressure downstream from exceeding a specified level, in order to avoid pressures and flows that could otherwise have undesirable effects on the system. A pressure or a hydraulic grade is used to control the operation of a PRV.

Pressure-Sustaining Valves (PSVs)

Pressure-sustaining valves (PSVs) maintain a specified pressure upstream of the valve. Similar to the other regulating valves, PSVs are often used to ensure that pressures in the system (upstream, in this case) will not drop to unacceptable levels. A pressure or a hydraulic grade is used to control the operation of a PSV.

Pressure Breaker Valves (PBVs)

Pressure breaker valves create a specified head loss across the valve and are often used to model components that cannot be easily modeled using standard minor loss elements.

Throttle Control Valves (TCVs)

Throttle control valves (TCVs) simulate minor loss elements, the head loss characteristics of which change over time. With a TCV, the minor loss K is adjusted based on some other system flow or head.

6.5 Pipe Networks

In practice, pipe networks consist not only of pipes but also of miscellaneous fittings, services, storage tanks, reservoirs, meters, regulating valves, pumps, and electronic and mechanical controls. For modeling purposes, these system elements can be organized into four fundamental categories:

- **Junction nodes:** Junctions are specific points (nodes) in the system where an event of interest is occurring. Junctions include points where pipes intersect, points where major demands on the system (such as a large industry, a cluster of houses, or a fire hydrant) are located, or critical points in the system where pressures are important for analysis purposes.
- **Boundary nodes:** Boundaries are nodes in the system where the hydraulic grade is known, and they define the initial hydraulic grades for any computational cycle. They set the HGL used to determine the condition of all other nodes during system operation. Boundary nodes are elements such as tanks, reservoirs, and pressure sources. A model must contain at least one boundary node for the HGLs and pressures to be calculated.
- **Links:** Links are system components such as pipes that connect to junctions or boundaries and control the flow rates and energy losses (or gains) between nodes.
- **Pumps and valves:** Pumps and valves are similar to nodes in that they occupy a single point in space, but they also have link properties because head changes occur across them.

An event or condition at one point in the system can affect all other locations in the system. Although this fact complicates the approach that the engineer must take to find a solution, there are some governing principles that drive the behavior of the network, such as the conservation of mass and the conservation of energy.

Conservation of Mass—Flows and Demands

This principle is a simple one. At any node in the system under incompressible flow conditions, the total volumetric or mass flow entering must equal the mass flow leaving (plus the change in storage).

Separating the total volumetric flow into flows from connecting pipes, demands, and storage, we obtain the following equation:

$$\sum Q_{in} \Delta t = \sum Q_{out} \Delta t + \Delta \forall_s$$

where

$\sum Q_{in}$ = total flow into the node,

$\sum Q_{out}$ = total flow out of the node,

$\Delta \forall_s$ = change in storage volume, and

Δt = change in time.

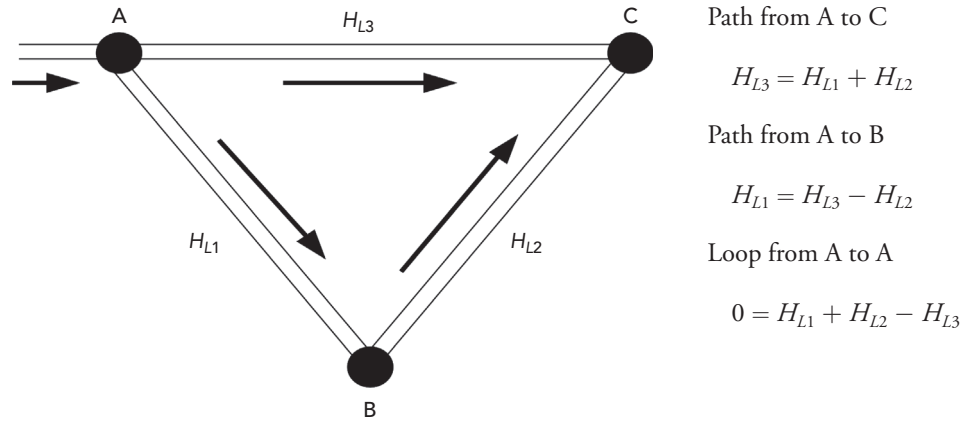


Figure 6.5: Conservation of Energy.

Conservation of Energy

Chapter 1 introduced the application of the energy equation to hydraulic analysis. The principle of conservation of energy dictates that the head losses through the system must balance at each point (Figure 6.5). For pressure networks, this means that the total head loss between any two nodes in the system must be the same regardless of the path taken between the two points. The head loss must be “sign consistent” with the assumed flow direction (that is, head loss occurs in the direction of flow, and head gain occurs in the direction opposite that of the flow).

Although the equality can become more complicated with minor losses and controlling valves, the same basic principle can be applied to any path between two points. As shown in Figure 6.5, the combined head loss around a loop must equal zero in order to compute the same hydraulic grade for a given point.

6.6 Network Analysis

Steady-State Network Hydraulics

Steady-state analysis is used to determine the operating behavior of a system at a specific point in time, or under steady-state conditions. This type of analysis can be useful in discovering the short-term effect of fire flows or average demand conditions on the system.

For this type of analysis, the network equations are determined and solved with tanks being treated as fixed grade boundaries. The results that are obtained from this type of analysis are instantaneous values and may not be representative of the values of the system a few hours—or even a few minutes—later in time.

Extended-Period Simulation

An extended-period simulation is used to determine the behavior of the system over time. This type of analysis allows the user to model tanks filling and draining, regulating valves opening and closing, and pressures and flow rates changing throughout the system in response to varying demand conditions and automatic control strategies formulated by the modeler.

While a steady-state model may tell the user whether the system has the capability to meet a specific demand, an extended-period simulation indicates whether the system has the ability to provide acceptable levels of service over a period of minutes, hours, or days. Extended-period simulations can also be used for energy consumption and cost studies, as well as for water quality modeling.

Data requirements for an extended-period simulation go beyond what is needed for a steady-state analysis. The user must determine water usage patterns, provide more detailed tank information, and enter operational rules for pumps and valves.

6.7 Water Quality Analysis

In the past, water distribution systems were designed and operated with little consideration of water quality, due in part to the difficulty and expense of analyzing a dynamic system. The cost of extensive sampling and the complex interaction between fluids and constituents makes numeric modeling the ideal method for predicting water quality.

To predict water quality parameters, an assumption is made that there is complete mixing across finite distances, such as at a junction node or in a short segment of pipe. Complete mixing is essentially a mass balance given by:

$$C_a = \frac{\sum Q_i C_i}{\sum Q_i}$$

where

C_a = average (mixed) constituent concentration,

Q_i = inflow rates, and

C_i = constituent concentrations of the inflows.

Age

Water age provides a general indication of the overall water quality at any given point in the system. Age is typically measured from the time that the water enters the system from a tank or reservoir until it reaches a junction.

Along a given link, water age is computed as:

$$A_j = A_{j-1} + \frac{x}{V}$$

where

A_j = age of water at j th node,

x = distance from node $j - 1$ to node j , and

V = velocity from node $j - 1$ to node j .

If there are several paths for water to travel to the j th node, the water age is computed as a weighted average using the equation:

$$AA_j = \frac{\sum Q_i [AA_i + (x / V)_i]}{\sum Q_i}$$

where

AA_j = average age at the node immediately upstream of node j and

Q_i = flow rate to the j th node from the i th node.

Trace

Identifying the origin of flow at a point in the system is referred to as **flow tracking** or **trace modeling**. In systems that receive water from more than one source, trace studies can be used to determine the percentage of flow from each source at each point in the system. These studies can be very useful in delineating the area influenced by an individual source, observing the degree of mixing of water from several sources, and viewing changes in origins over time.

Constituents

Reactions can occur within pipes that cause the concentration of substances to change as the water travels through the system. Based on conservation of mass for a substance within a link (for extended-period simulations only):

$$\frac{\partial c}{\partial t} = V \frac{\partial c}{\partial x} + \theta(c)$$

where

c = substance concentration as a function of distance and time,

t = time increment,

V = velocity,

x = distance along the link, and

$\theta(c)$ = substance rate of reaction within the link.

In some applications, there is an additional term for dispersion, but this term is usually negligible (plug flow is assumed through the system).

Assuming that complete and instantaneous mixing occurs at all junction nodes, additional equations can be written for each junction node with the following conservation of mass equation:

$$C_k|_{x=0} = \frac{\sum Q_j C_j|_{x=L} + Q_e C_e}{\sum Q_j + Q_e}$$

where

C_k = concentration at node k ,

j = pipe flowing into node k ,

L = length of pipe j ,

Q_j = flow in pipe j ,

C_j = concentration in pipe j ,

Q_e = external source flow into node k , and

C_e = external source concentration into node k .

Once the hydraulic model has solved the network, the velocities and the mixing at the nodes are known. Using this information, the water quality behavior can be derived using a numerical method.

Initial Conditions

Just as a hydraulic simulation starts with some amount of water in each storage tank, initial conditions must be set for a water age, trace, or constituent concentration analysis. These initial water quality conditions are usually unknown, so the modeler must estimate these values from field data, a previous water quality model, or some other source of information.

To overcome the problem of unknown initial conditions at the vast majority of locations within the water distribution model, the duration of the analysis must be long enough for the system to reach equilibrium conditions. Note that a constant value does not have to be reached for equilibrium to be achieved; rather, equilibrium conditions are reached when a repeating pattern in age, trace, or constituent concentration is established.

Pipes usually reach equilibrium conditions in a short time, but storage tanks are much slower to show a repeating pattern. For this reason, extra care must be taken when setting a tank's initial conditions, in order to ensure the model's accuracy.

Numerical Methods

There are several theoretical approaches available for solving water quality models. These methods can generally be grouped as either Eulerian or Lagrangian in nature, depending on the volumetric control approach that is taken. Eulerian models divide the system into fixed pipe segments, and then

track the changes that occur as water flows through these segments. Lagrangian models also break the system into control volumes but then track these water volumes as they travel through the system. This chapter presents two alternative approaches for performing water quality constituent analyses.

Discrete Volume Method

The Discrete Volume Method (DVM) is an Eulerian approach that divides each pipe into equal segments with completely mixed volumes (Figure 6.6). Reactions are calculated within each segment, and the constituents are then transferred to the adjacent downstream segment. At nodes, mass and flow entering from all connecting pipes are combined (assuming total mixing). The resulting concentration is then transported to all adjacent downstream pipe segments. This process is repeated for each water quality time step until a different hydraulic condition is encountered. When this occurs, the pipes are divided again under the new hydraulic conditions, and the process continues.

Time-Driven Method

The Time-Driven Method (TDM) is an example of a Lagrangian approach (Figure 6.7). This method also breaks the system into segments, but rather than using fixed control volumes as in Eulerian methods, the concentration and size of water parcels are tracked as they travel through the pipes. With each time step, the farthest upstream parcel of each pipe elongates as water travels into the pipe, and the farthest downstream parcel shortens as water exits the pipe.

Similar to the DVM, the reactions of a constituent within each parcel are calculated, and the mass and flow entering each node are summed to determine

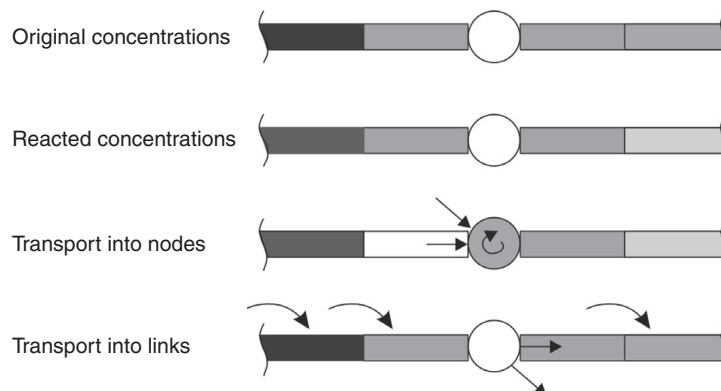


Figure 6.6: Eulerian DVM.

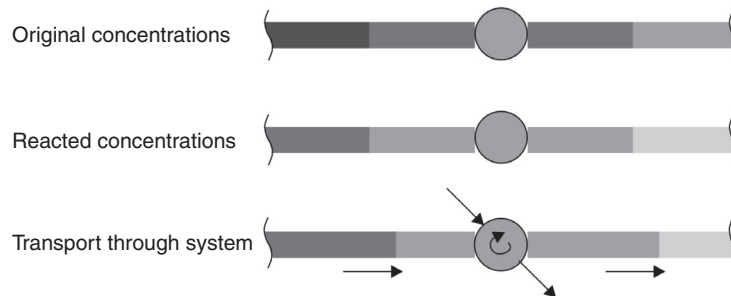


Figure 6.7: Lagrangian TDM.

the resulting concentration. If the resulting nodal concentration is significantly different from the concentration of a downstream parcel, a new parcel will be created rather than elongating the existing one. These calculations are repeated for each water quality time step until the next hydraulic change is encountered and the procedure begins again.

6.8 Automated Optimization

WaterGEMS has the capability to optimize a model based on field data or design criteria. Many times, water utility managers will use a model to make design decisions or gather field data to calibrate a model. This process is typically a trial-and-error approach where the modeler will modify a few parameters in a model to either compare design solutions based on cost or benefit or have the model better predict the real conditions. Because this can be very time-consuming, WaterGEMS has the capability to create many potential solutions and provide a measure of which solution is the “better” solution based on specific boundary conditions and input criteria.

WaterGEMS employs a genetic algorithm search method to find “better” solutions based on the principles of natural selection and biological reproduction. This genetic algorithm program first creates a population of trial solutions based on modeled parameters. The hydraulic solver then simulates each trial solution to predict the HGL and flow rates within the network and compares them to any input criteria. Based on this comparison, a goodness-to-fit value is assigned. This information is now used to create a new population of trial solutions which again are used to find new solutions. The program compares these solutions to the specific boundary conditions and input criteria until the goodness-to-fit value is optimized. In other words no better solution can be generated.

Model Calibration

Model calibration is the process of modifying parameters or values in a model so it better matches what is happening in the real system. The calibration of

water distribution models is very complicated. There are many unknown values and parameters that are needed at any one time to reduce the discrepancy between the model and real system. Many times the pipe roughness value is adjusted to have the model results match the measured or expected values in the real system. However, there are many other parameters that could influence the modeled results. The water demand at junctions and the status of pipes and valves in the system could also be adjusted when calibrating a model.

Calibrating a model relies on accurate field measurement data. Field measurements of pressures in the system, pipe flow rates, water levels in tanks, valve status, and pump operating status and speed are all used to calibrate models. Critical to all these measurements is the time for which the measurements are made. The time of these measurements must all be synchronized to the time frame of the model. In addition, because the conditions within a real system changes throughout the day or year, field data should be collected for many different conditions and times. The calibration process is used to adjust the model to simulate multiple demand loading and operational boundary conditions. Only then can the modeler be confident that the model is valid for many different conditions.

WaterGEMS uses Darwin Calibrator to assist in optimizing the model to match field measurement data. The Darwin Calibrator allows the modeler to input field data then request the software to determine the optimal solution of pipe roughness values, junction demands, or status (on/off). Pipes that have the same hydraulic characteristics where one roughness values is assigned to all pipes can be grouped together. Junctions can also be grouped based on the demand pattern and location. Caution should be used grouping pipes and junctions since this could greatly affect the model calibration accuracy.

System Design

The goal of water distribution system design is to maximize the benefits of the system while minimizing the cost. The optimal solution is a design that meets all the needs of the system at minimal cost. Some planning is needed to account for additional future needs of the system, including potential growth of the system in terms of demand and its location. The modeler must work with the system owner and planning groups to account for both the current and future needs.

WaterGEMS uses Darwin Designer to assist engineers to plan and design water distribution networks. The Darwin Designer can be used to size new pipe and/or rehabilitate old pipes to minimize cost, maximize benefit, or create a scenario to trade-off cost and benefit. The least cost optimization is used to determine the pipe material and size to satisfy the needed design requirements. The maximum benefit optimization is used to determine the most beneficial solution based on a known budget. The Darwin Designer will generate a number of solutions that meet the design requirements at minimal cost or maximum

benefit. In either case, the best solution of new pipe or rehabilitation of old pipe will be based upon the following input hydraulic criteria:

- minimum and maximum allowable pressures;
- minimum and maximum allowable pipe flow velocity;
- additional demand requirements; and
- pipe, pump, tank, valve, etc. status change requirements.

Critical to creating an accurate designed system is time and peak demand requirements. The peak demand and fire flow conditions are used to size pipes since the pipe network must work for all conditions. Using average demand values to size pipe without accurately accounting for peaking factors can create networks that are either undersized and will not deliver the required water needs, or oversized and much more expensive than need be. The daily and seasonal variations can also greatly affect the final design. Demand variations need to be synchronized in the model to accurately reflect what could happen in the real system.

6.9 WaterGEMS and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software.

What Does WaterGEMS Do?

WaterGEMS is a powerful, easy-to-use program that helps civil engineers design and analyze water distribution systems. It is also used by water utility managers to as a tool for the efficient operation of distribution systems. WaterGEMS provides intuitive access to the tools you need to model complex hydraulic situations. WaterGEMS's sophisticated modeling capabilities can

- perform steady-state, extended-period, and water quality simulations;
- analyze multiple time-variable demands at any junction node;
- model FCVs, PRVs, PSVs, pressure-breaking valves, and TCVs;
- model cylindrical and noncylindrical tanks and constant hydraulic grade source nodes
- track conservative and nonconservative chemical constituents;
- determine water source and age at any element in the system;
- estimate construction costs;
- simulate the operating cycles of constant or variable speed pumps;

- estimate the cost of pumping over any time period;
- use field measurement data to calibrate a model; and
- provide design solutions to maximize benefits at minimal cost.

WaterGEMS can be used as a stand-alone program, integrated with AutoCAD, or linked to a Geographical Information System via the GEMS component. The theory and background used in WaterGEMS are presented in this chapter and in the WaterGEMS online help system.

How Can You Use WaterGEMS?

WaterGEMS can analyze complex distribution systems under a variety of conditions. For a typical WaterGEMS project, you may be interested in determining system pressures and flow rates under average loading, peak loading, or fire flow conditions. Extended-period analysis tools also allow you to model the system's response to varying supply and demand schedules over a period of time; you can even track chlorine residuals or determine the source of the water at any point in the distribution system.

In summary, you can use WaterGEMS for pipe sizing

- pump sizing
- master planning
- operational studies
- rehabilitation studies
- vulnerability studies
- water quality studies.

WaterGEMS is a state-of-the-art software tool primarily for use in the modeling and analysis of water distribution systems. Although the emphasis is on water distribution systems, the methodology is applicable to any fluid system with the following characteristics:

- steady or slowly changing turbulent flow
- incompressible, Newtonian, single-phase fluid
- full, closed conduits (pressure system).

Examples of systems with these characteristics include potable water systems, sewage force mains, fire protection systems, well pumps, and raw water pumping.

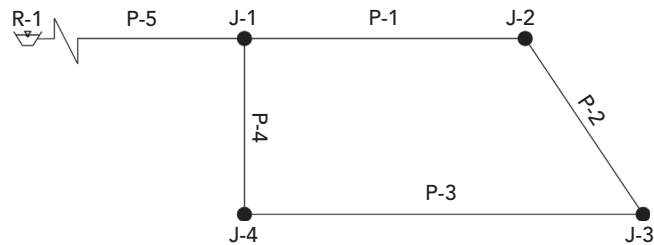
6.10 Problems

Solve the following problems using the WaterGEMS computer program.

1. The ductile iron pipe network shown below carries water at 20°C. Assume that the junctions all have an elevation of 0 m and the reservoir

is at 30 m. Use the Hazen–Williams formula ($C = 130$) and the pipe and demand data below to perform a steady-state analysis and answer the following questions:

- Which pipe has the lowest discharge? What is the discharge (in l/min)?
- Which pipe has the highest velocity? What is the velocity (in m/s)?
- Calculate the problem using the Darcy–Weisbach equation ($k = 0.26$ mm) and compare the results.
- What effect would raising the reservoir by 20 m have on the pipe flow rates? What effect would it have on the hydraulic grade lines at the junctions?



Schematic for Problem 1

Pipe and Junction Information for Problem 1

Pipe	Diameter (mm)	Length (m)	Junction	Demand (l/min)
P-1	150	50	J-1	570
P-2	100	25	J-2	660
P-3	100	60	J-3	550
P-4	100	20	J-4	550
P-5	250	760		

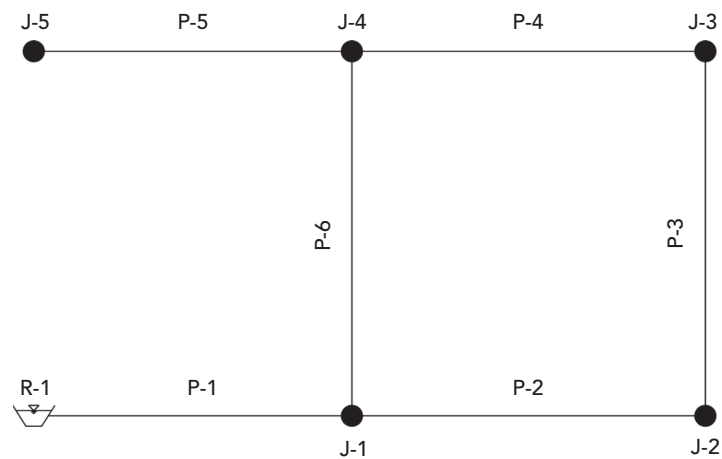
- A pressure gage reading of 288 kPa was taken at J-5 in the pipe network shown below. Assuming a reservoir elevation of 100 m, find the appropriate Darcy–Weisbach roughness height (to the hundredths place) to bring the model into agreement with these field records. Use the same roughness value for all pipes.

Pressure Piping Systems and Water Quality Analysis

Problems

The pipe and junction data are shown below.

- (a) What roughness factor yields the best results?
- (b) What is the calculated pressure at J-5 using this factor?
- (c) Other than the pipe roughness, what other factors could cause the model to disagree with field-recorded values for flow and pressure?

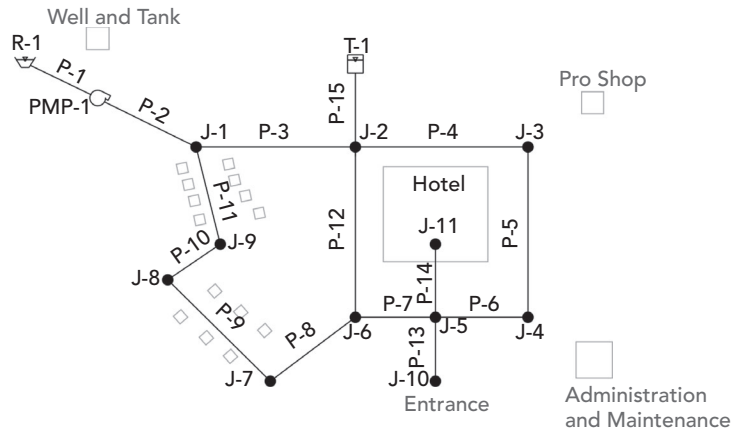


Schematic for Problem 2

Pipe and Junction Information for Problem 2

Pipe	Diameter (mm)	Length (m)	Junction	Elevation (m)	Demand (l/min)
P-1	250	1,525	J-1	55	950
P-2	150	300	J-2	49	1,060
P-3	150	240	J-3	58	1,440
P-4	150	275	J-4	46	1,175
P-5	150	245	J-5	44	980
P-6	200	230			

3. A distribution system is needed to supply water to a resort development for normal usage and emergency purposes (such as fighting a fire). The proposed system layout is shown in the following figure:



Proposed Network for Problem 3

The source of water for the system is a pumped well. The water is treated and placed in a ground-level tank (shown above as a reservoir because of its plentiful supply), which is maintained at a water surface elevation of 210 ft. The water is then pumped from this tank into the rest of the system.

The well system alone cannot efficiently provide the amount of water needed for fire protection, so an elevated storage tank is also needed. The bottom of the tank is at 376 ft (high enough to produce 35 psi at the highest node), and the top is approximately 20-ft higher. To avoid the cost of an elevated tank, this 80-ft diameter tank is located on a hillside, 2,000 ft away from the main system. Assume that the tank starts with a water surface elevation of 380 ft.

The pump was originally sized to deliver 300 gpm with enough head to pump against the tank when it is full. Three defining points on the pump curve are as follows: 0 gpm at 200 ft of head, 300 gpm at 180 ft of head, and 600 gpm at 150 ft of head. The pump elevation is assumed to be the same as the elevation at J-1, although the precise pump elevation is not crucial to the analysis.

The system is to be analyzed under several demand conditions with minimum and maximum pressure constraints. During normal operations, the junction pressures should be between 35 and 80 psi. Under fire flow conditions, however, the minimum pressure is allowed to drop to 20 psi. Fire protection is being considered both with and without a sprinkler system.

Demand alternatives: WaterGEMS enables you to store multiple demand alternatives corresponding to various conditions (such as average day and peak hour). This feature allows you run different scenarios that incorporate various demand conditions within a single project file without losing any input data.

Pressure Piping Systems and Water Quality Analysis

Problems

For an introduction and more information about scenarios and alternatives, see WaterGEMS's online help system and Appendices A and B.

Junction Information for Problem 3

Junction	Elevation (ft)	Average Day (gpm)	Peak Hour (gpm)	Minimum Hour (gpm)	Fire with Sprinkler (gpm)	Fire without Sprinkler (gpm)
J-1	250	0	0	0	0	0
J-2	260	0	0	0	0	0
J-3	262	20	50	2	520	800
J-4	262	20	50	2	520	800
J-5	270	0	0	0	0	800
J-6	280	0	0	0	0	800
J-7	295	40	100	2	40	40
J-8	290	40	100	2	40	40
J-9	285	0	0	0	0	0
J-10	280	0	0	0	360	160
J-11	270	160	400	30	160	160

Pipe network: The pipe network consists of the pipes listed in the following tables. The diameters shown are based on the preliminary design and may not be adequate for the final design. For all pipes, use ductile iron as the material and a Hazen–Williams *C*-factor of 130.

Pipe Information for Problem 3

Pipe	Diameter (in.)	Length (ft)
P-1	8	20
P-2	8	300
P-3	8	600
P-4	6	450
P-5	6	500
P-6	6	300
P-7	8	250
P-8	6	400

Pipe	Diameter (in.)	Length (ft)
P-9	6	400
P-10	6	200
P-11	6	500
P-12	8	500
P-13	6	400
P-14	6	200
P-15	10	2,000

To help keep track of important system characteristics (such maximum velocity and lowest pressure), you may find it helpful to keep a table such as the following:

Results Summary for Problem 3

Variable	Average Day	Peak Hour	Minimum Hour	Fire with Sprinkler	Fire without Sprinkler
Node w/low pressure					
Low pressure (psi)					
Node w/high pressure					
High pressure (psi)					
Pipe w/max. velocity					
Max. velocity (ft/s)					
Tank in/out flow (gpm)					
Pump discharge (gpm)					

Another way to quickly determine the performance of the system is to color-code the pipes according to some indicator. In hydraulic design, a good performance indicator is often the velocity in the pipes. Pipes consistently flowing below 0.5 ft/s may be oversized. Pipes with velocities over 5 ft/s are fairly heavily stressed, and those with velocities above 8 ft/s are usually bottlenecks in the system under that flow pattern. Color-code the system using the ranges in the table below. After you define the color-coding, place a legend in the drawing.

Color-Coding Range for Problem 3

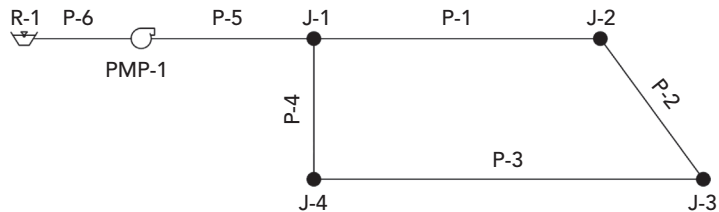
Max. Velocity (ft/s)	Color
0.5	Magenta
2.5	Blue
5.0	Green
8.0	Yellow
20.0	Red

- Fill in or reproduce the Results Summary table after each run to get a feel for some of the key indicators during various scenarios.
- For the average day run, what is the pump discharge?
- If the pump has a best efficiency point at 300 gpm, what can you say about its performance on an average day?

Pressure Piping Systems and Water Quality Analysis

Problems

- (d) For the peak hour run, the velocities are fairly low. Does this mean you have oversized the pipes? Explain.
 - (e) For the minimum hour run, what was the highest pressure in the system? Why would you expect the highest pressure to occur during the minimum hour demand?
 - (f) Was the system (as currently designed) acceptable for the fire flow case with the sprinkled building? On what did you base this decision?
 - (g) Was the system (as currently designed) acceptable for the fire flow case with all the flow provided by hose streams (no sprinklers)? If not, how would you modify the system so that it will work?
4. A ductile iron pipe network ($C = 130$) is shown below. Use the Hazen–Williams equation to calculate friction losses in the system. The junctions and pump are at an elevation of 5 ft and all pipes are 6 in. in diameter. (*Note:* Use a standard, three-point pump curve. The data for the pump, junctions, and pipes are in the tables below.) The water surface of the reservoir is at an elevation of 30 ft.



Schematic for Problem 4

Pump Information for Problem 4

Head (ft)	Flow (gpm)
200	0
175	1,000
100	2,000

Junction Information for Problem 4

Junction Label	Demand (gpm)
J-1	400
J-2	550
J-3	550
J-4	350

Pipe Information for Problem 4

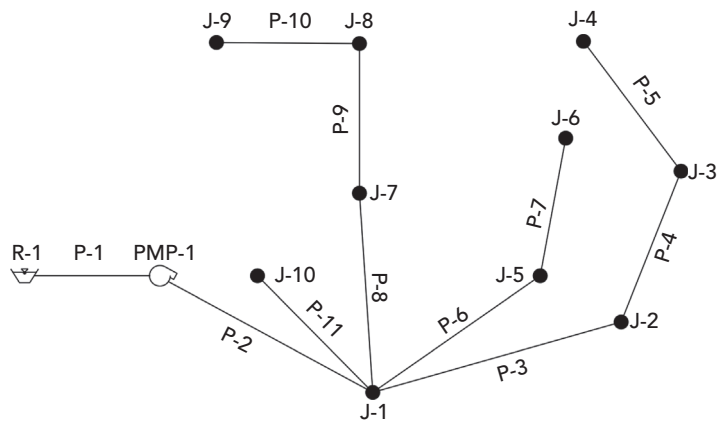
Pipe Label	Length (ft)
P-1	78
P-2	40
P-3	90
P-4	39
P-5	10
P-6	10

- (a) What are the resulting flows and velocities in the pipes?
 - (b) What are the resulting pressures at the junction nodes?
 - (c) Place a check valve on pipe P-3 such that the valve only allows flow from J-3 to J-4. What happens to the flow in pipe P-3? Why does this occur?
 - (d) When the check valve is placed on pipe P-3, what happens to the pressures throughout the system?
 - (e) Remove the check valve on pipe P-3. Place a 6-in. flow control valve node at an elevation of 5 ft on pipe P-3. The FCV should be set so that it only allows a flow of 100 gpm from J-4 to J-3 (Hint: a check valve is a pipe property). What is the resulting difference in flows in the network? How are the pressures affected?
 - (f) Why doesn't the pressure at J-1 change when the FCV is added?
 - (g) What happens if you increase the FCV's allowable flow to 2,000 gpm? What happens if you reduce the allowable flow to zero?
5. A local country club has hired you to design a sprinkler system that will water the greens of their nine-hole golf course. The system must be able to water all nine holes at once. The water supply has a water surface elevation of 10 ft. All pipes are PVC ($C = 150$, use the Hazen–Williams equation to determine friction losses). Use a standard, three-point pump curve for the pump, which is at an elevation of 5 ft. The flow at the sprinkler is modeled using an emitter coefficient. The data for the junctions, pipes,

Pressure Piping Systems and Water Quality Analysis

Problems

and pump curve are given in the tables that follow. The initial network layout is shown below.



Schematic for Problem 5

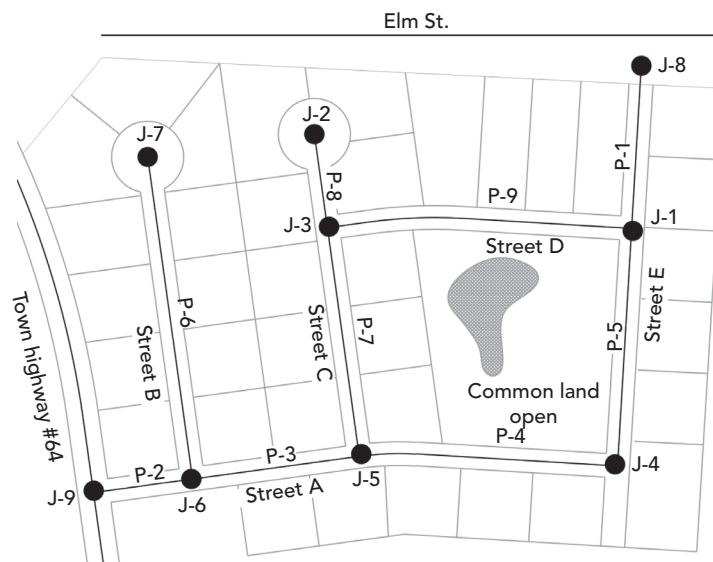
Junction and Pipe Information for Problem 5

Junction Label	Emitter Coefficient (gpm/psi ^{0.5})	Elevation (ft)	Pipe Label	Diameter (in.)	Length (ft)
J-1	—	10	P-1	4	10
Hole 1	8	7	P-2	4	1,000
Hole 2	10	7	P-3	4	800
Hole 3	15	40	P-4	3	750
Hole 4	12	5	P-5	3	500
Hole 5	8	5	P-6	3	700
Hole 6	8	15	P-7	2	400
Hole 7	10	20	P-8	4	800
Hole 8	15	10	P-9	3	500
Hole 9	8	12	P-10	2	400
			P-11	2	500

Pump Information for Problem 5

Head (ft)	Flow (gpm)
170	0
135	300
100	450

- (a) Determine the discharge at each hole.
 - (b) What is the operating point of the pump?
6. A subdivision of 36 homes is being constructed in a new area of town. Each home will require 1.7 l/s during peak periods. All junction nodes are 192 m in elevation. All pipes are ductile iron ($C = 130$, use the Hazen–Williams equation to determine the friction losses in the pipe). The current lot and network layout is shown below.



Schematic for Problem 6

Junction and Pipe Information for Problem 6

Junction Label	Number of Lots Served
J-1	5
J-2	4
J-3	4
J-4	5
J-5	6
J-6	6
J-7	6

Pipe Label	Length (m)	Diameter (mm)
P-1	60.0	150
P-2	60.0	150
P-3	110.5	150
P-4	164.0	150
P-5	152.5	150
P-6	204.0	100
P-7	148.0	150
P-8	61.0	100
P-9	194.0	150

Pressure Piping Systems and Water Quality Analysis

Problems

Currently, a model of the entire water system does not exist. However, hydrant tests were conducted using hydrants located on two water mains, one in Town Highway 64 and the other in Elm Street. The following data were obtained:

Town Highway 64 Hydrant Test
Static pressure 310.3 kPa
Residual pressure 98.5 kPa at 32 l/s
Elevation of pressure gauge 190 m

Elm Street Hydrant Test
Static pressure 413.7 kPa
Residual pressure 319.3 kPa at 40 l/s
Elevation of pressure gauge 191.5 m

The subdivision will connect to existing system mains in these streets at nodes J-8 and J-9. (Hint: Model the connection to an existing water main with a reservoir and a pump.)

- (a) What are the demands at each of the junction nodes? What is the total demand?
 - (b) Does the present water distribution system have enough capacity to supply the new subdivision?
 - (c) Which connection to the existing main is supplying more water to the subdivision? Why?
 - (d) Are the proposed pipe sizes adequate to maintain velocities between 0.15 and 2.44 m/s, and pressures of at least 140 kPa?
 - (e) Would the subdivision have enough water if only one connection were used? If so, which one?
 - (f) What do you think are some possible pitfalls of modeling two connections to existing mains within the same system, as opposed to modeling back to the water source?
- 7) Use the pipe sizes given in the table below for the subdivision in Problem 6.

Pipe Information for Problem 7

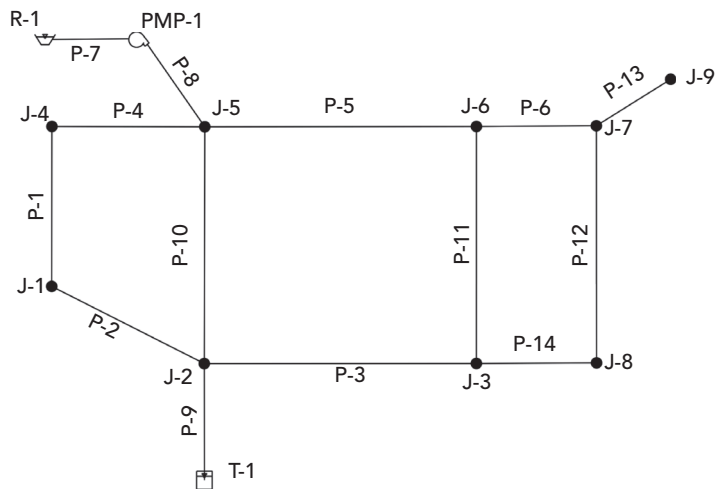
Pipe Label	Diameter (mm)
P-1	200
P-2	150
P-3	150
P-4	150
P-5	150
P-6	150
P-7	150
P-8	150
P-9	150

City ordinances require the following:

The pressure at the fire flow discharge and at other points in the distribution system cannot fall below 125 kPa during a fire flow of 34 l/s. (Hint: The total flow at the fire flow node does not need to include the baseline demand.)

- (a) If a residential fire occurs at J-7, would the current system be able to meet the fire flow requirements set by the city?
 - (b) If not, what can be done to increase the available flow to provide adequate fire flow to that hydrant?
 - (c) If a fire flow is placed at J-4, does the system meet the requirements with the proposed improvements? Without the proposed improvements?
8. A local water company is concerned with the water quality within its water distribution network. They want to determine the age and the chlorine concentration of the water as it exits the system at different junctions. The water surface at the reservoir is 70 m.

Chlorine is injected into the system at the source of flow, R-1, at a concentration of 1 mg/l. It has been determined through a series of bottle tests that the average bulk reaction rate of the chlorine in the system (including all pipes and tanks) is approximately $-0.5/\text{day}$.



Schematic for Problem 8

The cylindrical tank has a diameter of 15 m. The base and minimum elevations are 99 m. The maximum elevation is 104 m, and the initial elevation is 103.4 m.

Pressure Piping Systems and Water Quality Analysis

Problems

Pump Information for Problem 8

Head (m)	Discharge (l/min)	Controls
40	0	Off if node T-1 above 103.5 m On if node T-1 below 100.5 m
35	3,000	
24	6,000	

Stepwise Demand Pattern Data for Problem 8

Time from Start (h)	Multiplier	Time from Start (h)	Multiplier
0	0.80	13	1.30
1	0.60	14	1.40
2	0.50	15	1.50
3	0.50	16	1.60
4	0.55	17	1.80
5	0.60	18	1.80
6	0.80	19	1.40
7	1.10	20	1.20
8	1.50	21	1.00
9	1.40	22	0.90
10	1.30	23	0.80
11	1.40	24	0.80
12	1.40		

Junction Data for Problem 8

Junction	Elevation (m)	Demand (l/min)
J-1	73	151
J-2	67	227
J-3	81	229
J-4	56	219
J-5	67	215
J-6	73	219
J-7	55	215
J-8	84	180
J-9	88	151

Pipe Data for Problem 8

Pipe	Length (m)	Diameter (mm)	Roughness
P-1	300	200	130
P-2	305	200	130
P-3	300	200	130
P-4	200	200	130
P-5	300	300	130
P-6	200	200	130
P-7	1	300	130
P-8	5,000	300	130
P-9	300	300	130
P-10	500	200	130
P-11	500	200	130
P-12	500	200	130
P-13	150	150	130
P-14	200	200	130

- Perform an age analysis on the system using a duration of 300 h and a time step of 2 h. Fill in the results table, indicating the maximum water age at each junction and tank after the system reaches equilibrium (a pattern of average water age vs. time becomes evident). What point in the system generally has the oldest water? Explain why the water is oldest at this location.
- Perform a constituent analysis using the same duration and time step as in part (a). Fill in the results table, indicating the minimum chlorine concentration for each junction and tank after the system has reached equilibrium (a pattern of concentration versus time becomes evident). What point in the system has the lowest chlorine concentration? Explain why the chlorine residual is lowest at this location.

Results Table for Problem 8

Junction	J-1	J-2	J-3	J-4	J-5	J-6	J-7	J-8	J-9	T-1
Age (h)										
Chlorine Concentration (mg/l)										

Pressure Piping Systems and Water Quality Analysis

Problems

- (c) From the above table and graphs of demand, age, and concentration versus time generated within WaterGEMS, determine the following correlations:
 - (1) age and chlorine concentration
 - (2) demand and chlorine concentration at a junction
 - (3) demand and water age at a junction
 - (d) Why is it necessary to run the model for such a long time? Do you feel that 300 h is too long or too short a time period for testing the model? Why?
9. A planning commission has indicated a new industry may be connected to the water system described in Problem 8. You are to determine the pipe diameters in the network to minimize the installation cost assuming all the pipes are ductile iron. Use the Darwin Designer to determine the total cost and size each pipe for each of the following conditions. Use the pipe cost information from Tutorial 4 for the ductile iron pipe.
- (a) Size the pipes using a demand multiplier (peaking factor) of 3.2. The pressure must remain between 170 and 550 kPa during peak demand. Exclude pipes P-7 and P-8 in your analysis when determining the pipe sizes. Hint: you will need to specify an additional demand of zero (0 l/min) with the default pressure constraints at the junctions or a fatal error will occur.
 - (b) It is expected that a new industry with an expected additional demand of 2,000 l/min with a required minimum pressure of 260 kPa will be added to the system. It could be tapped into the network at either junction 6, 7, or 8. Size the pipes for the conditions in part (a) above along with the industry added to all proposed junctions. You will need to analyze the network three times, once for the industry at J-6, again with the industry at J-7, then finally with the industry at J-8.
 - a. Indicate which option(s) would work.
 - b. Which junction should the industry be tapped into to be the least costly and what is the expected cost?
 - c. What is the size of each pipe for the best solution for the least costly option with the industry added?
 - d. What is the calculated minimum pressure at the industry for the best solution?

7

Hydraulic Transients in Distribution Systems

7.1 Hydraulic Transients

A hydraulic transient occurs when there is a sudden change in flow. Hydraulic transient flow occurs between the initial steady-state condition and the final steady-state condition after the change has occurred. The most common cause of transient flow is when the fluid velocity suddenly changes due to a change to a system component such as valve closure, pump shut down, or pump failure. The result of transient flow is a series of pressure waves traveling at approximately the speed of sound through the distribution system that can temporarily increase or decrease the system pressures to values that are much larger or much smaller than pressures expected during steady-state conditions. A hydraulic transient is sometimes also referred to as a surge or water hammer.

Hydraulic transients are important in water distribution systems because they can cause pipe failure (both rupture and collapse), vibrations, damage to pipe support structures and joints, vapor cavities, and negative (vacuum) pressure regions. In regions with a negative pressure, nontreated water from outside the pipe can be pulled into it and contaminate the water distribution system.

Hydraulic transients can also occur in pressurized wastewater force mains, surcharging sewer systems, hydropower piping systems, slurry or oil pumping, and industrial systems such as heaters, boilers, steam, coolers, and other cogeneration systems.

Transient Flow

Transient flow is the transition from an initial flow condition to the final flow condition due to a change in the system. A simple illustration of this is when a valve suddenly closes in a single pipe fed by a tank (Figure 7.1). When the valve in the pipe instantaneously closes, the fluid velocity suddenly drops behind the valve. A positive pressure wave is formed and travels up the pipe toward the tank.

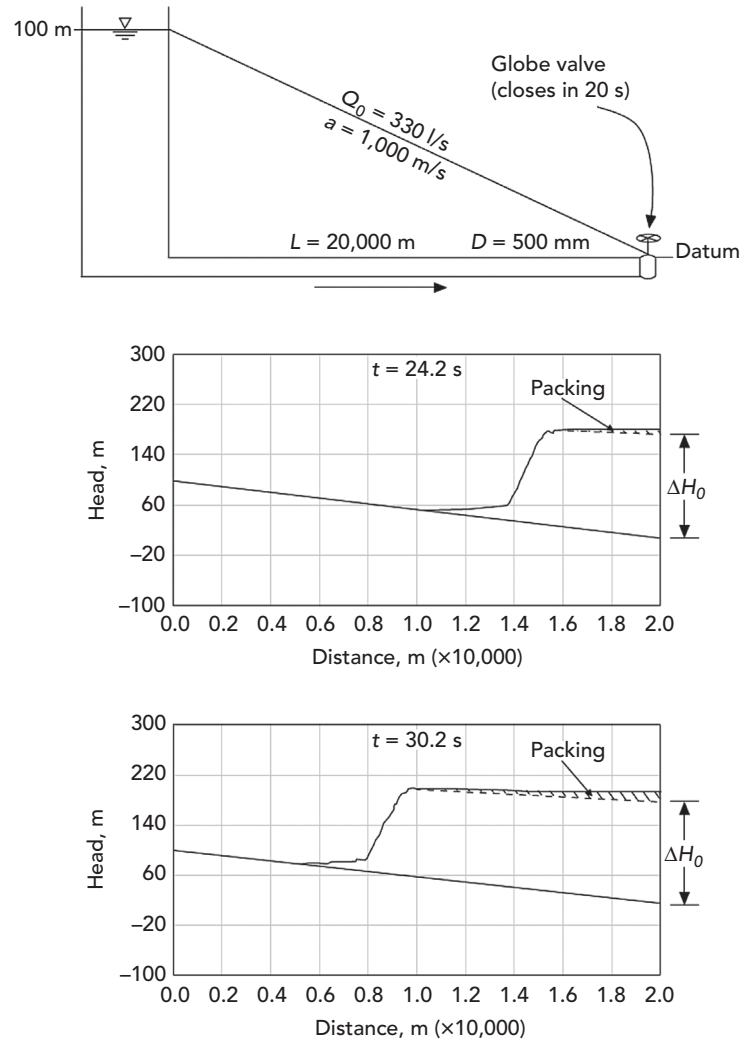


Figure 7.1: Pressure wave traveling up a pipe after valve closure.

As the pressure wave in the pipe moves toward the tank, the velocity behind the wave is zero and the velocity in front of the wave is approximately equal to the initial velocity. When the pressure wave reaches the tank, water from the pipe reenters the tank causing a small increase of the water level in the tank. At that instant, the pressure wave reaches the tank and the pipe velocity is zero, but the head in the pipe is higher than in the tank. This head imbalance causes flow to reverse in the pipe and the pressure in the pipeline to drop, creating a “pressure reduction wave” that moves back toward the valve. The pressure wave will oscillate back and forth in the pipe until it is dampened by the system and reaches the final steady-state condition (Figure 7.2).

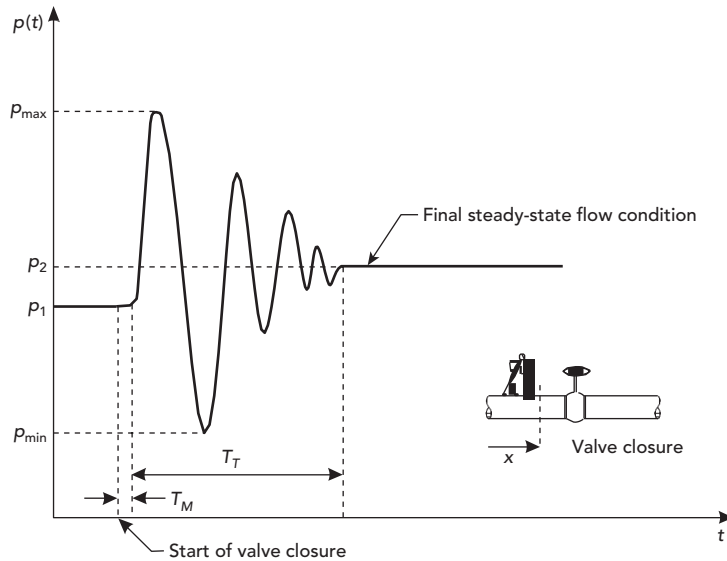


Figure 7.2: Oscillating pressure wave near a recently closed valve.

On the downstream side of the valve, the water will initially move away from the closed valve causing a region where the pressure is suddenly reduced. There is the potential for the pressure to drop to a level where dissolved gasses can come out of the water, and also below the vapor pressure of the fluid itself to create vaporous cavitation (column separation). If the flow (liquid columns) reverses, these pockets of gasses or vapor could suddenly collapse creating a strong transient with the potential to damage the pipe lining, or rupture the pipe.

Common Causes

The majority of hydraulic transients are the result of changes at system boundaries, typically at the upstream or downstream ends of the systems, or at high points (locations of high elevation) in the pipe network. Three common causes of transients in hydraulic systems are described below.

Pumps—A pump delivers energy from the pump impeller to the fluid as the fluid passes from the suction side to the discharge side of the pump. Pumps force water through the piping network and components on the discharge side of the pump. When the pump starts, it can greatly increase the pressure rapidly, or when a pump sags or fails, a sudden drop on pressure can propagate downstream.

Turbines—A turbine is located on the downstream end of a penstock (the feeding conduit). As water flows through the penstock and enters the turbine, the water energy rotates a turbine impeller, which creates an electrical current

from the generator. If electricity from the turbine is no longer needed, the flow through the penstock is stopped, which can create a pressure increase that propagates upstream into the penstock.

Valves—A valve can change, start, or stop flow suddenly. The rate of closing or opening of the valve can allow for a gradual increase or decrease of flow to reduce the magnitude of the transient pressure wave. However, if a valve suddenly closes (valve slam), there is a sudden high-pressure pulse that propagates upstream of the valve and a sudden low-pressure pulse that propagates downstream. If the low-pressure pulse is severe enough, gas or vapor pockets may form, or external water could be pulled into the pipe network. Also, there is the potential for the pipe to rupture or collapse.

Sudden changes in flow (fluid momentum) create a pressure pulse that propagates away from the disruption in every direction and throughout the entire pipe network. If no other transient event occurs, then the pressure wave will continue to propagate as unsteady flow until the transient is dampened by friction. After the transient is dampened, the system reaches the final steady-state condition.

7.2 Hydraulic Transient Theory

Hydraulic transient flow is classified as unsteady flow. When the flow rate changes rapidly in a hydraulic system the flow momentum changes as the fluid accelerates or decelerates. A hydraulic transient is analyzed mathematically by solving for the fluid velocity and pressure, and both are a function of position and time. The continuity equation and momentum equations are needed to determine the fluid velocity and pressure. Solving these two equations simultaneously has shown to reflect actual field measurements as long as the data and assumptions used are valid. Hydraulic transients are analyzed using one of two types of models.

Rigid-Column Model—The unsteady flow is modeled as an incompressible fluid within a rigid system. When the continuity equation is used with this model, the fluid flow changes instantaneously throughout the system and travels as a single mass inside a nonflexible pipe. This model is limited because it does not accurately describe the pressure wave propagation in a pipe and does not apply to very rapid changes in flow. This model is used for slow-flow transients known as surge or mass oscillation.

Elastic Model—The unsteady flow is modeled as a compressible fluid within a flexible system. Since the fluid is compressible, it can experience density changes and the propagation of the pressure wave in the fluid column can be described. The pressure wave velocity (celerity) is dependent upon the elasticity of the fluid and pipeline (pipe material and type of joint). This model is more commonly used to describe hydraulic transient flow for rapid operational system changes.

The derivation of the complete equations for transient analysis is beyond the scope of this book but can be found in Almeida and Koelle (1992) and Wylie and Streeter (1993). Also, for the remainder of this book, only the Elastic Model will be used to describe hydraulic transient flow.

General Modeling Equations

The continuity equation for a fluid is based on the principle of conservation of mass. The simplified general form of the equation for unsteady one-dimensional pipe flow that is used in transient flow models is as follows:

$$\frac{\partial H}{\partial t} + \frac{a^2}{gA} \frac{\partial Q}{\partial x} = 0$$

where

$\partial H / \partial t$ = change in hydraulic head (m, ft) with time (s),

$\partial Q / \partial x$ = change in flow (m³/s, cfs) with position (m, ft),

a = pressure wave celerity in the fluid (m/s, ft/s),

A = cross-sectional area of the fluid (m², ft²), and

g = gravity constant (m/s², ft/s²).

The momentum equation for a fluid can be derived from considering the forces acting upon a small fluid volume (control volume), including the shear stresses generated by the fluid motion and viscosity. The three-dimensional momentum equations for a fluid are also known as the Navier–Stokes equations. The simplified general form of the equation for unsteady one-dimensional flow that is used in transient flow models is as follows:

$$\frac{\partial Q}{\partial t} + gA \frac{\partial H}{\partial x} + \frac{fQ|Q|}{2DA} = 0$$

where

$\partial Q / \partial t$ = change in flow (m³/s, cfs) with time (s),

$\partial H / \partial x$ = change in hydraulic head (m, ft) with position (m, ft),

Q = fluid flow rate (m³/s, cfs),

D = inside pipe diameter (m, ft), and

f = Darcy–Weisbach friction coefficient.

The last term in the momentum equation represents the friction losses in the direction of fluid flow.

Hydraulic transient modeling consists of solving these equations for a wide variety of boundary conditions and system topography. The equations cannot be analytically solved so various approximate methods are used. Except for the simplest forms of these equations, the solutions are found by using computers. Bentley's HAMMER software uses the method of characteristics to convert the two partial differential equations into four ordinary differential equations. It does this by determining the characteristic curves when the partial differential equations become ordinary differential equations. The ordinary differential equations are solved along the characteristic curve and then transformed into a solution for the original partial differential equations.

Speed and Pressure of Transients

The pressure wave that is formed during a hydraulic transient will propagate through the pipe network at a speed up to the speed of sound in the fluid. The speed of sound in fresh water is approximately 1,447 m/s (4,748 fps). The pressure wave celerity (a) is less than the speed of sound since the pipe material and joints are flexible, which dampens the pressure wave. A more flexible pipe material or expanding joints will have a much slower hydraulic transient pressure wave speed.

In 1878, Diederik Korteweg developed an equation that relates the pressure wave celerity to the properties of the pipe material and fluid. The pressure wave celerity used in the Elastic Model is determined from this equation.

$$a = \sqrt{\frac{E_f / \rho_f}{1 + (DE_f / eE)\Psi}}$$

where

a = pressure wave celerity (m/s, fps),

E_f = modulus of elasticity for the fluid (Pa, lbf/ft²),

ρ_f = fluid density (kg/m³, slugs/ft³),

D = inside pipe diameter (mm, in.),

e = pipe wall thickness (mm, in.),

E = modulus of elasticity for the pipe material (Pa, lbf/ft²), and

Ψ = pipeline support factor.

The pipeline support factor (Ψ) is determined from the Poisson's ratio of the pipe material. For thin-walled pipes, if the pipeline has functioning expanding joints, $\Psi = 1$; if the pipeline anchored against axial (longitudinally) movement, $\Psi = 1 - \mu^2$; if the pipeline is anchored at one end only, $\Psi = 5/4 - \mu$ (see *Advanced Water Distribution Modeling and Management*. Heastad Press, 2003). Properties of pipe materials and common fluids can be found in Tables 7.1 and 7.2.

Hydraulic Transients in Distribution Systems

Hydraulic Transient Theory

In a single pipeline, the pressure wave will reach the other end of the pipe in a time interval equal to L/a seconds where L is the pipe length (m, ft). For the pressure wave to return to the original location that caused the transient, the time interval will be $2L/a$ seconds. The quantity $2L/a$ is known as the characteristic time for the pipeline. Flow control operations that occur faster than the characteristic time are considered “rapid” and have a high risk to create very dangerous transients, flow control operations that are slower than the characteristic time are considered “gradual,” and operations with a time much greater than $2L/a$ are considered “slow.” The characteristic time can be used to determine which hydraulic transient flow model to use. For “slow” operations, it is possible to use the Rigid-Column Model, and for all other conditions, the Elastic Model is used.

Table 7.1 Physical Properties of Some Common Pipe Materials

Material	Modulus of Elasticity		Poisson's Ratio, μ
	10^9 lbf/ft^2	GPa	
Steel	4.32	207	0.30
Cast iron	1.67–3.55	80–170	0.25–0.27
Ductile iron	3.59	172	0.28
Concrete	0.42–0.63	20–30	0.10–0.15
Reinforced concrete	0.63–1.25	30–60	0.25
Asbestos cement (AC)	0.50	24	0.30
Polyvinylchloride (PVC)	0.069	3.3	0.45
Polyethylene (PE)	0.017	0.8	0.46
Polystyrene (PS)	0.10	5.0	0.40

Table 7.2 Physical Properties of Some Common Fluids

Material	Modulus of Elasticity		Density	
	10^6 lbf/ft^2	GPa	Slugs/ft ³	kg/m ³
Fresh water	43.7	2.15	1.94	998
Salt water	47.4	2.27	1.99	1,025
Mineral oils	31.0–40.0	1.5–1.9	1.67–1.73	860–890
SAE 30 oil	31.0	1.5	1.77	912

In 1897, Joukowsky demonstrated that the momentum equation can be used to correctly predict the maximum pressure created by a hydraulic transient in the Moscow, Russia water system (Rouse and Ince, 1957). This equation can be used to estimate the pressure for a hydraulic transient in a single pipeline connected to a reservoir.

$$\Delta P = a \rho_f \Delta v$$

where

ΔP = pressure change due to hydraulic transient (Pa, psf) and

Δv = change in fluid velocity (m/s, fps).

For the case where the fluid velocity goes to zero rapidly due to the fluid stopping suddenly, the maximum pressure near the control that stopped the flow can be estimated:

where

ΔP_{max} = maximum pressure change (Pa, psf) and

v_i = initial fluid velocity (m/s, fps).

Example 7.1: Estimating Maximum Hydraulic Transient Pressure

Estimate the maximum internal pressure for a 24-in. ductile iron pipe (I.D. = 24.86 in.; e = 0.47 in.) with expanding joints after a valve suddenly closes 2,500 ft downstream from a reservoir. The regular operating pressure is 95 psi and velocity is 6.5 fps at the valve location.

Solution

Determine the pressure wave celerity using the fluid and pipe material properties. Since the joints are allowed to expand, the pipeline support factor (Ψ) is equal to 1.

$$a = \sqrt{\frac{E_f / \rho_f}{1 + (DE_f / eE) \psi}} = \sqrt{\frac{43,700,000 \text{ psf} / 1.94 \text{ slugs/ft}^3}{1 + \frac{24.86 \text{ in.} \times 43,700,000 \text{ psf}}{0.47 \text{ in.} \times 3,590,000,000 \text{ psf}} \times 1}} = 3,701.8 \text{ fps}$$

For the fluid velocity going to zero, $v_i = 6.5$ fps the maximum pressure due to the hydraulic transient can be determined.

$$\Delta P_{max} = a \rho_f v_i = \frac{3,701.8 \text{ fps} \times 1.94 \text{ slugs / ft}^3 \times 6.5 \text{ fps}}{144 \text{ in.}^2 / \text{ft}^2} = 324 \text{ psi}$$

The total pressure in the pipeline would be the operating pressure plus the pressure increase caused by the hydraulic transient.

$$P_{total} = P_{op} + \Delta P_{max} = 95 \text{ psi} + 324 \text{ psi} = 419 \text{ psi}$$

The characteristic time for this pipeline at the valve can be determined.

$$\text{Characteristic time} = 2L/a = 2 \times 2,500 \text{ ft} / 3,830.7 \text{ fps} = 1.35 \text{ s}$$

Since this valve closed suddenly, this would be considered a “rapid” flow operational control.

7.3 Minimizing Transients in Systems

Network systems should be designed and operations set to minimize the likelihood of hydraulic transient events. However, transients will still occur and methods must be evaluated to minimize damage caused by transients. There are two general strategies used to control transients. First is to minimize the possibility of transient conditions during project design and system flow control operations. Emergency backup operational systems are placed to avoid unusual system conditions. The second is to strategically place transient control devices.

Design Strategies

When designing pipe network systems, the engineer will need to consider the technical requirements and demands on the system. Sufficient safety factors, such as the pipe's wall thickness, sufficient storage, and emergency operations need to be evaluated to minimize transient conditions. A long pipe with a high flow velocity and minimal branching is most vulnerable to hydraulic transients. Emergency flow control scenarios should be analyzed and tested during the design phase to ensure that the proper pipe diameter and wall thickness, and system components are specified.

The topographic conditions of the pipe network need to be considered. It is common to have local high points along the pipe network. It may be more desirable to reroute the pipeline to avoid high points where low-pressure conditions exist that are prone to the low-pressure pulses caused by transients. The presence of high points could lead to pipe collapse or column separation within the pipe if the pressure pulse is extreme.

Selecting larger diameter pipe will lower the fluid velocity within the pipe. This could reduce the risk of damage caused by transients; however, the economic cost would increase, and water age within the pipe would be higher which could reduce the water quality and disinfectant residual. For long pipeline systems, the diameter should be selected to optimize the construction and operational cost. In most cases, long pipelines will require transient protection devices.

Steel, plastic, and thin-walled ductile iron pipes are most prone to damage caused by hydraulic transients. The pipe will be weakened by repeated exposure to transients either due to the pressure wave itself or by vibrations caused by the transient. A pipe weakened by age or corrosion would also be vulnerable. The engineer may specify a more expensive, thick-walled pipe in areas that are most vulnerable to transients. Also, the type of joint specified could be influenced by the possibility of transients.

With the use of a transient model, the engineer can test different valve and pump operations to minimize the effect of hydraulic transients. The most severe transients with pumps typically occur with a power outage when the pump

suddenly stops. For normal pump operations, a slow opening/closing valve will gradually change the fluid velocity, which reduces the pressure pulse created by transients. Different ramping speeds with soft starts, or using variable speed drive motors on pumps, can be tested to minimize transients, although this is typically accomplished much more economically with slow opening/closing valves near the pump. The control valve after the pump should be opened slowly after the pump started and closed slowly before the pump shuts down.

Protection Devices

When possible, design strategies should be used to minimize transients. However, if transients cannot be avoided, then specific protection devices to control transients may be needed.

Surge tanks—Surge tanks work by allowing water to bleed out of the system during high pressures and add water to the system during low pressures. A surge tank is a small tank that has a water height equal to the hydraulic grade line (HGL) and feeds the protected pipe by gravity (see Figure 7.3). The surge tank is sized to provide adequate hydraulic capacity during low-pressure conditions, and the connecting valve is sized to prevent the tank from filling or draining too slowly or quickly. The transient is dissipated as water enters or leaves the tank. These devices should be equipped to allow for water to circulate into the tank to keep water fresh and if necessary, these devices should be protected from freezing in colder climates.

A one-way tank is a type of surge tank. It will store water under atmospheric conditions, and through a check valve allow water to enter the protected pipe during low-pressure transients. This will reduce the low-pressure transient.

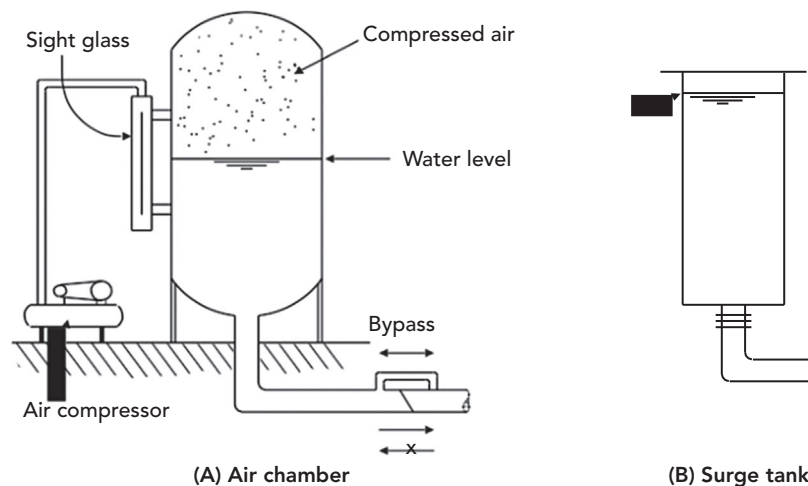


Figure 7.3: Examples of transient protection devices.

Hydraulic Transients in Distribution Systems

Minimizing Transients in Systems

After the transient event, a float valve opens and allows for the gradual refilling of the one-way tank. The advantage of using the one-way tank is the water level in the one-way tank can be much lower than the HGL. It is common to place one-way tanks on the discharge pipe after a pump in locations that are vulnerable to low-pressure transients after the pump shuts off or experiences a sudden failure (e.g., power failure).

Hydropneumatic tanks—A hydropneumatic tank is also known as an air chamber or gas vessel. A hydropneumatic tank is a closed tank that contains compressed air, or nitrogen, above a volume of water (see Figure 7.3). The gas or air pressure in the chamber can be preset to gauge zero or higher. During conditions of low pressure in the protected pipe, the compressed air will expand and force water from the tank into the pipe, which could avoid or reduce column separation and surges. During conditions of high pressure in the protected pipe, the air or gas in the tank acts as a “cushion” against the transient. These devices should be protected from freezing in colder climates.

There are three types of hydropneumatic tanks in HAMMER. A sealed tank is simply a closed pressure vessel allowing water to move between the tank and protected pipe based on the pressure within the pipe. A hydropneumatic tank can also be vented with an air valve to allow air to enter the tank during conditions of low pressure and effectively become a one-way surge tank. They can also be vented with an air valve and dipping (ventilation) tube inside. If the HGL drops and lowers the tank water level below the dipping tube, the air valve opens and the tank effectively becomes a surge tank open to the atmosphere. When the HGL later increases, the air is expelled from the tank through the air valve and the water level will go above the bottom of the dipping tube, and the tank will act as a sealed pressure tank again.

Surge relief valves—In the case of a high-pressure transient, a surge relief (pressure relief) valve can be placed to allow the transient pressure to be relieved. These valves are set to open and allow water to discharge when the pressure in the protected pipe exceeds a predetermined value. This controls the maximum pressure within the protected pipe. After the high-pressure transient is relieved, the valve will close slowly to avoid causing another transient. Similar to a surge relief valve is a surge-anticipator relief valve. This valve is controlled by the system emergency operations and will start to open immediately in anticipation of a high transient, usually because of a power failure.

Air inlet valves—Air inlet valves are also called vacuum breakers. These valves are installed in high points of the pipe network to control vacuum conditions. During low-pressure transients, these valves will allow air to enter the system. This will control the minimum pressure within the protected pipe to avoid column separation or pipe collapse. An adequate period of time should be allowed for the air that entered the system to be expelled before normal operations are allowed to restart.

Rupture disk—A rupture disk are also called bursting disk or burst diaphragm. A rupture disk is a one-time use diaphragm or membrane that will rupture

when a predetermined high or low pressure ruptures the disk. This will create a “designed” opening in the protected pipe that can be later be easily repaired by replacing the ruptured disk with a new one. Many times a rupture disk is used as a backup protection device to a surge relief or air inlet valve.

7.4 Modeling Hydraulic Transients

The primary objectives of transient modeling are to determine the extreme pressure values that can result from sudden changes to the flow in pipe networks and analyze transient control alternatives. This can be due to operational changes within the system or due to unforeseen events such as power failure or pipe breaks. Hydraulic transients are pressure waves caused during the transition from an initial steady-state condition to a final steady-state condition. Because of the complexity of the equations needed to model transients, numerical computer models are used to analyze transient flow hydraulics. A well-constructed model is needed for the engineer to have confidence with identifying and analyzing alternative scenarios for controlling hydraulic transients. These models are used to establish design specifications for pipes and system components, as well test alternative normal and emergency operational controls to minimize the impact caused by hydraulic transients.

When the steady-state flow conditions are altered, the flow characteristics (velocity and pressure) at positions along the pipe network are altered with time until the final steady-state conditions are established. The time interval can be altered to provide enough modeling points to effectively observe the effect of the hydraulic transient. The time frame for a hydraulic transient can be a few seconds to a couple minutes in length depending on the size of the pipe network. The results of a transient model are the time-dependent change in pressure at different locations within the pipe network. The modeled maximum and minimum pressures are evaluated by the engineer, and operational controls or protection devices are tested to determine alternatives to minimize the hydraulic transient.

Normally, the entire pipe network does not need to be evaluated for hydraulic transients. Transient analysis should be performed on large, high-velocity pipelines, especially near pumping stations. Normal flow conditions, along with emergency operations and uncommon flow conditions should be evaluated. Since most hydraulic models used to analyze water distribution systems such as WaterGEMS do not consider momentum in their analysis, a hydraulic transient analysis model is required after the piping network is skeletonized.

Skeletonization is the process of selecting only parts of the piping network that have a significant impact of the modeled behavior. Capturing every feature of a piping network in a transient model would require a tremendous amount of data and computational power. Also, the impact of including small pipes to the modeled results would be minimal since it is relatively easy to identify the most vulnerable locations to hydraulic transients. Much of a pipe network would have small pipes or pipes with a relatively low velocity which would not likely be damaged by a

hydraulic transient. Skeletonization is a more practical approach to modeling transients that allows the modeler to focus on the piping that is more likely to be damaged by a transient without investing unnecessary time and resources. For a more detailed explanation and examples of skeletonization, see Heastad Press (2003).

The validation and calibration of hydraulic transient models depend on adequate field instrumentation and dynamic measurements. Instead of conventional pressure gauges and SCADA systems, high-speed datalogging equipment is needed to accurately track a hydraulic transient pressure wave. A pressure sensor will need to be very sensitive over a large pressure range, have high resolution, and be connected to a high speed data acquisition unit. With the measured data, the pressure wave celerity and extreme pressure values can be recorded. It is common to have at least two locations established in the system for measurements. Also, the operational conditions such as valve open/closed positions, pump impeller rotational speed, and water levels in transient protection devices must be measured.

7.5 HAMMER and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software.

What Does HAMMER Do?

HAMMER is a powerful, easy-to-use program that helps engineers design and analyze hydraulic transients in pumping systems and water distribution networks as they transition from one steady state to another. Since the hydraulic transient can last for a few seconds to a few minutes, the time frame for analysis is very short. But because of the extreme pressures, both positive and negative, a hydraulic transient can cause significant damage to a pumping system or pipe network.

HAMMER is also used by water utility managers to as a tool to analyze operation strategies or protection devices to minimize hydraulic transients. HAMMER's sophisticated modeling capabilities can be used to

- analyze operation strategies and protection devices to reduce the risk of transient-related damage to piping networks;
- set operations to reduce wear and tear on pumps and pumping systems;
- prepare operations checklists for use in emergency failures such as power outages, pipe breakage, and component (valve, pump), and/or control failures;

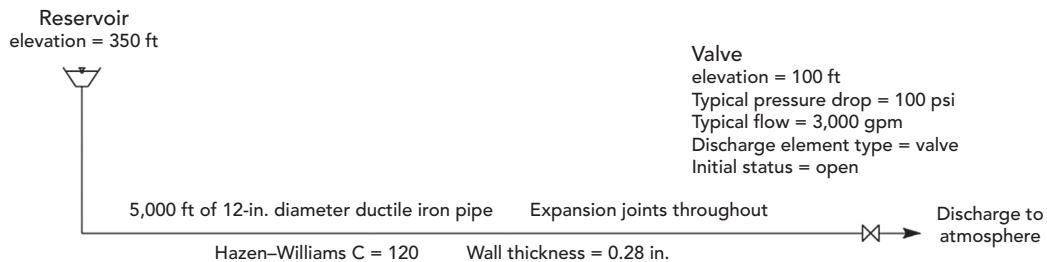
- reduce the risk of external water entering a piping network that can contaminate distribution water; and
- identify locations or pipes that are of high risk to damage caused by a hydraulic transient.

HAMMER can be used as a stand-alone program, or interface with MicroStation, AutoCAD or ArcGIS, HAMMER shares the same file format of WaterCAD and WaterGEMS so models can be easily shared between these applications. It will also import files from EPANET.

7.6 Problems

Solve the following problems using the HAMMER computer program.

1. A reservoir discharges to atmosphere through 5,000 ft of 12-in. pipeline, as shown in the following diagram.



Schematic for Problem 1

Use HAMMER's "Discharge to Atmosphere" element to represent the valve in your model. Assume the fluid is water at 39°F. Also set the Transient Friction Method to Unsteady, Run Duration Type to Time, Run Duration to 600 s, and use a Vapor Pressure of -14.2 psi.

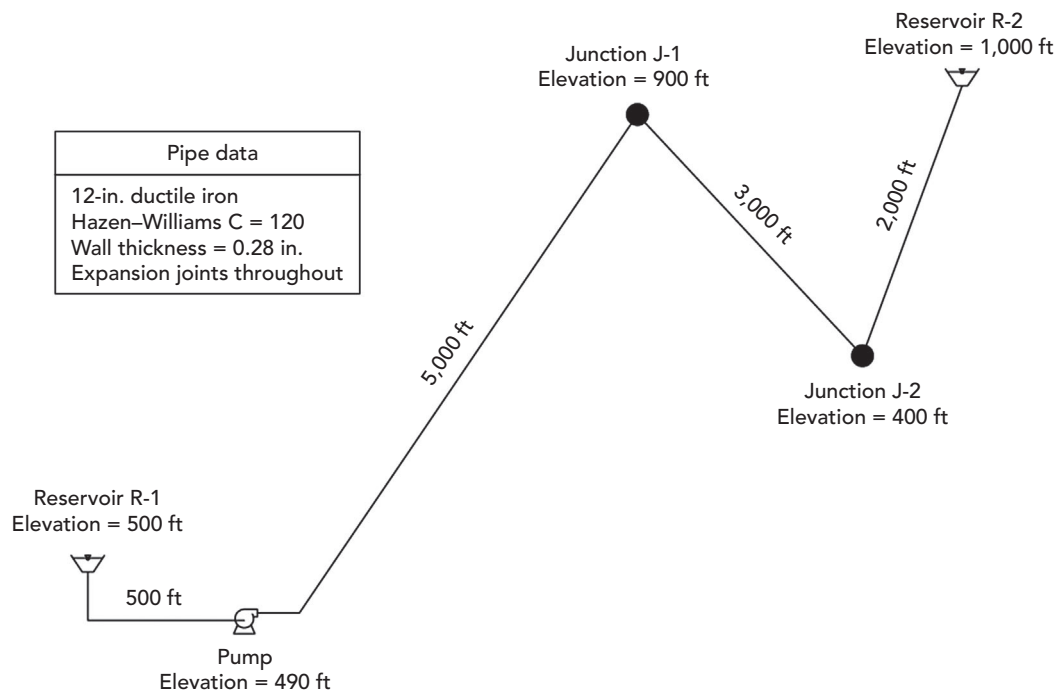
- (a) What are the static, nontransient pressure and hydraulic grade line (HGL) at the valve when the valve is closed?
- (b) What are the maximum and minimum pressures and HGLs at the valve for closing times of 2, 10, 30, and 60 s?

Valve Closing Time (s)	P_{Max} (psi)	P_{Min} (psi)	HGL Max (ft)	HGL Min (ft)
2				
10				
30				
60				

Hydraulic Transients in Distribution Systems

Problems

2. A pump draws water from reservoir R-1 and delivers it to reservoir R-2 through 10,500 ft of 12-in. pipeline. The pipeline passes over a high point at Junction J-1, as shown in the following diagram.



Schematic for Problem 2

Pump definition data follows:

Flow (gpm)	TDH (ft)	
0	840	
2,500	630	(BEP)
5,000	0	

Additional pump information: Pump efficiency at BEP = 65%

RPM = 1,800 rpm

Specific speed = 980 (US)

Reverse spin allowed

Pump control valve = 8-in. diameter

Valve closure on power failure = 10 s

Motor = constant speed

Motor efficiency = 90%

Assume the fluid is water at 68°F. Set the Transient Friction Method to Unsteady, Run Duration Type to Time, Run Duration to 240 s, and use a Vapor Pressure of -14.2 psi.

- (a) What are the steady-state pressure and hydraulic grade line (HGL) at Junction J-1 with the pump running?
- (b) What are the maximum pressure and maximum HGL at Junction J-1 following a power failure?
- (c) If Junction J-1 is replaced with a combination air/vacuum valve (4-in. air inlet diameter, 1/4-in. air outlet diameter), what are the maximum pressure and maximum HGL at the air/vacuum valve following a power failure?

8

Sanitary Sewer Design

8.1 Sanitary Sewer Systems

Sanitary sewers collect wastewater from its sources and transport it to a treatment facility. Sanitary sewers are intended to convey only wastewater. However, their design must allow for the infiltration of groundwater that invariably occurs in sewers. Older systems that convey both sanitary wastes and stormwater are referred to as **combined sewers**.

Wherever practicable, sewers are gravity flow systems. When the natural slopes are not sufficient to convey flow, a combination of gravity and pressure flow systems is used. The gravity sewer transports flow to a collection point such as a wet well. The wastewater is pumped from the wet well through a force main over some obstruction or hill to another gravity sewer, or directly to a wastewater treatment facility.

The hydraulics in the gravity portion of a sanitary sewer are analyzed with the same techniques used in storm sewers hydraulics, as discussed in Section 1.6 and in Chapter 3. When the flow depth is constant, Manning's equation is used. When obstructions or changes in pipe slope exist, the gradually varied flow analysis procedure presented in Section 3.6 is applied.

The pressure portion of the sanitary sewer is analyzed in the same manner as a water distribution system, as discussed in Chapter 6. The primary difference is that flow is generally withdrawn from a water distribution network, whereas in a sewer system, flow is injected into force mains.

In this chapter, a brief introduction to sanitary sewer design is presented. The use of SewerCAD to assist in the design process is demonstrated. More comprehensive reviews of sewer design may be found elsewhere (Tchobanoglous, 1981, or ASCE and WEF, 2007). Additional discussion on the development and application of hydraulic sewer models is presented in *Wastewater Collection System Modeling and Design* (Haestad Press, 2004).

Common Sanitary Sewer Elements

Although all of the various types of sewer appurtenances and structures are too numerous to mention in this text, the most commonly modeled sanitary sewer elements include the following:

- **Manholes:** Manholes provide access for maintenance of a sanitary sewer. They are generally located at regular intervals (as dictated by local ordinances), at places where pipe characteristics such as diameter and slope change, and at places where several pipelines join. In sanitary sewer models, such as SewerCAD, manholes also represent points where loads enter the system.
- **Junction chambers:** Junction chambers are underground structures built at points in the sewer where multiple pipes join. Junction chambers are rarely used in sanitary sewers because their underground construction makes access and maintenance difficult.
- **Wet wells:** Wet wells are structures that collect and store wastewater until it is pumped into force mains for transport to another gravity system or to a wastewater treatment facility. Wet wells are generally designed to minimize wastewater retention time in order to prevent odors and the buildup of gases, while allowing the pumps to cycle in an efficient manner.
- **Pumps and pump stations:** Pumps, as discussed in Chapter 6, add head to flow as it passes through the sanitary sewer system.
- **Pressure junctions:** Pressure junctions represent points in the pressure system where pressure pipes are joined together. They also represent points where loads directly enter a force main, or points where the modeler wishes to know a pressure.
- **Pipes:** Pipes are conduits through which flow is transferred either by the influence of gravity or by energy supplied from pumping stations. The most common pipe section shapes in a gravity system are circular, box (rectangular), arched, elliptical, and egg-shaped. In pressure systems, pipes are usually circular. Pressurized sewer pipes are commonly referred to as *force mains*.
- **Regulators:** When stormwater loads cause flows in sewers to exceed capacity, control devices are used to divert flows. These devices are called *regulators*. They split the flow between an interceptor sewer and an overflow. Possible destinations for controlled releases include wet wells, temporary storage basins, treatment facilities, parallel relief sewers, or receiving waters.

8.2 Loading

Sanitary sewer loads are divided into two categories: sanitary loads and wet-weather loads.

- **Sanitary loads** (*dry-weather loads*) result from human activity and are not weather dependent. Common sources of sanitary loads are various residential, commercial, recreational, and industrial uses.
- **Wet-weather loads** are related to rainfall activity such as groundwater infiltration (water leaking into a pipe through cracks, joints, and defects) and structure inflow (surface water entering a structure through openings around the cover, or due to a missing cover).

Time-based loads can be classified as either wet-weather or sanitary loads, and they may have hydrograph- or pattern-based formats.

Common Load Types

Two loading types that can be applied as either wet-weather or sanitary loads are

- hydrographs and
- pattern loads.

Hydrographs

Flow versus time data, also known as a hydrograph, can be entered as a load. The hydrograph will then be directly added to any other loads coming to that point, and then routed downstream.

In the case of wet-weather loading, the hydrograph may reflect infiltration or inflow resulting from an actual or design storm event. Sanitary load hydrographs show the variation in customer loading over time, such as over the course of a typical day. An example of a hydrograph is shown in Figure 8.1. Hydrographs may be obtained from measurements collected in a combined sewer during a rainfall event or they may be developed using the hydrologic procedures described in Chapter 2.

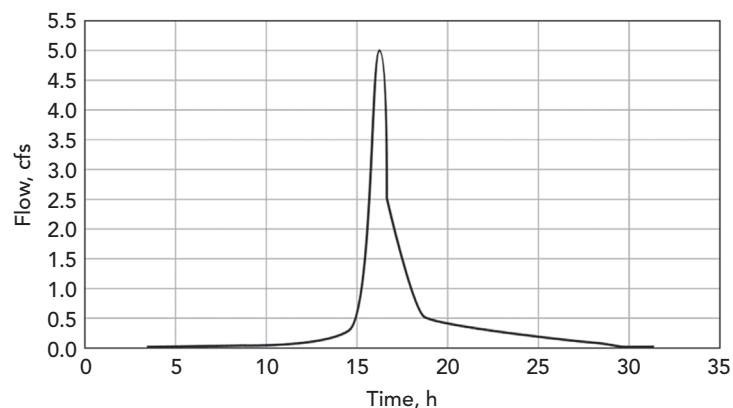


Figure 8.1: A hydrograph.

During a steady-state analysis, a hydrograph load must be converted into an instantaneous load. There are several ways of choosing an appropriate instantaneous load, and the method chosen depends on the purpose of the analysis. For instance, the hydrograph peak flow, average flow, or minimum flow may be used. If other data are used to generate steady-state loading, the hydrograph may be disregarded entirely.

Pattern Loads

A pattern load comprises a base load and an associated loading *pattern*. The *pattern* is a series of multipliers relating sanitary sewer loading to time, and it allows the user to automatically apply time-variable changes within the system.

Different categories of users, such as residential or industrial customers, may be assigned different patterns to accurately reflect their particular load variations. A *diurnal curve* is a type of pattern that describes change in loading over the course of a daily cycle. A dry-weather diurnal curve for sanitary loading often resembles the diurnal curve for water use at the same location (see Section 6.1), reflecting times when people are using more and less water than average.

Patterns are typically based on a multiplication factor versus time relationship, in which a multiplication factor of 1.0 represents the base value (usually the average value). This relationship can be expressed:

$$Q_t = A_t \times Q_{Base}$$

where

Q_t = load at time t ,

A_t = multiplier for time t , and

Q_{Base} = baseline load.

For the residential usage, diurnal curve in Figure 8.2, we see that there is a peak in the diurnal curve in the morning, another slight peak around noon, and a third peak in the evening. During the nighttime hours, the pattern reflects the relative inactivity of the system. (Note that this curve is conceptual and should not be construed as representative of any particular network.)

As explained in Chapter 6, there are two basic forms for representing a pattern: stepwise and continuous. A *stepwise pattern* is one that assumes a constant level of usage over a period of time and then jumps instantaneously to another level where it again remains steady until the next jump. A *continuous pattern* is one for which several points in the pattern are known and sections in between are transitional, resulting in a smoother pattern. For the continuous pattern in Figure 8.2, the magnitude and slope of the pattern at the start and end times are the same—a continuity that is recommended for patterns that repeat.

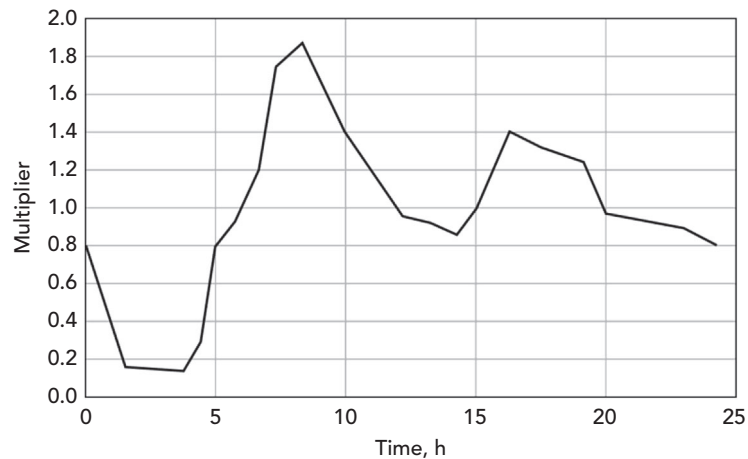


Figure 8.2: An example of a diurnal pattern for sanitary loading.

Sanitary Unit Loading

Ideally, sanitary loads are based on data measured at individual loading points throughout the sewer system. In other words, the load from each source is modeled as a direct flow into the sewer system. However, data with this level of detail are not typically available. Instead, wastewater sources are grouped together, and average loads are computed based on land usage or the activities of the population generating the load.

For example, consider a small, middle-class subdivision that houses approximately 50 people. Studies have shown that, on average, a resident of middle class housing generates 280 l/day (74 gpd) of wastewater. Thus the average load entering the sewer system at this subdivision is 14,000 l/day (3,700 gpd).

Sanitary loads are typically estimated based on a number of contributing units, with a specified average load per unit (such as 260 l/day per apartment resident). The unit is typically a measure of population such as *residents* or *employees*; however, loads can also be based on other criteria such as *contributing area* or user-defined counts of items indicative of loading behavior. See Table 8.1 for a list of standard loading sources and their associated average base loads.

Extreme Flow Factors

Sewer design and analysis generally consider a variety of loading conditions, such as minimum, average, and peak conditions. Base (average) sanitary loads are transformed into minimum or peak loads using an **extreme flow factor (EFF)** method (Figure 8.3).

Table 8.1 Typical Unit Sanitary Loads from Different Sources
(Tchobanoglous, 1981)

Unit Sanitary Load	Loading Unit	Base Load (l/day per Loading Unit)
Airport	Passenger	10
Apartment	Resident	260
Automobile service	Employee	50
Bar	Customer	8
Cabin resort	Guest	160
Campground	Guest	120
Coffee shop	Guest	20
Department store	Toilet room	2,000
Dormitory	Guest	150
Home (average)	Resident	280
Home (better)	Resident	310
Home (luxury)	Resident	380
Hospital (medical)	Bed	650
Hotel	Employee	40
Prison	Inmate	450
Restaurant	Meal	10
School (large)	Student	80
School (medium)	Student	60
School (small)	Student	40
Swimming pool	Employee	40
Shopping center	Parking space	4

The most common type of EFF is the variable peaking factor (PF). In this method, a computed EFF is multiplied by the specified base flow.

$$Q_{peak} = Q_{base} \times EFF$$

where

Q_{peak} = transformed flow (l/day, gpd),

Q_{base} = base flow (l/day, gpd), and

EFF = extreme flow factor (unitless).

Common Variable Peaking Factors (PF)

Some of the most common variable PF calculation methods are as follows:

- **Babbitt** (1958)

$$PF = \frac{5.0}{(P/1,000)^{0.2}}$$

where P = contributing population.

- **Harmon** (1918)

$$PF = 1.0 + \frac{14.0}{4.0 + (P/1,000)^{0.5}}$$

- **10 States Standard** (Great Lakes Upper Mississippi River Board, 1997)

$$PF = \frac{18 + \sqrt{\frac{P}{1000}}}{4 + \sqrt{\frac{P}{1000}}}$$

- **Federov** (Jakovlev et al., 1975)

$$PF = \frac{2.69}{Q^{0.121}}$$

where Q = base sanitary flow (l/s).

For population-based PF methods, the PF decreases as the population increases. A larger population means that peak loads from different sources are likely to occur in a more staggered manner. Thus the peak loads are likely to be less pronounced compared to the average loading rate. In systems servicing smaller populations, peak loads from different sources are more likely to coincide, causing more pronounced differences in average loading rates. The graph in Figure 8.3 shows how the EFF decreases with increasing population when applying the 10 States Standard.

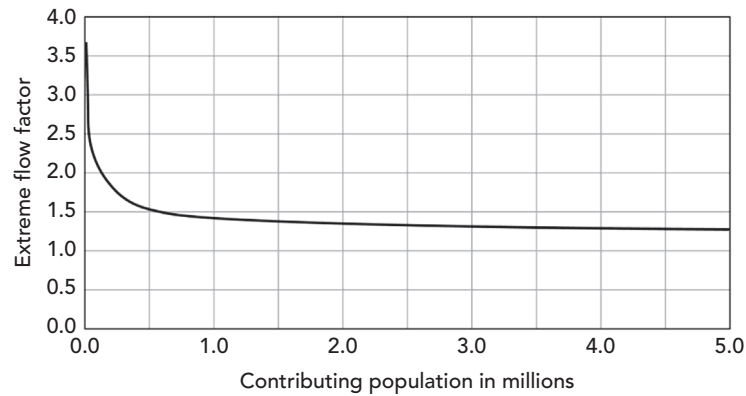


Figure 8.3: EFF versus contributing population, from the 10 States Standard.

Example 8.1: Determining Peak Sanitary Loads

A shopping center and a residential subdivision of average and luxury homes load a gravity sanitary sewer. The shopping center has 100 parking spaces. Eighty people live in the “average” housing, whereas 30 people live in the “luxury” housing. What is the average base load contributing to the sanitary sewer at this point? What would be an approximate peak flow applied to the system? The local ordinance dictates that the Harmon’s method must be applied to peak all population-based unit sanitary loads, whereas a constant PF of 2 should be applied to all nonpopulation-based loads.

Solution

First, to get the average base load, find the amount of load per unit for the three loading categories mentioned in Table 8.1. Then, multiply this unit flow by the number of units.

Shopping center	100	Parking spaces	×	41/(day-space)	=	400l/day
Average housing	80	Residents	×	280l/(day-resident)	=	22,400l/day
Luxury housing	30	Residents	×	380l/(day-resident)	=	11,400l/day
Total base flow						= 34,200l/day

Apply Harmon’s method to find the PF for the residential population of 80 residents of “average” housing and 30 residents of “luxury” housing.

- Assuming that both population-based loads have similar peaking behavior, sum the two populations:

$$80 \text{ Residents of average housing} + 30 \text{ residents of luxury housing} = 110 \text{ Residents}$$

- Apply the Harmon's method using a contributing population of 110:

$$PF = 1.0 + \frac{14}{4 + (110/1,000)^{0.5}} = 4.23$$

Apply this PF to the base loads calculated from the subdivision. Then, apply the given PF of 2 to the load derived from the shopping center to compute the approximate peak flow.

$$4.23 \times (22,400 \text{ l/day} + 11,400 \text{ l/day}) + 2 \times 400 \text{ l/day} = 144,000 \text{ l/day}$$

Wet-Weather Loading

Wet-weather loading represents the intrusion of ground and surface water into the sewer system and is divided into two components: infiltration and inflow. **Infiltration** represents groundwater entering pipes through cracks, holes, and other defects. **Inflow** represents wet-weather flow that enters the sewer through manhole covers, roof drains, catch basins, sump pumps, and so forth. Collectively, these flows are referred to as **III**. It is important to minimize intrusion of rain water into the sewer system because of the cost of treating the added flow. Also, wet-weather loads increase the potential for overloading the system, which causes flow to back up into houses or overflow into the environment.

Infiltration causes noticeable increases in sewer flow, especially after rainstorms and subsequent rises in the groundwater table. Sewer pipes placed in low areas, such as near creek beds, are more subject to infiltration than those placed in higher ground because they are more likely to be located below the water table. Older sanitary sewers are more susceptible to infiltration than newer sewer systems because modern techniques and materials used in newer systems can generally prevent or greatly reduce infiltration.

Infiltration at an individual pipe is generally hard to quantify. Instead, infiltration is estimated for the whole system or a portion of the system. This global infiltration is then distributed to individual pipes using one of the following methods:

- proportional to pipe length
- proportional to pipe diameter/length
- proportional to pipe surface area
- proportional to some user-defined count (e.g., number of defects in the pipe)

Inflows are unwanted flows entering from the surface and are modeled based on estimates or requirements from local or state regulatory agencies. Like infiltration, inflows can be quite significant and can vary considerably depending on the depth of runoff, the size of the manhole cover, and the number of openings in the manhole cover.

8.3 Extended-Period Simulations

Overview

An *extended-period simulation (EPS)* models how a sewer network will behave over time. This type of analysis allows the user to model how wet wells fill and drain; pumps toggle on and off; and pressures, hydraulic grades, and flow rates change in response to variable loading conditions and automatic control strategies formulated by the modeler. EPS is a useful tool for assessing the hydraulic performance of alternative pump and wet well sizes.

The SewerCAD algorithm proceeds in a general downstream direction toward the outfall, the whole procedure of this is described here:

1. The analysis begins in the gravity portion of the network. The hydrographs enter the gravity system and are successively routed and summed as the flows approach the downstream wet well or outfall. Ultimately, the total inflow hydrograph to the wet well is determined.
2. Knowing the inflow into the wet well, the pressure calculations for the force main system bounded by the wet well are performed. In addition to flow velocities and pressures, the levels in the wet well over time are determined.
3. The calculation then returns to the gravity portion of the network discussed in step 1. The hydraulics and hydraulic grade line (HGL) profiles are calculated through out the gravity system for each time step using the known level of the wet well as the boundary condition for the backwater analysis.

The process repeats, continuing through the system downstream of the pressure network until an outlet is reached.

Routing Overview

As a hydrograph is routed through a conduit, it undergoes changes in shape and temporal distribution caused by translation and storage effects.

SewerCAD uses one of two methods to determine the shape and distribution of a hydrograph routed through a gravity pipe.

- convex routing
- weighted translation routing

Convex Routing

The underlying assumptions of the *convex routing method* are that the routed outflow for a time step is based on the inflow and outflow for the previous time step, and that the flow does not back up in the pipe (i.e., no reverse flow or reduced flow due to tailwater effects exists). Each outflow ordinate is calculated as:

$$O_{t+\Delta t} = cI_t + (1 - c)O_t$$

where

$O_{t+\Delta t}$ = outflow at time $t + \Delta t$;

t = current time (s, min),

Δt = hydrologic time step (s, min),

c = convex routing coefficient,

I_t = inflow at time t (l/s, gpm), and

O_t = outflow at time t (l/s, gpm).

The convex routing coefficient is essentially a ratio of the hydrologic time step and representative flow travel time through the pipe and is calculated as follows:

$$c = \Delta t \frac{V}{L} = \frac{\Delta t}{t_r}$$

where

Δt = hydrologic time step (s),

t_r = travel time (s),

V = velocity established for representative flow (m/s, ft/s), and

L = length of pipe (m, ft).

In SewerCAD, the velocity used to calculate the coefficient is either the normal velocity or full velocity generated for a user-specified percentage of the peak of the inflow hydrograph. In other words, if the percentage of the peak flow is greater than the capacity of the pipe, the full flow velocity is used. If the percentage of the peak flow is less than the pipe capacity, the flow velocity for normal depth is used.

The higher the percentage of flow, the faster the velocity used to calculate the convex routing coefficient, and the closer the routed hydrograph will be to a pure translation of the inflow hydrograph.

The user-specified percentage can be modified in the calculation options. A typical value is around 75%, but this value may be modified for oddly shaped hydrographs with sharp, uncharacteristic peaks or for calibration purposes.

Weighted Translation Routing

The convex routing method is only valid when the convex routing coefficient, c , is less than 1 or when the hydrologic time step is less than the calculated travel time. For certain cases in which the travel time exceeds the hydrologic time step, SewerCAD automatically uses an alternative method of routing called weighted translation routing.

Each ordinate of the outflow hydrograph is derived from a weighted average of the ordinates for the current and previous time steps of the inflow hydrograph. The weights are calculated based on the convex routing coefficient.

Each ordinate of the outflow hydrograph is calculated as follows:

$$O_t = \frac{1}{c} I_{t-\Delta t} + \left(1 - \frac{1}{c}\right) I_t$$

where

O_t = outflow at current time step (l/s, gpm),

c = convex routing coefficient

$I_{t-\Delta t}$ = inflow at previous time step (l/s, gpm), and

I_t = inflow at current time step (l/s, gpm).

Hydrologic and Hydraulic Time Steps

SewerCAD uses two distinct time steps when running an EPS.

- **Hydrologic time step:** This time step is used to calculate the routed hydrographs and represents the time increment of all hydrographs generated during the analysis. The hydrologic time step is also used as the calculation increment for the pressure calculations.
- **Hydraulic time step:** This time step represents how often the hydraulic calculations are performed for gravity flow. Flows are interpolated from the previously generated hydrographs using the hydraulic time step and are then used to perform the gradually varied flow analysis for that time step.

The hydrologic time step should be less than or equal to the hydraulic time step, and the hydraulic time step should be a multiple of the hydrologic time step.

8.4 SewerCAD, SewerGEMS and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software. Information about SewerGEMS can be found in Chapter 9.

What Does SewerCAD Do?

SewerCAD is a powerful, easy-to-use program for the design and analysis of wastewater collection systems. SewerCAD's intuitive graphical interface and powerful data exchange capabilities make it easy to develop and load complex models of combined gravity and pressure networks. Using SewerCAD, you can design and analyze the gravity portion of the system according to either a gradually varied flow calculation or a standard capacity analysis.

The powerful automatic design capabilities of SewerCAD allow you to design the system based on user-defined constraints for velocity, cover, and slope. You can also design for partially full pipes, multiple parallel sections, maximum section size, invert/crown-matching criteria, and the allowance of drop structures. You can disable the design feature on a pipe-by-pipe basis, to design all or only a portion of your system.

SewerCAD's powerful loading model provides complete support for all types of sanitary and wet-weather loads. The unit sanitary load library is completely user-customizable and supports population, area, discharge, and count-based sanitary loads, as well as hydrographs and load patterns. Wet-weather inflow can also be added to the model as instantaneous loads, hydrographs, or base loads with associated patterns. Infiltration can be computed based on pipe length, pipe diameter/length, pipe surface area, or unit count. Loads are peaked according to a PF or EFF equations that you select from a user-customizable EFF library.

How Can You Use SewerCAD?

You can use SewerCAD to design new systems and analyze the performance of existing systems. SewerCAD's intuitive graphical editor and powerful scenario management capabilities facilitate the process of analyzing a large number of design alternatives and finding potential problems in an existing system. In summary, SewerCAD's sophisticated modeling techniques enable you to

- design and analyze multiple sanitary sewer networks in a single project;
- examine your system using SewerCAD's gradually varied flow algorithms or a standard capacity analysis;
- design the system using SewerCAD's automatic, constraint-based design;
- load your model based on contributing population, service area, total sanitary flow, or your own loading type;
- peak your loads using Babbitt's, Harmon's, 10 States, and Federov's equations, or use your own formulas or tables;
- calculate infiltration based on pipe length, diameter, surface area, length/diameter, or user-defined data;
- generate plan and profile plots of a network;

- perform EPSs that include time-variable loads and hydrologic routing; and
- animate plans and profiles showing sanitary sewer system performance over time.

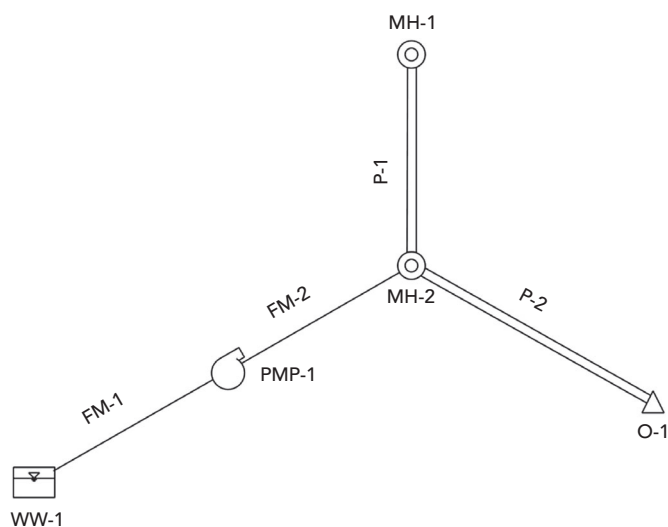
The theory and background used in SewerCAD are presented in this chapter and in the SewerCAD online help system.

SewerCAD can analyze complex sewage collection systems under a variety of loading conditions. SewerCAD's powerful loading capabilities can be used to track system response to an unlimited range of sanitary and wet-weather loading combinations. It can be used for

- pipe sizing
- pump sizing
- master planning
- operational studies
- rehabilitation studies

8.5 Problems

1. In the sanitary sewer represented by the following diagram, a load from a development of high-rise apartments (at wet well WW-1) with 10,000 residents is pumped to the top of a hill, where it is then transported via a circular gravity line to a treatment plant (represented by O-1). At manhole MH-1 is a resort apartment with 300 guests. The load generated here flows down a circular gravity pipe to the same gravity line mentioned above. A bar that serves, on average, 50 people per day and 2 large cafeterias with 20 employees each are located near manhole MH-2.



Schematic for Problem 1

Pipe Data for Problem 1

Pipe	Pipe Type	Material	Diameter (mm)	Length (m)	Upstream Invert (m)	Downstream Invert (m)
P-1	Gravity	PVC $n = 0.010$	200	100	98.90	97.62
P-2	Gravity	PVC $n = 0.010$	375	200	97.62	96.62
FM-1	Pressure	Ductile iron $C = 130$	300	1	70.00	70.00
FM-2	Pressure	Ductile iron $C = 130$	350	1,000	70.00	97.62

Node Data for Problem 1

Node	Ground Elevation (m)	Rim Elevation (m)	Sump Elevation (m)	Structure Diameter (m)
MH-1	100.30	100.30	98.90	1.00
MH-2	99.10	99.10	97.62	1.00
O-1	100.00	100.00	96.62	N/A
WW-1	74.00	—	—	—

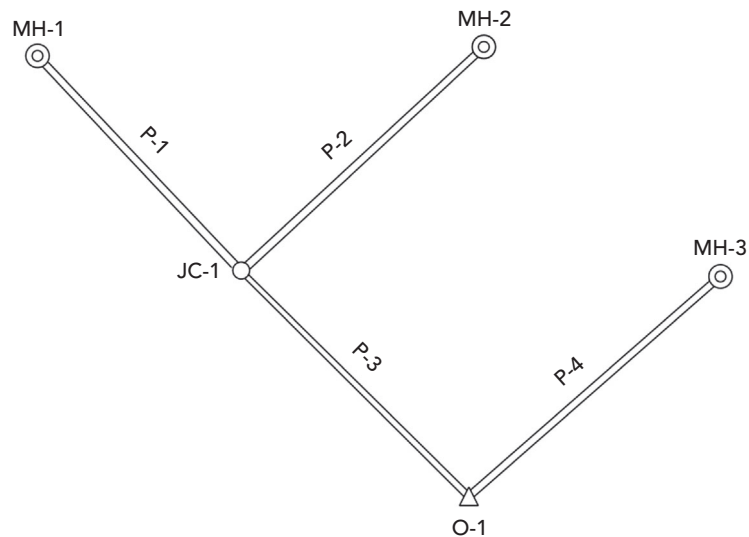
The base and minimum elevations for the wet well are both 70 m. The initial elevation is 73 m, and the maximum elevation is 73.5 m. The diameter of the circular wet well is 3 m.

The tailwater elevation at outlet O-1 is 98 m.

The three defining points of the pump curve are 0 m³/min at 33.33 m, 5.00 m³/min at 25.00 m, and 10 m³/min at 0 m. The pump's elevation is 70 m.

- (a) What is the total peak sanitary outflow if no peaking factor method is applied to the four unit sanitary loads mentioned above? If Babbitt's peaking factor method is applied? If Harmon is applied? Which peaking factor method is the most conservative?
- (b) With the Harmon peaking factor applied to each of the four unit dry loads, what is the hydraulic grade at MH-2? How does this peaking factor change the hydraulic load and the flow velocity of pipe P-1 from when no peaking factor was applied?
- (c) Identify and describe any problems for each of the three scenarios from part (a).
- (d) The estimated infiltration rate for each of the concrete gravity pipes in the proposed sanitary sewer system represented below is 20 l/day/mm km. The estimated inflow into each of the manholes during a 5-year storm event is approximately 75 l/day. The ground elevations for MH-1, MH-2, MH-3, JC-1, and O-1 are 12.10, 12.10, 11.20, 11.80, and 10.25 m, respectively. The top of junction chamber JC-1 is 11.8 m. The tailwater condition is a free outfall.

Use SewerCAD's automatic design feature to design the inverts and sizes of concrete pipes in the proposed sanitary sewer. The design constraints and pipe lengths are given in the tables below. Apply both the wet-weather (given above) and dry-weather sanitary loads (given in the table below). Use Federov's equation to calculate the peaking factor for each of the unit sanitary loads.



Schematic for Problem 2

Pipe and Constraint Data for Problem 2

Pipe	Length (m)	Constraint	Minimum	Maximum
P-1	200	Velocity (m/s)	0.5	3.0
P-2	150	Cover (m)	1.0	3.0
P-3	200	Slope (m/m)	0.005	0.1
P-4	300			

Node Data for Problem 2

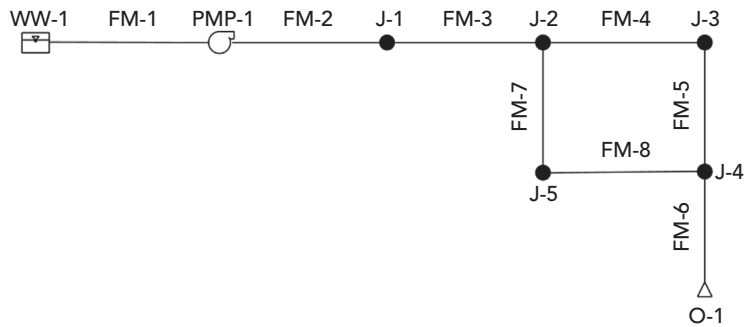
Node	Unit Sanitary Load	Loading Unit	Loading Unit Count
MH-1	Hospital (medical) per bed	Bed	400
	Cafeteria per employee	Employee	20
	Apartment resort	Guest	100
MH-2	Apartment	Resident	400
	Shopping center per employee	Employee	100
	Laundromat per wash	Wash	200
MH-3	School (boarding)	Student	500

Sanitary Sewer Design

Problems

- (a) List the diameter and slope for each newly designed pipe. Are there any problems with the designed system?
 - (b) What percentage of the total flow is wet-weather flow? Which pipe has the most infiltration?
 - (c) For a larger magnitude storm, the inflow rate into each manhole is estimated at 100 l/day. Analyze the model using the previously designed system and apply the larger wet-weather loading. What percentage of the total flow is wet-weather loading?
 - (d) Do you consider the amount of wet-weather flow into the system significant? What are some methods for alleviating infiltration?
3. The following network is an initial design for a system of force mains in a sanitary sewer. All pipes are ductile iron (Hazen–Williams' $C = 130$). The pump, PMP-1, is at an elevation of 691 m. Enter the system data given in the tables below and answer the following questions. The ground elevation at the outlet is 715 m and the sump elevation is 712 m.

Hints: Make sure that the **Fixed Level in Steady State** box is checked under the "Section" tab of the Wet Well dialog. In addition, before running the model, make the following modification to the calculation options: within the GO dialog, click the **Options** button, scroll to the right, and in the **Pressure Hydraulics** tab, check the **Use Pump Loads** box.



Schematic for Problem 3

Pump and Wet Well Data for Problem 3

Pump Curve Point	Head (m)	Flow (m ³ /min)	Ground elevation (m)	694.7
Shutoff	30	0	Maximum elevation (m)	694.0
Design	20	4	Initial elevation (m)	691.5
Max operating	5	6	Minimum elevation (m)	688.5
			Base elevation (m)	687.0
			Diameter (m)	3.0

Junction Data for Problem 3

Pressure Junction	Elevation (m)
J-1	698
J-2	701
J-3	703
J-4	705
J-5	703

Pipe Data for Problem 3

Pipe	Diameter (mm)	Length (m)	Upstream Invert (m)	Downstream Invert (m)
FM-1	400	2	687.00	691.00
FM-2	250	58	691.00	698.00
FM-3	350	45	698.00	701.00
FM-4	350	88	701.00	703.00
FM-5	350	71	703.00	705.00
FM-6	450	67	705.00	712.00
FM-7	450	39	701.00	703.00
FM-8	450	68	703.00	705.00

- (a) What is the head loss across the entire system?
- (b) Why is there more flow in FM-7 than FM-4?
- (c) Fill in the following table.

Answer Table for Part (c) of Problem 3

Pipe	Flow (m ³ /s)	Velocity (m/s)	Head Loss (m)
FM-2			
FM-3			
FM-4			
FM-5			
FM-6			
FM-7			
FM-8			

- (d) If the minimum velocity required in the force main to keep particles from settling is 0.6 m/s, which areas are going to have problems?
- (e) What are some possible changes to the design to fix the problem portions of the system?
- (f) Implement a solution suggested in part (e). Describe the fix(es) and fill in the following chart.

Answer Table for Part (f) of Problem 3

Pipe	Diameter (mm)	Flow (m ³ /s)	Velocity (m/s)	Head Loss (m)
FM-2				
FM-3				
FM-4				
FM-5				
FM-6				
FM-7				
FM-8				

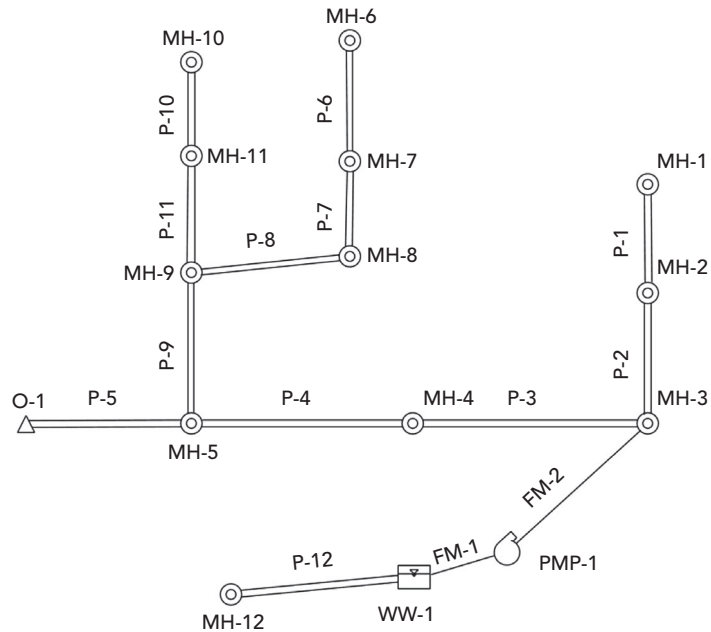
4. A major interceptor along a river collects laterals from subdivisions. The lower residential area loads are collected in a wet well and pumped to the major interceptor on the other side of a hill. The layout of the system is shown below. All pipes shown as double lines are circular, concrete, gravity-flow pipes ($n = 0.013$). The two pressure pipes (FM-1 and FM-2) are ductile iron (Hazen–Williams’ $C = 130$). There is an overflow diversion at MH-5. All input data is given below.

To determine the performance of the system, set up and run three scenarios:

1. a steady-state analysis of the average (base) sanitary loading only,
2. an extended-period analysis of the sanitary loading only, and
3. an extended-period analysis of both the sanitary and wet-weather loading.

For the extended-period analyses, use a 24-h duration with a 1-h hydraulic time step and a 0.1-h hydrologic time step.

Hint: For this problem, it is only necessary to create one Sanitary Loading Alternative that will contain the base and pattern loads. During a steady-state analysis, SewerCAD will ignore the time-based pattern. However, it will be necessary to create two infiltration and inflow loading alternatives (one without the wet-weather loads and the other with the wet-weather loads) because Scenario 2 should not consider the wet-weather loads. Define the wet-weather loading pattern in the manhole prototype before laying out the network.



Schematic for Problem 4

Pipe Data for Problem 4

Pipe	Section Size (in)	Upstream Invert Elevation (ft)	Downstream Invert Elevation (ft)	Length (ft)
P-1	18	146.50	141.50	400
P-2	18	141.50	136.00	400
P-3	24	136.00	119.00	400
P-4	36	119.00	114.00	400
P-5	36	114.00	94.00	400
P-6	12	157.00	146.50	400
P-7	18	146.50	136.50	400
P-8	18	136.50	126.00	400
P-9	24	126.00	114.00	400
P-10	12	147.00	136.50	400
P-11	18	136.50	126.00	400
P-12	18	74.50	45.00	400
FM-1	24	45.00	45.00	2
FM-2	24	45.00	136.00	60

Node Data for Problem 4

Node	Ground Elevationn (ft)	Sump Elevation (ft)
MH-1	150.00	146.50
MH-2	145.00	141.50
MH-3	140.00	136.00
MH-4	129.00	119.00
MH-5	124.00	114.00
MH-6	160.00	157.00
MH-7	150.00	146.50
MH-8	140.00	136.50
MH-9	130.00	126.00
MH-10	150.00	147.00
MH-11	140.00	136.50
MH-12	80.00	74.50
Outlet	100.00	94.00
PMP-1	45.00	N/A
WW-1	60.00	N/A

Overflow Data for MH-5 in Problem 4

System Flow (gpm)	Divertedw Flow (gpm)
0	0
20,000	0
30,000	10,000

All manholes have a sanitary base load of 100 gpm and a continuous diurnal pattern applied to them as defined below. Each manhole also has wet-weather loading given by the following hydrograph.

Domestic Pattern Data for Problem 4

Time(h)	Multiplier
Starting	0.4
3	1.0
6	1.4
9	1.2
12	1.4
15	0.9
21	0.6
24	0.4

Wet-Weather Loading Data for Problem 4

Time (h)	Flow (gpm)
0.00	0
3.00	0
4.00	100
5.00	300
6.00	900
7.00	1,500
8.00	1,800
9.00	1,600
10.00	1,000
11.00	600
12.00	300
13.00	100
14.00	0

Wet Well Data for Problem 4

Wet Well	Ground Elevation (ft)	Base Elevation (ft)	Minimum Elevation (ft)	Initial Elevation (ft)	Alarm Elevation (ft)	Maximum Elevation (ft)	Wet Well Diameter (ft)
WW-1	60.00	45.00	45.00	55.00	59.00	60.00	20.00

Pump Data for Problem 4

Pump Curve Point	Head (ft)	Flow (gpm)
Shutoff	100	0
Design	80	4,000
Max Operating	40	8,000

The pump turns on when the elevation in the wet well rises to 57.0 ft and shuts off when the elevation drops to 45.0 ft.

To verify important information such as minimum and maximum velocities, it will be helpful to keep a table like the following.

Variable	Steady State	EPS Sanitary Only	EPS Dry and Wet
Maximum flow at outlet (gpm)			
Time of max. flow at outlet (h)	N/A		
Maximum velocity in system at this time (ft/s)			

Sanitary Sewer Design

Problems

Pipe with max. velocity at this time			
Minimum flow at outlet (gpm)	N/A		
Time of min. flow at outlet (h)	N/A		
Min. velocity in system at this time (ft/s)			
Pipe with min. velocity at this time			
Maximum diverted flow (gpm)			
At what hour?			

- (a) Fill in the table for each scenario.
- (b) Is there surcharging or flooding in the system? Explain the difference between surcharging and flooding.
- (c) Does the diversion divert any flow? If so, for which scenario?
- (d) If the pump has a best efficiency point at 4,000 gpm, what can you say about its performance for the dry- and wet-weather scenarios?
- (e) Plot profiles from MH-1 to the outlet for all three scenarios and compare the HGLs.

9

Low-Impact Development and Dynamic Modeling

9.1 Introduction

It was a typical summer, but one accentuated with periods of extreme heat and intense rainfall. One area in the city was nearing the completion of a 2-year beautification and utility project that had local residents and merchants convinced would solve a history of minor, inconvenience flooding. Unfortunately, one of the thunderstorms resulted in damaging flooding at multiple locations. The headlines in the local paper the next morning read “2.7 Inches of Rain in 70 Minutes: \$50 Million to End Flooding.” The State Climatologist reported the event return period was between 5 and 10 years. Certainly, a modern urban drainage system should accommodate runoff from such an event, particularly since local regulations require sewer systems designed to accommodate peak flows from a 25-year event. What happened? Why was there so much flooding?

When they designed the local drainage system, engineers used traditional methods to size the storm sewer pipes, culverts, and open channels to accommodate estimated peak flows. They did not consider the impact of limited storage in conduits, and the additive effects and timing of local inflows. Nonuniform flow profiles were determined to delineate the floodplain. Otherwise, downstream boundary conditions and interactions among the different drainage system components were not considered. All calculations were performed assuming steady flows.

The crux of the problem was that the steady flow methods did not account for the dynamic response of the watershed and its internal drainage system. No true picture, that is, understanding of the drainage system performance was established. The results were insufficient and potentially misleading. Consequently, there was a lack of adequate planning and flood prevention measures.

To overcome these deficiencies, the city undertook a study to identify the causes of flooding and possible solutions. That study included a hydraulic analysis of the drainage system using a dynamic model. The results provided a clear understanding of the performance of individual components and their interactions at a system wide scale.

The purposes for this chapter are to introduce dynamic modeling and demonstrate its application to the analysis and design of hydraulic systems using the SewerGEMS computer software. The option to include low-impact development (LID) to reduce the overall surface water runoff due to land planning and green infrastructure will be discussed. Example applications include stormwater conveyance systems that include green infrastructure design to size stormwater sewer networks.

9.2 Why to Use an Unsteady Flow Model?

Although unsteady flow models are computationally more sophisticated than steady flow models, they represent the physical processes influencing flow and its properties better and provide a more detailed understanding of real flow events. Also in many scenarios, LID structures (that is, green roof, rain garden, and parking lot with porous pavement) are relatively small, could be numerous in an urban design, and have dynamically different impacts on the runoff hydrograph from a wide range of storm events. An unsteady flow model always should be used when hydrograph attenuation and flow restrictions are considered. Storage within a reach can be significant, which leads to substantial attenuation. Flow restrictions increase storage, thereby amplifying attenuation. These effects have further implications at downstream confluences and structures. An unsteady flow model accounts for these storage effects and thereby predicts total system performance more accurately.

A steady flow model projects the peak flow entering one branch of a conveyance system and translates it downstream, where it is added to the peak flows from other branches. As such, the peak flow in the downstream links likely is overpredicted. This approach is conservative, but not realistic, particularly for input to planning for system operation and maintenance, and potential expansion. In other words, an unrealistic picture of system performance can amount to substantial time and capital improvement costs.

Typically, the steady flow approach is conservative in its determination of water levels in that, other things being equal, steady flow analysis tends to overestimate flow and thus depth. This is due to the assumption flow is constant within a reach; as a consequence, the effects of in-stream storage on the hydrograph shape, peak, and timing are ignored.

There are instances, however, in which a steady flow model may not be conservative in its estimation of water levels. For cases in which flow reversal may occur, the designer must be aware of the implications of his or her choice of flow modeling techniques. Examples of these situations include stream junctions where flow may reverse such as the confluence of a tributary and a tidal river where the flood tide can generate a wave as it moves upstream and the waterway narrows.

Dynamic modeling opens a whole new realm of design alternatives and solutions than were previously possible with steady flow modeling. It is more

Low-Impact Development and Dynamic Modeling

Low-Impact Development

accurate because it improves the hydraulic wave effects observed in the field during periods when there are significant changes in loading patterns, such as weekends and holidays when water consumption and wastewater production patterns change, sometimes dramatically.

This discussion has identified some of the reasons and applications for unsteady modeling. Although steady flow equations are significantly simpler and easier to use, their solutions are accurate only for those situations in which the channel and flow conditions do not vary greatly. When faced with more complex modeling problems, the unsteady flow equations provide far more appropriate solutions.

9.3 Low-Impact Development

The urbanization of land has a direct impact on the volume of runoff and the peak flow rate. Urban areas tend to reduce the amount of water that would infiltrate into the subsurface, eliminate vegetation that would intercept rainfall, and increase the velocity that the runoff travels over the developed surfaces (that is, roads, rooftops, and parking lots). This will increase the volume of runoff, and since the runoff travel time to a discharge location is reduced, the peak flow rate will be greatly increased. Traditional strategies for stormwater management focused on reducing the peak flow rate from urban runoff by storing runoff (that is, detention ponds), and channeling the stormwater out of the urbanized areas using engineered channels and piping systems. However, there is a growing awareness of constructing engineered systems to mimic natural processes in urban areas can better manage stormwater runoff.

LIDs are techniques to mimic the natural hydrology that existed before urbanization. They tend to be small-scale systems that are located near the origin of the runoff to store stormwater and slow its travel time. This would not only decrease the peak flow rate but also change the timing of runoff from urban areas that utilize these strategies. A further benefit is the potential to allow for stormwater to infiltrate then replenish groundwater systems, have more “green” space in urban areas, and can improve the water quality leaving the urban environment.

There are several LID techniques that can be utilized in urban design. Most LID techniques are considered to be structural best management strategies (BMPs) used to control and treat runoff close to the origin of the runoff. They tend to be smaller than traditional stormwater structures and spread out throughout the urban area. The Water Environment & Reuse Foundation maintains a database on the effectiveness of these LID techniques (www.bmpdatabase.org). Table 9.1 compares some LID techniques to traditional stormwater management structures. These LID structures will reduce the volume of runoff, increase the time of concentration (travel time), trap or reduce pollutants, and allow for increased infiltration and evapotranspiration.

Table 9.1: Comparison of Traditional Stormwater Structures and Low-Impact Development Structures.

Traditional Structure	Low-Impact Development Structure
Catch basins and pipes	Vegetated swales, infiltration trench, constructed runoff filters
Stormwater detention basins	Planted basins, bioretention cells, rain barrels, subsurface infiltration beds, wet ponds
Large centralized detention basin	Several small detention basins, rain gardens, constructed wetlands
Impervious surfaces	Pervious pavement, green roofs, landscape restoration, soil restoration

The design objectives for LID in urban areas is to manage the peak flow rates and maximum water level of stored water, while simultaneously protecting the urban infrastructure (buildings, roads, and other structures), human lives, and surrounding environment. The use of LIDs, since they tend to be smaller and decentralized, requires the hydraulic modeler to carefully examine the predicted runoff peak flows rates, volumes, and resulting hydrographs throughout the system design since potentially many LID structures are in use at one time. As the smaller LID structures fill and overflow into other traditional structures used to manage the runoff, a dynamic model with detail of how all these structures are hydraulically connected is needed.

A challenge to add LIDs to existing urban areas is to find locations to place proposed LIDs. The acquisition of land to build these structures is limited in many urbanized areas and if a site is found, it may not be optimally placed. Also, the soil for many urban areas tends to have a low infiltration rate or additional water in the urban subsurface could have undesirable consequences, which would make LIDs that use infiltration impractical to implement. Regardless of the challenges, the use of LIDs for managing stormwater is gaining popularity, and it is common to evaluate many design alternatives when these structures are considered. A detailed hydraulic model of the stormwater system can be used to evaluate how LIDs should be placed and sized to manage the runoff and improve the water quality leaving urban areas.

9.4 Dynamic Modeling

Simply stated, dynamic means change. Flow in sewers and open channels changes in time and space. Flow that changes with time is unsteady; flow that changes in space is nonuniform (see Chapter 1). Dynamic models account for unsteady, nonuniform flow.

In unsteady open-channel flow, the velocity and depth at any location (cross section) continually change due to changes in loading patterns and boundary conditions. Unsteady open channel flow by nature is also nonuniform. Consequently, dynamic models are also known as unsteady flow models.

St. Venant Equations

Flows in stormwater conveyance systems and sewers are usually free surface open-channel flows. The governing equations are the St. Venant equations, based on the principles for the conservation of mass and linear momentum.

$$\frac{\partial Q}{\partial x} + \frac{\partial(A + A_o)}{\partial t} - q = 0$$

$$\frac{\partial Q}{\partial t} + \frac{\partial(\beta Q^2 / A)}{\partial x} + gA \left(\frac{\partial y}{\partial x} + S_o + S_f + S_i \right) + L = 0$$

where

Q = discharge (cfs, m³/s)

x = distance along the longitudinal axis of the channel (ft, m)

A = active cross-sectional area of flow (ft², m²)

A_o = inactive (off-channel storage) cross-sectional area of flow (ft, m)

t = time (s)

q = lateral inflow or outflow (ft²/s, m²/s)

β = coefficient for nonuniform velocity distribution in cross section

g = gravitational acceleration constant (ft/s², m/s²)

y = flow depth (m, ft)

S_o = slope of channel bed in longitudinal direction

S_f = friction slope

S_i = slope due to severe local expansion and contraction effects

L = momentum effect of lateral flow (ft³/s², m³/s²)

The first equation is the conservation of mass or continuity equation. It relates the change in the cross-sectional area of flow at a point over time to the change in flow over space.

The second equation is the conservation of momentum equation based on Newton's second law, which states if a net force is acting on a body, the momentum of that body will change, where momentum is mass times acceleration. This equation was derived by considering how forces applied to a control volume affect the movement of water through the control volume. For example, if the driving forces on the control volume (due to gravity and water surface slope) are greater than the resistive forces, the flow will accelerate. The acceleration can be an increase in velocity at one point over time (local or temporal

acceleration), an increase in velocity over space (convective or spatial acceleration), or both. Conversely, if the resistive forces are greater, the flow will decelerate, meaning the flow slows and the depth increases, which frequently happens upstream of undersized road cross-drains during periods of stormwater runoff.

The local and convective acceleration terms (respectively, the first two terms in the momentum equation) are inertial terms. The other terms are the various applied forces (pressure imbalance, gravity, friction, and expansion/convergence effects), expressed as slope terms. The friction slope generally is evaluated using an empirical equation in the same way as the loss term in the energy equation when performing flow profile analyses. The Manning equation is commonly used, but other similar equations may also be used. This loss term encompasses not only the effects of boundary friction but also all processes creating flow resistance, notably turbulence and internal shear.

The St. Venant equations are also known as the shallow water or dynamic wave equations. Two assumptions implicit in the derivation of these equations are: (a) the flow is one-dimensional and (b) the flow is gradually varied. The first assumption means it is only necessary to consider velocities in the downstream direction and not in the transverse or vertical directions. This reduces the cross-sectional properties to single parameters, that is, cross-sectional area and average velocity. The second implies the streamlines are nearly parallel. This means the vertical pressure distribution is essentially hydrostatic, that is, increases linearly with depth, and the friction slope term can be approximated with one of the uniform flow formulae.

Numerical Solution

There is no known analytical solution to the St. Venant equations; therefore they are solved numerically. The solution method in SewerGEMS uses a finite difference scheme that converts the system of partial differential equations to a system of algebraic equations involving the unknowns at discrete points in time and space.

Specifically, a weighted four-point implicit scheme is used. An implicit scheme is preferred over explicit since these schemes have the advantage of maintaining numerical stability for large computational time steps and exhibit robustness in modeling systems that integrate the complex hydraulic interactions encountered in gravity sewer systems. The scheme was chosen because it handles unequal distance steps, its stability–convergence properties can be conveniently modified, and the internal and external boundary conditions can be easily applied.

The numerical analog uses the following four-point finite-difference quotients to approximate the first- and zero-order derivative terms:

$$\frac{\partial f}{\partial t} = \frac{f_i^{j+1} + f_{i+1}^{j+1} - f_i^j - f_{i+1}^j}{\Delta t}$$

$$\frac{\partial f}{\partial t} = \frac{f_i^{j+1} + f_{i+1}^{j+1} - f_i^j - f_{i+1}^j}{\Delta t}$$

$$f = \frac{\theta(f_i^{j+1} + f_{i+1}^{j+1}) + (1 - \theta)(f_i^j + f_{i+1}^j)}{2}$$

in which f represents the unknown variables, Q and A , and θ is a weighting factor. The weighted four-point implicit scheme is unconditionally stable for $\theta > 0.5$. The optimal range of θ for maintaining stability and accuracy for large computational time steps is 0.6–0.8.

The finite difference equations are solved with a Newton–Raphson iteration routine that includes an algorithm that iterates banded matrixes and is computationally efficiency.

Initial and Boundary Conditions

Solution of the dynamic wave equations requires initial and boundary conditions. Initial conditions prescribe the state of the system for the first time step and usually are specified as dry channel or baseflow. Boundary conditions are functional relationships between depth (or area) and flow at the upstream and downstream ends of the reach of interest. (*Note:* Reach of interest indicates the length of channel or sewer through which a hydrograph is being routed.) Effectively, boundary conditions are constraints the numerical solution must meet, particularly if a stable and accurate solution is to be obtained. Boundary conditions usually consist of the inflow hydrograph and a tailwater condition at the outlet.

The numerical solution may use one or both sets of boundary conditions, depending on flow classification. This is explained with the concept of characteristics, which are mathematical expressions that describe the speed and path by which a physical disturbance, or its influence, propagates though the x, t solution domain. A physical disturbance is any change in the rate and/or depth of flow due to influences such as an inflow hydrograph and flow constriction.

There are two characteristics: forward and backward, indicated symbolically as C^+ and C^- and expressed mathematically as

$$C^+ : \frac{dx}{dt} = V + c$$

and

$$C^- : \frac{dx}{dt} = V - c$$

where x, t are longitudinal distance and time, v is cross-sectional average velocity, and c is the speed at which the wave effect created by the disturbance travels. This wave speed is called celerity and is evaluated as

$$c = \sqrt{gD}$$

where g is gravity and D is the hydraulic depth of water (see Chapter 1).

For flow in a rectangular channel, the Froude number is expressed in terms of wave celerity as

$$F = \frac{V}{\sqrt{gD}} = \frac{V}{c}$$

For subcritical flow, $F < 1$, meaning $c > V$. The backward characteristic is nonzero and negatively sloped, which means there is a mathematical basis for information to travel upstream. That is, the influence of a downstream disturbance will propagate upstream and affect the flow conditions there. A well-known example is the backwater curve that forms upstream of an under-sized culvert. This causes an increased depth of flow at the culvert inlet as the headwater increases to a higher state of specific energy to overcome the flow resistance. This influence propagates upstream, forming the backwater curve. Determining the upstream conditions depends greatly on the downstream boundary condition. In other words, accurate solution of the St. Venant equations for subcritical flow requires properly defined upstream and downstream boundary conditions.

For supercritical flow, $F > 1$, meaning $V > c$. Both the forward and backward characteristics are nonzero and positively sloped. Therefore all physical disturbances are swept downstream. As such, determining flow conditions in the reach of interest depends only on the upstream boundary condition.

For flows near critical depth, the location of the boundary conditions is changing and the numerical solution may become unstable. Numerical instability when the flow is near critical depth is one of the greatest problems encountered when modeling unsteady flow.

9.5 Hydrograph Routing Effects

The dynamic equations are solved to route a hydrograph through a link from the upstream or inflow node to the downstream or outflow node. Routing is a mathematical procedure to describe the movement and change in shape of a stormwater hydrograph as it travels through a channel, storm sewer, or pond, and of a wastewater hydrograph as it passes through a pump station, force main, or gravity sewer. Routing involves the processes of translation and attenuation, where translation typically is measured by the time-shift between

Low-Impact Development and Dynamic Modeling

Simplified Routing Models

the occurrence of the inflow and outflow hydrograph peaks or the centers of mass. Attenuation is the reduction in peak flow rate resulting from the spread of the hydrograph shape. Spreading occurs due to diffusion and dispersion caused by varying conduit conditions that alternately accelerate and decelerate the flow, for the cases of channel and sewer routing, and hydrograph storage and release, for the cases of pond and pump station routing. Spreading results in elongation of the time base, reduction of the peak flow rate, and skewing of the hydrograph shape. This leads to the development of a “tail” to the recession limb. Diffusion is spreading that does not involve mass (or volume) shift, whereas dispersion involves mass transfer from the rising limb to the recession limb.

9.6 Simplified Routing Models

The dynamic wave equations are the standard for routing hydrographs through channels and sewers. Historically, two perceived drawbacks to their wholesale general use have been the large data requirements and the necessity for sophisticated numerical integration. Very often, based on channel conditions and hydrograph characteristics, it is possible to make simplifying assumptions that allow one to use approximations to these equations. When this is possible, advantages in terms of ease of solution and reduced data requirements are often realized, but at the cost of potential loss of solution accuracy. Currently, many widely accepted stormwater conveyance models incorporate such approximations. Some of the approximate methods are discussed in this section.

The range of models available for evaluating unsteady flow in conveyance systems can be classified into two categories: hydraulic and hydrologic. The dynamic, diffusion, and kinematic wave models are hydraulic routing models. They are based on the conservation equations for mass and momentum, or simplified forms of these equations, and analyze both flow and hydraulic grade through the system. They also are referred to as distributed parameter models because they explicitly account for spatial variations in channel properties. As such, they calculate flow and depth at several cross sections along the reach of interest simultaneously.

Hydrologic models, such as the Muskingum and Muskingum–Cunge, are based on continuity and an empirical relationship between channel storage and discharge; therefore they are not as rigorous as the hydraulic models and do not accommodate spatial variations in channel properties. They are termed lumped parameter models in that the channel properties are assumed constant along a reach. They calculate flow at one location—the downstream end of a reach.

Diffusion Wave

Two approximations to the dynamic wave model are the diffusion and kinematic wave models. The diffusion wave model that assumes the inertia terms

in the momentum equation are negligible compared with the force terms for pressure, friction, and gravity. The diffusion wave equations are continuity and the following reduced form of momentum.

$$\frac{\partial y}{\partial x} = S_o - S_f$$

For prismatic channels, the diffusion wave equations can be combined into the single equation

$$\frac{\partial Q}{\partial t} + c \frac{\partial Q}{\partial x} = D \frac{\partial^2 Q}{\partial x^2}$$

where c is wave celerity and D is a hydrograph diffusion term. This equation is in the classic form of the advective–diffusive transport equation widely used for water quality modeling. This equivalence indicates the diffusion wave model transports mass—the hydrograph—and accounts somewhat for the diffusion of the hydrograph shape. This model applies to situations where the acceleration effects are minimal and there are minor backwater effects.

One of the challenging features associated with flow in stormwater conveyance systems is interchanging regimes between subcritical and supercritical flows. This is due to the fact that an urban conveyance system may experience a wide range of conduit slopes and it is common to have significant slope changes at pipe junctions. A good numerical model must be able to handle the mixed flow regimes and interchanges with robustness. For certain mixed flow applications, the dynamic routing technique using the four-point implicit numerical scheme tends to be less numerically stable than diffusion routing. This is attributed to the fact that the diffusion wave model neglects the acceleration terms. To take advantage of the diffusion model's stability and to retain the accuracy of the dynamic wave model, the solution technique in SewerGEMS employs a numerical filter on the acceleration terms.

Kinematic Wave

The kinematic wave model neglects all dynamic terms, which includes the derivative terms for local and convective acceleration, and pressure imbalance. This reduces the momentum equation to

$$S_f \approx S_o$$

which means it can be approximated with one of the uniform flow formulae. The kinematic wave model accounts only for hydrograph translation and not for any diffusion or dispersion. The most common application for kinematic wave modeling is overland flow during periods of stormwater runoff.

Muskingum Routing

The Muskingum routing method was developed for use in a flood control study in the Muskingum River basin in Ohio, hence the origin of its name. This method uses a lumped parameter model that treats the channel reach of interest as a single spatial element, that is, the reach is not divided into shorter segments, and the channel cross-sectional geometry, slope, and roughness are assumed constant. Model input is the inflow hydrograph at the upstream end and model output is the outflow hydrograph from the downstream end. This method does not compute flows at intermediate locations along the reach as do hydraulic models.

The model governing equations are the hydrologic form of continuity

$$I - Q = \frac{dS}{dt}$$

and a conceptual model for reach storage

$$S = K[\theta I + (1 - \theta)Q]$$

I is inflow, Q is outflow, S is reach storage, K is a timing parameter interpreted as travel time, and θ is a weighting coefficient.

These equations are solved using a simple finite difference technique that yields the following routing equation:

$$Q_2 = \frac{-K\theta + 0.5\Delta t}{K(1 - \theta) + 0.5\Delta t} I_2 + \frac{K\theta + 0.5\Delta t}{K(1 - \theta) + 0.5\Delta t} I_1 + \frac{K(1 - \theta) - 0.5\Delta t}{K(1 - \theta) + 0.5\Delta t} Q_1$$

The subscripts refer to inflow and outflow values at the beginning and end of a computational time step, Δt .

The Muskingum method traditionally has been applied to channels where there are recorded inflow and outflow hydrographs that are used to determine the model parameters that give a best fit between observed and simulated outflow hydrographs. As such, the model is calibrated to the channel. It then is applied to other inflow hydrographs to predict the outflow hydrographs.

While the Muskingum method has been used for decades, values for the coefficients, K and θ , are not readily available or easily obtained for measured hydrographs. A general rule of thumb is that K can be estimated by the travel time through the reach and θ lies between 0.2 and 0.4. The upper limit for θ is 0.5. Values less than 0.5 define dispersing hydrograph shapes. θ equal to 0.5 defines a hydrograph that does not change shape and only exhibits translation effects. A value of 0.0 is the practical lower limit for θ and suggests inflow has little effect. Some users have reported values for θ from 0.4 to 0.5 for natural

streams, while others have suggested 0.1–0.3. Unfortunately, these guidelines are imprecise and somewhat conflicting.

Muskingum–Cunge Method

To overcome the limitation with parameter estimation and extend the method to ungaged channels, Cunge (1969) developed hydraulic-based equations to estimate the routing parameters by matching coefficients of the advective–diffusive transport form of the Muskingum routing model

$$\frac{\partial Q}{\partial t} + \frac{\Delta x}{K} \frac{\partial Q}{\partial x} = \left[\Delta x (1 - \theta) c_k - \frac{1}{2} \frac{\Delta x^2}{K} \right] \frac{\partial^2 Q}{\partial x^2}$$

to those of the diffusion wave model.

$$\frac{\partial Q}{\partial t} + c_k \frac{\partial Q}{\partial x} = \frac{Q_R}{2TS_o} \frac{\partial^2 Q}{\partial x^2}$$

In these equations, c_k is kinematic celerity ($= (5Q_R / 3A)$), T is channel top width, and Q_R is a reference discharge value, typically chosen as two-thirds of the peak flow rate.

Cunge obtained the following expressions to estimate K and θ .

$$K = \frac{\Delta x}{c_k}$$

$$\theta = \frac{1}{2} \left[1 - \frac{Q_R}{TS_o c_k \Delta x} \right]$$

When these equations are used to estimate the routing parameters, the method is known as the Muskingum–Cunge routing method. Numerical experience that has shown negative values for θ can sometimes occur but do not cause numerical problems.

An important feature and a perceived advantage of the Muskingum–Cunge model is that it is diffusive for $\theta < 0.5$ and offers the same advantages of the diffusion wave model. This method is included in many contemporary stormwater models.

9.7 SewerGEMS and Tutorials

Tutorials and additional problems using the Bentley OpenFlows software can be found at aka.bentley.com/CAiHE.supplement. If you have trouble

Low-Impact Development and Dynamic Modeling

SewerGEMS and Tutorials

accessing the site, please send a message to bipress@bentley.com. The tutorials are designed for student users to become familiar with the design and analysis of hydraulic systems without needing to spend a lot of time inputting data. As new features are added to the software and as the software is upgraded, the tutorials will be updated too. There will also be additional problems added to challenge the students and encourage further use of the software. Information about SewerGEMS can be found in Chapter 9.

What Does SewerGEMS Do?

SewerGEMS is a fully dynamic, multiplatform program for the analysis and design of urban stormwater conveyance systems, and wastewater collection and pump station systems. SewerGEMS is a dynamic unsteady flow model rather than a steady state or standard step model. It is therefore capable of delivering results far more accurately and closer to real life than a steady flow model. Dynamic models allow the effects of storage and backwater in channels and sewers, and the timing of the hydrographs, to yield a true representation of the hydraulic grade line (HGL) at any point in space and time. It provides a fast solution for analyzing the performance of complex conveyance systems subject to time varying loading patterns and dynamic boundary conditions.

SewerGEMS includes several LID structures. These include a Bioretention Cell, Green Roof, Infiltration Trench, Porous Pavement, Rain Barrels, Rain Gardens, Rooftop Disconnection, and Vegetated Swales. These structures can be modeled for a wide range of storm events and design scenarios.

Some of the other features of SewerGEMS include geographic information systems (GIS) integration, results animation, and scenario management. The GIS integration allows the user to build complete data bases using shapefiles, coverages, and geodatabases. The results animation allows one to review and present model results in customizable animations that offer an unparalleled visualization of model results. Scenario management provides an easy way to examine “what if” scenarios within a single project without the need of adding redundant data.

SewerGEMS's implicit fully dynamic engine provides numerical solutions for the toughest interconnected pipe, pond, and open-channel networks, with stunning performance and numerical robustness. SewerGEMS will account for storage within structures and quantify overflows should they occur. The dynamic engine will precisely calculate flow splits at manholes with multiple outgoing pipes and can handle complex, looped systems.

SewerGEMS provides a broad range of functions for importing and allocating sewer loads in the form of hydrographs, patterned loads, and unit loads. SewerGEMS can model pump stations with both constant-speed and variable-speed pumps. It uses state-of-the-practice methods to estimate storm-water runoff.

How Can You Use SewerGEMS?

You can use SewerGEMS to accurately model every aspect of your stormwater system, including rainfall, runoff, inlet capture and bypass, gravity and pressure piping, detention ponds, outlet structures, open channels, and culverts. You can use SewerGEMS for analysis and design purposes, to generate planning input, and as an education and research tool. With SewerGEMS, you can analyze all sanitary and combined sewer system elements in one package and have the option of performing the analyses with the SWMM algorithm or the four-point implicit solution of the full St. Venant equations, which is faster and more stable.

Enjoy the easy-to-use stand-alone interface, or choose to work directly inside AutoCAD, ArcGIS, or MicroStation. As one of the four modeling environments in SewerGEMS, the ArcGIS interface let you leverage ESRI's geodatabase architecture to guarantee a single dataset for modeling and GIS. With SewerGEMS, you can build your sewer model from scratch with simple drag-and-drop layout tools or by leveraging existing data sources to drastically streamline the model development process. You can utilize an unlimited number of background images (DXF, SHP, TIF, SID, BMP, JPG, and others) to simplify model development and improve presentation. You can create customized output tables and graphs.

You can use the scenario manager with a single model to evaluate an unlimited number of design, operational, loading, and even network topology scenarios for better decision-making support. You can use this feature to answer a long list of the proverbial "what if" questions.

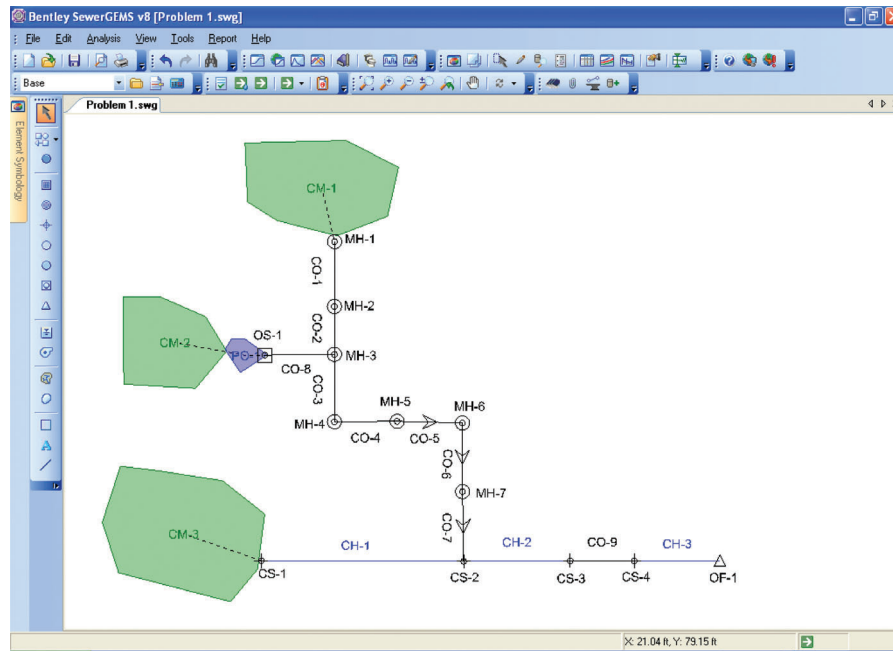
You can use the result's animation to view the real-time performance selected portions of the system and the impact of their interaction. This feature is important when analyzing system performance, particularly for forensic studies into why a system is not performing as expected.

9.8 Problems

The following diagram shows the stormwater conveyance system for an area that underwent staged development from agricultural to suburban land use. CM-1 was developed first, prior to any stormwater regulations. Following standard practice at the time, stormwater runoff was collected in the sewer system and diverted to an outfall into the creek at cross section of CS-2. CM-2 was developed after stormwater regulations were in place. To control the runoff peak from a 10-year design rainfall at or below the predevelopment rate, a detention pond was constructed between the outlet from CM-2 and the point where local runoff enters the sewer system at MH-3. During the development period, CM-3 was built out for mixed land use—multifamily residential, commercial, and a park. Runoff from CM-3 flows directly to the existing creek.

Low-Impact Development and Dynamic Modeling

Problems



Tables are included that provide data for the catchments, channel, cross sections, conduits, manholes, and pond. The pond outflow structure is a two-stage riser with lower stage orifice (two 6-in. diameter openings at elevation 533.25 ft MSL, orifice coefficient = 0.60) and an upper stage rectangular, sharp crested weir (length = 1 ft, weir coefficient = 3.10, and crest elevation at 536.75 ft MSL).

Downstream of the confluence of the storm sewer and creek, there is a road crossing with a culvert cross-drain. Conduit CO-9 is the culvert. Note that it connects CS-3 and CS-4. This system discharges to a free outfall.

Following build-out of CM-1, CM-2, and CM-3, the road crossing began to flood (overtop) more frequently. Local residents noted the sewer system between manholes MH-4 and MH-6 was surcharging with occasional aboveground flooding during the events when the crossing overtopped. This resulted in minor system flooding. It also was noted during these events the water depth in the creek was above the crown of the outfall sewer. The residents quickly concluded that the minor system flooding they were experiencing was due to an undersized culvert at the road crossing and demanded that the city make appropriate repairs. The city engineer analyzed the system and determined the inconvenience flooding they were experiencing was not due to an undersized culvert, but probably to an excessive pond release rate.

Apply SewerGEMS to analyze this system. Use the default curvilinear NRCS unit hydrograph, NRCS Type II rainfall distribution, and a 24-h rainfall depth of 5.15 in.

Were the citizens correct about the cause of flooding in the sewer system or was the city engineer correct?

Were both parties wrong? Was the flooding due to other conditions?

Would it be a more appropriate solution—not necessarily cheaper in terms of material costs—to replace one or more links of sewer with a larger pipe? If so, which one(s)?

Hint: Run animations for a profile between the detention pond and the system outfall.

Catchment Data for Problem 1

Catchment	Area (ac)	Curve Number	Time of Concentration (min)
CM-1	3.30	82	15
CM-2	4.20	85	15
CM-3	15.0	78	26

Channel Data for Problem 1

Channel	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Length (ft)
CH-1	527.00	526.00	2,000
CH-2	526.00	524.80	2,000
CH-3	524.20	524.00	1,000

Cross-Sectional Data for Problem 1

Cross Section	Geometry	Bottom Width (ft)	Left Side Z	Right Side Z	Manning <i>n</i> -Value	Invert Elevation (ft MSL)
CS-1	Trapezoid	5.0	1	0	0.030	527.00
CS-2	Trapezoid	7.0	1	1	0.050	526.00
CS-3	Trapezoid	7.0	1	1	0.050	524.80
CS-4	Trapezoid	10.0	1	1	0.500	524.20

Conduit Data for Problem 1

Conduit	Diameter (in.)	Length (ft)	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Manning <i>n</i> -Value
CO-1	24	325	533.80	533.00	0.013
CO-2	24	330	533.00	532.20	0.013

(Continued)

Low-Impact Development and Dynamic Modeling

Problems

Conduit	Diameter (in.)	Length (ft)	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Manning n -Value
CO-3	24	278	532.20	530.90	0.013
CO-4	24	290	530.90	530.10	0.013
CO-5	24	305	530.10	529.40	0.013
CO-6	24	345	529.40	529.00	0.013
CO-7	24	490	529.00	526.00	0.013
CO-8	24	88	533.25	532.20	0.013
CO-9	18	66	524.80	524.20	0.013

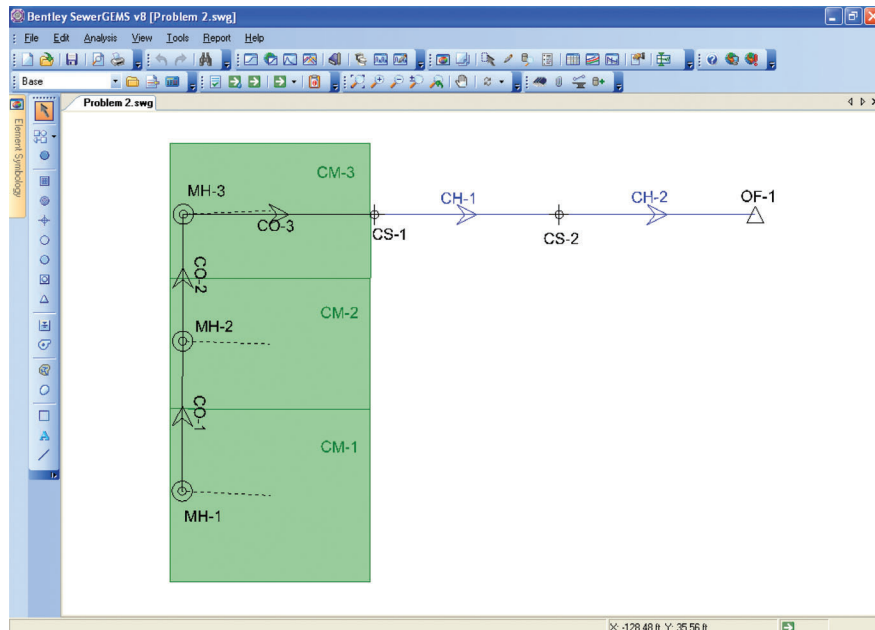
Manhole Data for Problem 1

Manhole	Diameter (ft)	Head Loss Method	Ground Elevation (ft MSL)	Invert Elevation (ft MSL)
MH-1	3	HEC-22	537.00	533.80
MH-2	3	HEC-22	536.80	533.00
MH-3	3	HEC-22	535.50	532.00
MH-4	3	HEC-22	535.00	530.90
MH-5	3	HEC-22	534.00	530.10
MH-6	3	HEC-22	532.00	529.40
MH-7	3	HEC-22	531.80	529.00

Pond Data for Problem 1

Elevation (ft MSL)	Surface Area (ac)
533.25	0.23
534.00	0.26
535.00	0.29
536.00	0.32
537.00	0.35

This figure shows a stormwater drainage system at an apartment complex located on a barrier island. Runoff from the apartment buildings flows across adjacent, contiguous paved parking lots to the stormwater collection system. The parking lots are graded and curbed to create surface storage along the low-lying, back edge. This design was chosen because the tidal control on the sewer outflow potentially can surcharge the sewer to the point of surface overflow. As such, the low-lying portion of the parking lot becomes a surcharge detention storage system. The stormwater inlets comprised a series of combination inlets located in sag meaning each inlet has 100% capture.



The accompanying tables provide data about the system components and tidal data from a nearby U.S. Geological Survey gaging station. The system has a free outfall into a tidal creek, that is, it has a tidal boundary condition. The outfall ground elevation is 5.00 ft MSL with invert elevation of 3.80 ft MSL.

Apply SewerGEMS to analyze this system. Use the default curvilinear NRCS unit hydrograph, NRCS Type III rainfall distribution, and a 24-h rainfall depth of 5.30 in.

Part of the design is setting curb height to contain any surcharge overflow. Is a curb height of 4 in. sufficient?

Does the flow in the sewer reverse during high tide? In other words does water flow from the tidal creek into the sewer? Or does any system surcharge result due to a “backwater effect?”

What is the duration of any about ground surcharge during the design event? What signage should be placed along the lower portion of the parking lot?

What would be the effect of upsizing one or more of the sewer pipes? Explain your answer in terms of the effects of in-line storage. Does this suggest in-line storage can be a solution at other, noncoastal locations?

Low-Impact Development and Dynamic Modeling

Problems

Tidal Curve Data for Problem 2

Hours	Elevation (ft)	Hours	Elevation (ft)	Hours	Elevation (ft)
0	4.90	9	2.73	18	1.33
1	5.08	10	3.78	19	0.97
2	4.74	11	4.70	20	1.23
3	3.96	12	5.33	21	1.93
4	2.99	13	5.55	22	2.84
5	2.03	14	5.22	23	3.76
6	1.35	15	4.35	24	4.53
7	1.26	16	3.24		
8	1.81	17	2.18		

Catchment Data for Problem 2

Catchment	Area (ac)	Curve Number	Time of Concentration (min)
CM-1	1.00	84	9.6
CM-2	0.67	85	6.0
CM-3	1.10	88	15.0

Channel Data for Problem 2

Channel	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Length (ft)
CH-1	4.00	3.90	500
CH-2	3.90	3.80	500

Cross-Sectional Data for Problem 2

Cross Section	Geometry	Bottom Width (ft)	Left Side Z	Left Side Z	Manning <i>n</i> -Value	Invert Elevation (ft MSL)
CS-1	Trapezoid	2.0	0.5	0.5	0.045	4.00
CS-2	Trapezoid	6.0	0.5	0.5	0.045	3.90

Conduit Data for Problem 2

Conduit	Diameter (in.)	Length (ft)	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Manning <i>n</i> -Value
CO-1	18	325	5.50	5.45	0.013
CO-2	18	330	5.45	5.40	0.013
CO-3	18	66	5.40	4.00	0.013

Manhole Data for Problem 2

Manhole	Diameter (ft)	Head Loss Method	Ground Elevation (ft MSL)	Invert Elevation (ft MSL)
MH-1	3	HEC-22	7.50	5.50
MH-2	3	HEC-22	7.45	5.45
MH-7	3	HEC-22	7.50	5.40

A nearby town proposes to add a new school onto its wastewater collection system. The new school will discharge into MH-4 and connect with the system trunk line at MH-3. From there, the wastewater flows to the local treatment plant located at OF-1. There are concerns about system capacity, particularly since this addition is immediately downstream of the location where a manifold pump station system discharges into the trunk line. Specific concerns include the potential for system surcharge with surface overflow at MH-3.

The figure and accompanying tables summarize pertinent system data. The system has a free outfall at the head of the wastewater treatment plant with ground elevation of 263.70 ft MSL and invert elevation of 260.00 ft MSL.

Use SewerGEMS to analyze this system. Perform simulations for a full diurnal loading pattern at MH-1, WW-1, WW-2, and MH-4. Both pump stations are duplex stations, so use two pumps at each station, operating in lead-lag status. It is highly recommended that each station has two identical pumps. Data are provided for two recommended pumps; evaluate system performance for both pumps at each pump station.

Will MH-3 surcharge? If so, will it surcharge aboveground? If so, is the flooding at MH-3 sensitive to which pump is used at the two pump stations?

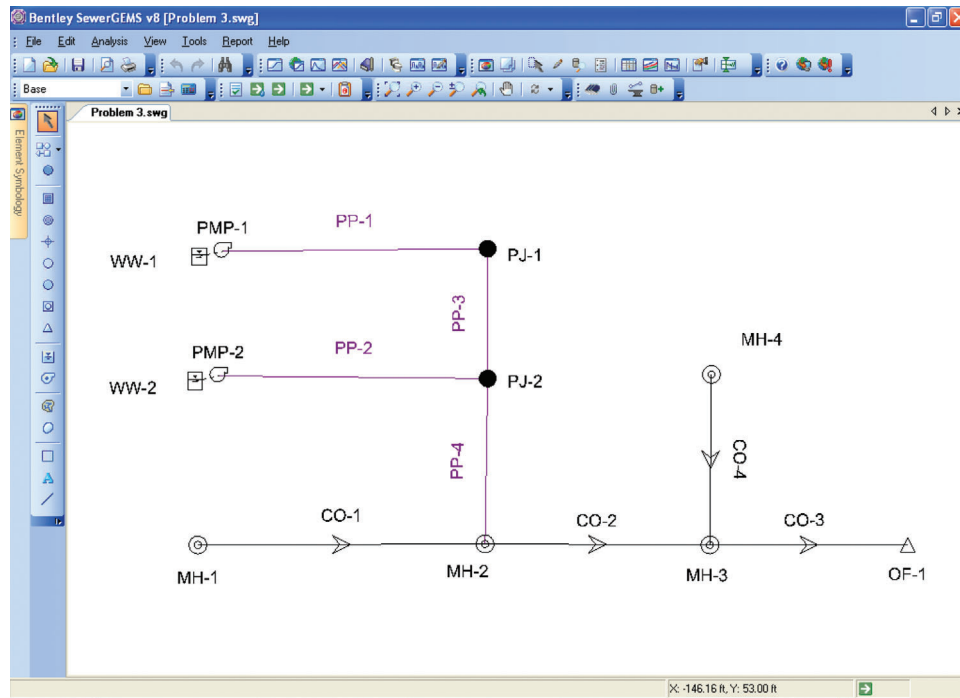
What are the maximum and minimum pressures at PJ-2?

Local regulations stipulate the minimum pump runtime must be 2 min or longer, the average number of pump cycles per hour should be in the range of 4–6, and wet well detention time should not exceed 30 min. Does each pump station meet regulations?

What are the minimum and maximum HGL values at MH-3?

Low-Impact Development and Dynamic Modeling

Problems



*Average Daily Wastewater Loading
Data for Problem 3*

Node	Flow (gpm)
WW-1	40
WW-2	50
MH-1	1,000
MH-4	1,125

Dimensionless Diurnal Wastewater Loading Curve Data for Problem 3

Hours	Multiplier	Hours	Multiplier	Hours	Multiplier	Hours	Multiplier
1	0.49	7	0.46	13	2.50	19	1.60
2	0.46	8	0.51	14	2.20	20	1.42
3	0.44	9	0.63	15	1.68	21	0.94
4	0.42	10	0.76	16	1.41	22	0.76
5	0.40	11	1.08	17	1.34	23	0.63
6	0.43	12	1.51	18	1.39	24	0.54

Pump 1 Data for Problem 3

Head (ft)	Flow (gpm)
86.0	0
66.0	200
44.0	300

Pump 2 Data for Problem 3

Head (ft)	Flow (gpm)
78.0	0
72.0	450
42.0	800

Pump Station Data for Problem 3

Pump Station	Lead-On Elevation (ft)	Lag-On Elevation (ft)	Pumps-Off Elevation (ft)	Ground Elevation (ft)	Cross-Sectional Area (sq ft)
WW-1	196.50	197.25	195.00	202.00	35
WW-2	250.50	251.50	248.75	255.00	30

Pressure Pipe (Force Main) Data for Problem 3

Pipe	Diameter (in.)	Length (ft)	Minor Loss Coefficient	Hazen-Williams Coefficient
PP-1	6	860	2	130
PP-2	6	900	2	130
PP-3	6	1000	0	130
PP-4	6	800	0	130

Pressure Junction Data for Problem 3

Junction	Ground Elevation (ft MSL)	Node Elevation (ft MSL)
PJ-1	222.00	219.33
PJ-2	264.00	260.00

Conduit Data for Problem 3

Conduit	Diameter (in.)	Length (ft)	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Manning <i>n</i> -Value
CO-1	18	500	271.00	267.00	0.013
CO-2	18	400	267.20	263.00	0.013

(Continued)

Low-Impact Development and Dynamic Modeling

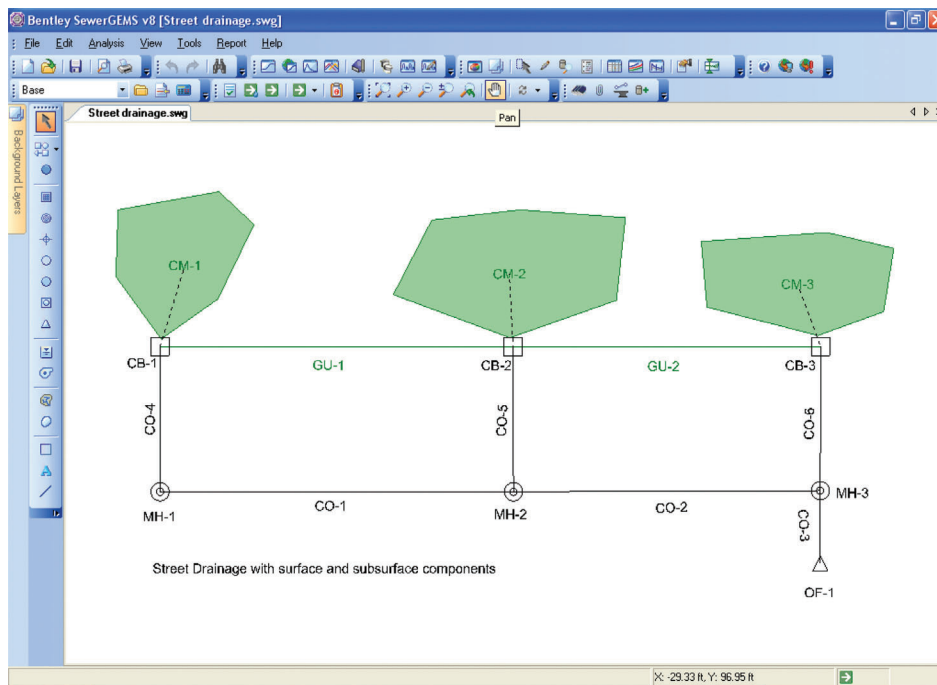
Problems

Conduit	Diameter (in.)	Length (ft)	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Manning <i>n</i> -Value
CO-3	18	700	271.00	260.00	0.013
CO-4	12	500	264.60	263.00	0.013

Manhole Data for Problem 3

Manhole	Diameter (ft)	Head Loss Method	Ground Elevation (ft MSL)	Invert Elevation (ft MSL)
MH-1	3	Standard	275.00	271.00
MH-2	3	HEC-22	270.00	267.00
MH-3	3	HEC-22	266.00	263.00
MH-4	3	HEC-22	268.00	264.60

The following diagram shows the layout for a stormwater conveyance system proposed for a 400-ft segment of a local street. Runoff from the street and adjacent areas (indicated as catchments CM-1, CM-2, and CM-3) flows into the gutter and enters the underground sewer system through inlets at three catch basins. The gutter is a triangular channel formed by the intersection of a 6-in. barrier curb and the transversely sloped ($H:V = 40:1$) street surface. Pertinent data about system components are given in the tables below.



The system has a free outfall with ground and invert elevations of 97.50 and 93.00 ft MSL, respectively.

Apply SewerGEMS to analyze the system. Use the default curvilinear NRCS unit hydrograph, NRCS Type III rainfall distribution, and a 24-h rainfall depth of 5.10 in. Use the HEC-22 Headloss Method to compute head losses in the catch basins and manholes.

Will the system surcharge, that is, have HGL above the crown of the pipe, at one or more locations if the tailwater at the free outfall is at elevation 95 ft? If so, which pipes, manholes, and catch basins surcharge? Will the roadway flood? If so, where?

If the system will surcharge, can the problem be resolved by upsizing one link of pipe? If so, which link should be upsized? What is the minimum upsized pipe diameter? What are the implications about in-line detention storage?

Are any of the proposed catch basins undersized?

Catchment Data for Problem 4

Catchment	Area (ac)	Curve Number	Time of Concentration (min)
CM-1	1.00	90	7
CM-2	1.10	92	8
CM-3	0.8	91	5

Gutter Data for Problem 4

Gutter	Length (ft)	Constructed Slope (ft/ft)	Manning n -Value
GU-1	200	0.003	0.013
GU-2	200	0.003	0.013

Catch Basin Data for Problem 4

Catch Basin	Length (ft)	Width (ft)	Ground ELEV (ft MSL)	Invert ELEV (ft MSL)	Inlet Type	Max Inflow or Capture Efficiency
CB-1	3.0	3.0	5.50	5.45	Maximum capacity	1.50 cfs
CB-2	3.0	3.0	5.45	5.40	Maximum capacity	2.00 cfs
CB-3	3.0	3.0	5.40	4.00	Full capture	100%

Low-Impact Development and Dynamic Modeling

Problems

Manhole Data for Problem 4

Manhole	Diameter (ft)	Ground Elevation (ft MSL)	Invert Elevation (ft MSL)
MH-1	3	100.00	96.00
MH-2	3	99.50	95.50
MH-7	3	99.00	95.00

Conduit Data for Problem 4

Conduit	Diameter (in.)	Length (ft)	Upstream Invert Elevation (ft MSL)	Downstream Invert Elevation (ft MSL)	Manning <i>n</i> -Value
CO-1	18	200	96.00	95.50	0.013
CO-2	18	200	95.50	95.00	0.013
CO-3	18	200	95.00	93.00	0.013
CO-4	18	20	97.00	96.00	0.013
CO-5	18	20	96.50	95.50	0.013
CO-65	18	20	96.00	95.00	0.013

Regulations governing stormwater ponds vary from the classic statement “the post development peak discharge cannot exceed the pre-development peak discharge for a given design storm event” to the 10% and no net increase rules. Under the 10% rule, the engineer must demonstrate no increase in peak flow over predevelopment conditions at a downstream location where the land use change area is no more than 10% of the total contributing area at that point. At watersheds where there is substantial hydrograph attenuation due to in-stream storage and routing effects or the presence of a pond would delay the timing of peak outflow from a downstream pond such that it more closely aligned with the peak runoff from upstream areas, thereby creating a greater peak than would have occurred naturally, this criterion may actually be used to show no pond is required on-site. Calculations to demonstrate this clearly must include unsteady pond and channel routing.

The no net increase rule extends the requirements for stormwater ponds beyond peak runoff control and also requires no net increase in runoff volume. Some interpretations also hold the outflow hydrograph shape and timing must not be altered substantially. Here again, unsteady pond and channel routing is required.

Given a 100-ac watershed with the following land use conditions: 20 ac in forest (good condition) over HSG C soils; 40 ac in open space (good condition) over HSG B soils; 35 ac in single-family residential over HSG B soils; and 5 ac

in commercial land use over HSG B soils. The watershed hydraulic length is 3,750 ft and the area weighted average watershed slope is 1.6%.

At the upper end of this watershed, it is proposed to develop 10 ac of the open space land use to single-family residential land use. Runoff from this development will be controlled by a stormwater pond (inverted quadrilateral frustum with 3:1 side slopes and length to width ratio of 2:1), which is to be designed according to controlling regulations. The 10-ac site currently has an average slope of 1.8%, which after development will be reduced to 1.5%. The hydraulic length is 1,280 ft.

The channel connecting outflow from the 10-ac development to the outlet of the 100-ac watershed (10 ac or 10% plus 90 ac or 90%) is 3,200 ft long with an approximate trapezoidal cross section with the following parameters: base width = 8 ft; side slope Hor:Vert = 2.5:1.0; longitudinal slope = 0.3%; Manning n -value = 0.065.

The design rainfall is a 24-h event with 4.80-in. accumulation. Use the NRCS synthetic unit hydrograph.

Apply SewerGEMS to design a stormwater pond at the outlet of the 10-ac development site such that (a) the peak flow at the outlet of the 100-ac basin (downstream point to check the 10% rule for the 10-ac site) does not exceed the predevelopment rate for the design storm event and (b) the peak and volume at the outlet of the 100 ac do not violate the “no net increase rule.”

Bibliography



- AASHTO Task Force on Hydrology and Hydraulics, 1991, *Model Drainage Manual*, American Association of State Highway and Transportation Officials, Washington, DC.
- Almeida A.B. and E. Koelle, 1992, *Fluid Transients in Pipe Networks*, International Series on Computational Engineering. Computational Mechanics Publications.
- American Society of Civil Engineers, 1963, *Friction Factors in Open Channels: Progress Report of the Task Force on Friction Factors in Open Channels of the Committee on Hydrodynamics of the Hydraulics Division*, American Society of Civil Engineers, New York, NY.
- American Society of Civil Engineers and the Water Environment Federation, 1992, *Design and Construction of Urban Stormwater Management Systems*, American Society of Civil Engineers, New York, NY.
- American Society of Civil Engineers and the Water Environment Federation, 2007, *Gravity Sanitary Sewer Design and Construction*, American Society of Civil Engineers, New York, NY.
- Babbitt, H.E. and E.R. Baumann, 1958, *Sewerage and Sewage Treatment*, John Wiley & Sons, New York, NY.
- Benedict, R.P., 1980, *Fundamentals of Pipe Flow*, John Wiley & Sons, New York, NY.
- Bodhaine, G.L., 1968, Measurement of Peak Discharge at Culverts by Indirect Methods, In *U.S. Geological Survey Techniques of Water-Resources Investigations*, U. S. Department of the Interior, Washington, DC, Book 3, Chapter A3.
- Brater, E.F. and H.W. King, 1976, *Handbook of Hydraulics*, McGraw-Hill, New York, NY.
- Brown, S.A., J.D. Schall, J.L. Morris, C.L. Doherty, S.M. Stein, and J.C. Warner, 2009, *Urban Drainage Design Manual Hydraulic Engineering Circular No. 22*, 3rd ed., U.S. Department of Transportation, Federal Highway Administration, Washington, DC.
- Cesario, A.L., 1995, *Modeling, Analysis, and Design of Water Distribution Systems*, American Water Works Association, Denver, CO.
- Chow, V.T., 1959, *Open-Channel Hydraulics*, McGraw-Hill, New York, NY.



Bibliography

- Clark, R.M., W.M. Grayman, R.M. Males, and A.F. Hess. 1993. Modeling Contaminant Propagation in Drinking Water Distribution Systems. *Journal of Environmental Engineering*, 119(2), 349.
- Cunge, J.A. 1969. On the Subject of a Flood Propagation Computation Method (Muskingum Method). *Journal of Hydraulic Research*, 7(2), 205.
- Debo, T.N. and A.J. Reese, 1995, *Municipal Storm Water Management*, CRC Press, Boca Raton, FL.
- Featherstone, R.E. and C. Nalluri, 1982, *Civil Engineering Hydraulics*, Granada, New York, NY.
- Federal Aviation Administration (FAA), 1970, Report A/C 150-5320-58, Advisory Circular on Airport Drainage, U.S. Department of Transportation, Washington, DC.
- Federation, 1982, *Gravity Sanitary Sewer Design and Construction*, American Society of Civil Engineers, New York, NY, ASCE Manual of Practice, No. 60.
- Frederick, R.H., V.A. Meyers, and E.P. Auciello, 1977, Five to 60-minute Precipitation Frequency for the Eastern and Central United States, In *NOAA Technical Memorandum NEW Hydro-35*. www.nws.noaa.gov/oh/hdsc/studies/prcpfreq.html (accessed June 2004).
- French, R.H., 1985, *Open-Channel Hydraulics*, McGraw-Hill, New York, NY.
- Great Lakes-Upper Mississippi River Board of Sanitary Engineers, 1997, *Recommended Standards for Wastewater Facilities*, Health Research, Inc., Albany, NY.
- Hathaway, G.A. 1945. Design of Drainage Facilities. *Transactions, American Society of Civil Engineers*, 110, 697.
- Harmon, W.G. 1918. Forecasting Sewage at Toledo Under Dry Weather Conditions. *Engineering News-Record*, 80.
- Haestad Methods, 2000, *CulvertMaster: Culvert Design & Analysis Software*, Bentley Systems.
- Haestad Methods, 2000, *FlowMaster: Open Channel, Pipe & Inlet Software*, Bentley Systems.
- Haestad Methods, 2000, *StormCAD: Storm Sewer Design & Analysis Software*, Bentley Systems.
- Haestad Methods, 2001, *SewerCAD: Sanitary Sewer Modeling Software*, Bentley Systems.
- Haestad Methods, 2002, *PondPack: Urban Hydrology & Detention Pond Modeling Software*, Bentley Systems.

- Haestad Methods, 2002, *WaterGEMS: Geospatial Water Distribution Software for GIS*, Bentley Systems.
- Haestad Methods, 2002, *Proceedings of the Water Security Summit*, Haestad Press, Waterbury, CT, December 3–4, 2001.
- Haestad Methods, 2007, *HAMMER V8 XM Edition User's Manual*, Bentley Systems.
- Haestad Methods and R. Durrans, 2003, *Stormwater Conveyance Modeling and Design*, Haestad Press, Waterbury, CT.
- Haestad Methods, G. Dyhouse, J. Hatchett, and J. Benn, 2003, *Floodplain Modeling Using HEC-RAS*, Haestad Press, Waterbury, CT.
- Haestad Methods, T.M. Walski, T.E. Barnard, L.B. Merritt, E. Harold, N. Walker, and B. Whitman, 2004, *Wastewater Collection System Modeling and Design*, Haestad Press, Waterbury, CT.
- Haestad Methods, T.M. Walski, D.V. Chase, D.A. Savic, W.A. Grayman, S. Beckwith, and E. Koelle, 2003, *Advanced Water Distribution Modeling and Management*, Haestad Press, Waterbury, CT.
- Henderson, F.M., 1966, *Open Channel Flow*, Macmillan Publishing Co. Inc., New York, NY.
- Henderson, F.M. and R.A. Wooding. 1964. Overland Flow and Groundwater Flow from a Steady Rain of Finite Duration. *Journal of Geophysical Research* 69(8), 1531.
- Hersfield, D.M., 1961, Rainfall Frequency Atlas of the United States for Durations from 30 Minutes to 24 Hours and Return Periods from 1 to 100 Years, In *Technical Paper 40*, U.S. Department of Commerce, Weather Bureau. www.nws.noaa.gov/oh/hdsc/studies/prcpfreq.html.
- Horton, R.E. 1939. Analysis of Runoff Experiments with Varying Infiltration Capacity. *Transactions of the American Geophysical Union* 20, 693-711.
- Huff, F.A. and J.R. Angle, 1992, Rainfall Frequency Atlas of the Midwest, In *Bulletin 71*, Illinois State Water Survey, Champaign, IL.
- Hwang, N.H.C. and C.E. Hita, 1987, *Hydraulic Engineering Systems*, Prentice-Hall, Englewood Cliffs, NJ.
- Hydraulics Research Station, 1951, *Velocity Equations for Hydraulic Design of Pipes*, Metric ed., Her Majesty's Stationary Office, London.
- Izzard, C.F. 1946. Hydraulics of Runoff from Developed Surfaces. *Proceedings, Highway Research Board* 26, 129.
- Jakovlev, S.V., J.A.A. Karelm, A.I. Zukov, S. Kolobanov, 1975, *Kanalizacija*, Moscow: Stroizdat.



Bibliography

- Johnstone, D. and W.P. Cross, 1949, *Elements of Applied Hydrology*, Ronald Press, New York, NY.
- Joint Task Force of the American Society of Civil Engineers and the Water Pollution Control.
- Kerby, W.S. 1959. Time of Concentration Studies. *Civil Engineering* 60, 174.
- Kirpich, Z.P. 1940. Time of Concentration of Small Agricultural Watersheds. *Civil Engineering* 10(6).
- Masch, F.D., 1984, *Hydrology*, Washington, DC: Federal Highway Administration, HEC No. 19.
- Males R.M., W.M. Grayman, and R.M. Lark. 1988. Modeling Water Quality in Distribution System. *Journal of Water Resources Planning and Management* 114, 197-209.
- Mays, L.W., 2001, *Stormwater Collection Systems Design Handbook*, McGraw-Hill, New York, NY.
- McKay, M. and D.S. Wilkes, 1995, *Atlas of Short-Duration Precipitation Extremes for the Northeastern United States and Canada*, Northeast Regional Climate Center Research Series Publication No. RR95-1, Ithaca, NY.
- Metcalf & Eddy, 1981, *Wastewater Engineering: Collection and Pumping of Wastewater*, McGraw-Hill, New York, NY.
- Miller, J.F., R.H. Frederick, and R.J. Tracey, 1973, *Precipitation-Frequency Atlas of the Coterminous Western United States*, Silver Spring, MD, National Weather Service, NOAA Atlas 2, 11 vols.
- National Resources Conservation Service, 2001, *National Engineering Handbook Part 630 – Hydrology*, U.S. Department of Agriculture, Washington, DC. <http://www.wcc.nrcs.usda.gov/water/quality/common/neh630/4content.html> (accessed June 2004).
- Normann, J.M., R.J. Houghtalen, and W.J. Johnson, 2001, *Hydraulic Design of Highway Culverts, Hydraulic Design Series No. 5*, 2nd ed., Federal Highway Administration, Washington, DC.
- Pilgrim, D.H., 1987, *Australian Rainfall and Runoff*, The Institute of Engineers, Barton, Australia.
- Rawls, W., P. Yates, and L. Asmussen, 1976, *Calibration of Selected Infiltration Equations for the Georgia Coastal Plain*, U.S. Department of Agriculture, Agricultural Research Service, Washington, DC, Technical Report ARS-S-113.
- Roberson, J.A., and C.T. Crowe, 1990, *Engineering Fluid Mechanics*, 4th ed., Houghton Mifflin Company, Boston, MA.
- Roberson, J.A., J.J. Cassidy, and H.M. Chaudhry, 1988, *Hydraulic Engineering*, Houghton Mifflin Company, Boston, MA.

- Rossman, L.A. and P.F. Boulos. 1996. Numerical Methods for Modeling Water Quality in Distribution Systems: A Comparison. *Journal of Water Resources Planning and Management* 122(2), 137.
- Rossman, L.A., 1993, *EPANet User's Manual (AWWA Workshop Edition)*, Risk Reduction Engineering Laboratory, Office of Research and Development, US Environmental Protection Agency, Cincinnati, OH.
- Rossman, L.A., R.M. Clark, and W.M. Grayman. 1994. Modeling Chlorine Residuals in Drinking-Water Distribution Systems. *Journal of Environmental Engineering* 120(4), 803.
- Rouse, H., and S. Ince. 1957. History of Hydraulics. Iowa City, Iowa. Iowa Institute of Hydraulic Research.
- Sanks, R.L., 1998, *Pumping Station Design*, 2nd ed., Butterworth-Heinemann, Inc., Stoneham, MA.
- Schaake, J.C. Jr., J.C. Geyer, and J.W. Knapp. 1967. Experimental Examination of the Rational Method. *Journal of the Hydraulics Division* 93(HY6).
- Schall, J.D., P.L. Thompson, S.M. Zerges, R.T. Kilgore, and J.L. Morris, 2012, *Hydraulic Design Of Highway Culverts, Hydraulic Design Series No. 5*, 2nd ed., Federal Highway Administration, Washington, DC.
- Simon, A.L., 1976, *Practical Hydraulics*, John Wiley & Sons, New York, NY.
- Singh, V.P., 1988, *Hydrologic Systems, Volume I: Rainfall-Runoff Modeling*, Prentice Hall, Englewood Cliffs, NJ.
- Singh, V.P., 1992, *Elementary Hydrology*, Prentice Hall, Englewood Cliffs, NJ.
- Soil Conservation Service, 1969, *National Engineering Handbook*, Section 4, Chapter 16, National Resources Conservation Service, Washington, DC. www.wcc.nrcs.usda.gov/water/quality/common/neh630/4content.html (accessed June 2004).
- Soil Conservation Service, 1986, *TR-55, Urban Hydrology for Small Watersheds*, Natural Resources Conservation Service, Washington, DC. <http://www.wcc.nrcs.usda.gov/water/quality/common/tr55/tr55.pdf> (accessed June 2004).
- Strafaci, A.M., 1998, *Essential Hydraulics and Hydrology*, Haestad Press, Waterbury, CT.
- Streeter, V.L. and E.B. Wylie, 1985, *Fluid Mechanics*, McGraw-Hill, New York, NY.
- Tchobanoglous, G., 1981, *Wastewater Engineering: Collection and Pumping of Wastewater*, Metcalf & Eddy, Inc. McGraw-Hill, Inc.
- Van Havern, B.P., 1986, *Water Resources Measurements*, American Water Works Association, Denver, CO.



Bibliography

- Wanielista, M.P., 1990, *Hydrology and Water Quantity Control*, John Wiley & Sons, New York, NY.
- Walski, T.M., 1984, *Analysis of Water Distribution Systems*, Van Nostrand-Reinhold, New York, NY.
- Walski, T.M., 1993, *Water System Modeling Using CYBERNET®*, Haestad Methods, Waterbury, CT.
- Williams, G.B. 1922. Flood Discharges and the Dimensions of Spillways in India. *Engineering (London)* 134, 321.
- Wylie, E.B. and V.L. Streeter, 1993, *Fluid Transients in Systems*, Prentice-Hall.
- Zipparro, V.J. and H. Hasen, 1993, *Davis' Handbook of Applied Hydraulics*, McGraw-Hill, New York, NY.



Bentley Systems Software

A.1 Software Products

Purchasing this book entitles the reader to use student versions of several Bentley OpenFlows products. These provide all software features available in the professional version. However, the licenses for programs incorporating networked hydraulic systems (i.e., StormCAD, WaterGEMS, HAMMER, SewerGEMS, SewerCAD, and PondPack) limit the number of network elements that may be included in a single project. Further, programs with academic licenses may be used only for academic purposes such as completing the exercises in this book. Professional or commercial use is prohibited.

Table A 1 describes the capabilities of the included licenses associated with the book, as well as the additional options that are available with the purchase of a professional license or upgraded academic license. (Academic license upgrades are available only to academic institutions.) For more information on these packages, visit www.bentley.com.

The software provided is intended to be used with the standalone version of the product. If the user wants to use the AutoCAD or ArcGIS platform versions, it is the user's responsibility to install and integrate those products.

A.2 Software Installation and Licensing

Readers of this book can access the OpenFlows software several ways:

1. If they own commercial versions of the software through their employment or other sources, those versions can be used, and there is no need to install student versions.
2. If their school owns academic licenses of the software, there is no need to install additional software.
3. If they don't have access to either source, they can install and license their software using the steps described in the "Downloading and Licensing" in the article: aka.bentley.com/CAiHE.supplement.

There is a need to supply a unique registration code to install the software, and obtaining the code varies depending on whether a paper or eBook is being used.

- a. Paper book—the unique registration code is included on an inserted sheet in the book. If you do not have the insert, please contact bipress@bentley.com
- b. eBook—contact bipress@bentley.com for a registration code

With this code, it is possible to download a license file and install the software according to the above article.

Table A.1: Software access provided with this book

Program	Description	Academic License	Professional License Options
FlowMaster	Hydraulic solver for pressure pipes, open channels, weirs, orifices and inlet structures	All features present	Same features as academic version but licensed for professional use
Culvert Master	Design and analysis tool for culvert hydraulics	All features present	Same features as academic version but licensed for professional use
StormCAD	Storm sewer design and modeling package	10 inlets; Stand-Alone interface	Additional inlets; AutoCAD integration
WaterGEMS	Water distribution design and modeling package	25 pipe segments; Stand-Alone, AutoCAD, ArcGIS, and MicroStation interfaces	Additional pipe segments
SewerCAD	Sanitary sewer design and modeling package	25 pipe segments; Stand-Alone interface	Additional pipe segments; AutoCAD integration
PondPack	Stormwater detention design package	limited to 1 pond	Additional detention basins; interconnected pond modeling
Sewer GEMS	Dynamic sanitary sewer and stormwater analysis	25 links; Stand- Alone, AutoCAD, ArcGIS, and Micro-Station interfaces	Additional link segments
HAMMER	Hydraulic transient analysis software	5 pipe segments; Stand-Alone, AutoCAD, ArcGIS and MicroStation interfaces	Additional pipe segments

Installation for the MicroStation, AutoCAD, or ArcGIS Platform

It is recommended that beginners use the stand-alone versions of the products. Some of the products are available with MicroStation, AutoCAD, and/or ArcGIS integration. (The user must already have one of these programs to use the integrated version. Currently AutoCAD integration is available only with WaterGEMS, SewerGEMS and HAMMER).

After completing the installation instructions previously described, you should see shortcuts for applicable integrated versions on your Desktop and in the product's Start menu folder. For example "WaterGEMS for AutoCAD 2020". If you do not see the shortcut, check the Readme.pdf file included with the installation to check if you have a compatible version of the integrated application. If the integrated application (such as AutoCAD) was installed after the Bentley software, you will need to run the Integrator tool, located under **Start Menu**

A.3. General Tips and Common Tools

Bentley OpenFlows' software programs share many common features and interface tools. This section summarizes the basic characteristics of some of these shared features. You are encouraged to "play" with the software to become more familiar with them. These features are quite intuitive and can help to increase the efficiency of your water resources modeling activities.

Online Help

Electronic documentation can be kept up-to-date more easily than paper documentation. It is accessible from within the program, and it is environmentally friendly. For these reasons, Bentley OpenFlows' programs have extensive online Help systems to guide the user through the analysis and design process. The Help system is used most often to:

- Obtain context-sensitive information about an area of the program
- Search for topics such as background theory and equations
- Use "How Do I" to get detailed step-by-step instructions for completing a task

Context-Sensitive Help

The quickest way to access the Help topic related to your current task (i.e., **context-sensitive help**) is to press the **F1** key, click a **Help** button, or right-click the mouse and select **Help** from the pop-up menu. Context-sensitive help provides instant access to the help topics related to the current dialog.

How Do I?

The **How Do I?** section of the Help system has step-by-step instructions for common applications of the model. You may find it beneficial to read over these topics before using the software or at least to familiarize yourself with them. When a related question arises, you can quickly find the answer.

Pop-ups and Jumps

Certain words within many Help topics are underlined and displayed in a different color. These words are known as *pop-ups* and *jumps*. When you select a pop-up (dotted underline), a definition of the word or phrase appears on the screen. To close the pop-up window, click anywhere outside the pop-up dialog. When you select a *jump* (solid underline), a related Help topic is displayed.

Graphical Editor

Network models such as StormCAD, WaterGEMS, SewerCAD, and PondPack have a graphical editor to aid the engineer in laying out the system. The Bentley OpenFlows graphical editor is an “intelligent” drafting environment, meaning that it recognizes the characteristics and behavior of different modeling elements and maintains connectivity when the user drags and drops elements on-screen.

This intelligent graphical editor prevents problems such as incorrect pipe connections and “hanging” pipes (pipes with a node defined at only one end). Models that do not have this type of behavior can be easily “broken,” and it may take hours (or days) to locate the source of the problem.

DXF Files

A drawing exchange file (DXF) is the standard format used for translating from one CAD system, such as AutoCAD®, to another, such as MicroStation™. DXF files can also be imported into StormCAD, WaterGEMS, and SewerCAD for use as background drawings. For example, a roadway network plan can be imported and used as a background map for laying out and designing a proposed storm sewer system, sanitary sewer system, or water distribution network.

If desired, drawing elements from DXF files can be automatically converted to network components using the polyline-to-pipe import feature. DXF lines and polylines can be converted to pipes, and DXF blocks can be converted to node elements.

To learn more about importing and using DXF backgrounds, look in the online help system of StormCAD, WaterGEMS, or SewerCAD. You can find topics in the **How Do I?** section or perform a search with a keyword such as “DXF” or “import.”

Scaled or Schematic?

In StormCAD, WaterGEMS, and SewerCAD, you may choose to either create a schematic drawing or define the horizontal and vertical distance scale. The scaled mode is typically used when a DXF or CAD background file is present. The scale is set so that one drawing unit is equal to a unit of measure (e.g., 1 DXF unit equals 1 foot). In a scaled plan, you can also set the drawing scale that you want to use when printing or plotting.

A schematic plan is not drawn at any defined scale, so there is no connection between dimensions shown on the plan and real-world dimensions. There may not even be a correlation between individual dimensions shown on a schematic plan. For example, in a schematic plan, the length representing a pipe with a length of 50 m may be less than, equal to, or greater than, the line representing a pipe with a length of 20 m.

To get to the drawing scale options, choose **Options** from the **Tools** menu and go to the **Drawing** tab. The available options are as follows:

- **Schematic**—You enter the pipe length in the **Pipe Properties** dialog for each pipe. Absolute positioning in the drawing editor is not used.
- **Scaled**—You enter the drawing unit scale to be used to determine the length of the drawn pipes. The pipe length will be calculated based on the starting and ending coordinates. The length in the drawing editor corresponds to a real-world length. You can also enter the horizontal and vertical drawing scales at which output should be plotted.

Color Coding

One way to determine the performance of a system is to color code the system according to some indicator. For hydraulic design work, pipeline velocity is often a useful indicator. For example, pipes consistently flowing below 0.15 m/s may be oversized. Pipes with velocities over 1.5 m/s are fairly heavily stressed, and those above 2.4 m/s are often bottlenecks in the system.

Table Manager and Table Customization

StormCAD, WaterGEMS, SewerCAD, and FlowMaster have tabular data sheets called FlexTables. These tables aid the modeler in entering and editing data, as well as in organizing and presenting the results of the model. FlexTables have several built-in features that are discussed in detail in the online help system. For information about customizing FlexTables, search the online help for topics such as “table” or “customize.” The following are a few of the topics related to table preferences.

- **Changing the Table Title**—When you choose to print a table, the table name is used as the title in the printed report. You can change the report title by using the **Table Manager** to rename the table.

- **Adding and Removing Columns**—You can add and remove column headings by using the table setup options from the table manager.
- **Drag-and-Drop Column and Row Placement**—Select the column or row that you would like to move by holding down the left mouse button on its heading and simply dragging it to the desired location.
- **Resizing Columns**—In the column heading section of the table, place your mouse over the vertical separator between columns. The cursor shape changes to indicate that you are over the separator. Hold down the left mouse button and drag the mouse to the left or right to set the new column width.
- **Changing Column Properties**—The display units for various data types can be changed in the tables. To view or edit the properties of any numeric column, right-click the heading area of the column and choose **Properties** from the pop-up menu. The unit properties will be displayed in the **Set Field Options** dialog.
- **Changing Column Labels**—To change the label of any column, right-click the column heading and choose **Edit Column Label** from the context menu.
- **Using Local Units**—Local units allow data columns to have fixed units and display precision, regardless of the current unit settings of the model. This option can be used to create standardized reports or to present the same data side by side in different units (especially handy for projects that use both English and SI units).
- In addition to changing the previous table properties, you can also perform data management operations such as sorting, filtering, and global editing.
- **Sorting**—A table can be sorted in ascending or descending order according to any variable. Right-click on the column heading, select **Sort**, and then select **Ascending** or **Descending**. In this way, the elements can be quickly reorganized based on the alphanumeric labels, pipe flow rates, pipe diameters, hydraulic grade lines, and so on.
- **Filtering**—Occasions often arise when a modeler wants to view only a subset of the entire system. For example, the modeler may be concerned with all pipes carrying more than 3 m³/s in a storm or sanitary sewer, or there may be problems with junction nodes having a pressure below 150 kPa in a water distribution network. The ability to show only some of a system's elements is called *filtering*, and it is accomplished by specifying the desired filter criteria in the table's **Filter** dialog box. Each element in the filter list requires three items:
- **Column**—The table column containing the element property by which to filter.

- **Operator**—The operator used to compare the specified filter value to the values in the specified column. Operators include: =, >, ≥, <, ≤, and ≠.
- **Value**—The comparison value for the filter.

Any number of criteria can be added to a filter. Multiple criteria are implicitly joined with a logical AND statement. So, when multiple criteria are defined, only rows that meet all of the specified criteria will be displayed.

The status pane at the bottom of the **Tabular** window shows the number of elements displayed and the total number of rows available (e.g., 10 of 20 elements displayed). When a filter is active, the element labels will turn blue. Table filtering allows you to perform global editing (see next bullet) on a subset of elements that you want to change.

- **Global Editing**—Factors and constants can be applied to every value of any editable variable within a FlexTable. Global editing operations include multiplication, division, addition, subtraction, or directly setting all values, depending on the modeler's needs. Only the elements displayed in the table will be changed, so global editing is often used in conjunction with filtering to obtain the desired results. To access global editing, right-click the column heading of the column that you want to edit and select the **Global Edit** option from the pop-up menu.

FlexUnits

You can set field options—the properties of a numeric value—from almost any field in any of Bentley OpenFlows' programs. These properties include the units, decimal precision, scientific notation display option, and the allowable range of values. This feature is referred to as **FlexUnits**.

To set these options for a particular field type, simply right-click the field and select **Properties** from the context menu. This action opens the **Set Field Properties** dialog where you can select the options that you want.

If you would like to set up the units for many data types from the same location in the program, choose **FlexUnits** from the **Tools** pull-down menu to open the FlexUnits dialog.

Units

FlexUnit offers a wide variety of possible units for any field. In the **Set Field Options** dialog, click the **Units** drop-down menu and select the desired unit from the list. The value in the field will be automatically converted to the new unit using the appropriate conversion factor. The FlexUnit feature allows you to mix and match any units, even if you are using a combination of English and SI units.

Once you begin changing units, you will probably notice that some units have multiple representations (e.g., psi and lbs/in², cfs and ft³/s). Computationally,

these units are the same; however, the additional choices allow you to control the way the units are presented on-screen and in program output.

Display Precision

Display precision can be used to control the number of digits displayed after a decimal point or the rounding of a numeric value. When the precision is greater than or equal to zero, it specifies the number of digits that are displayed after the decimal point. For example, π (3.14159265...) with a decimal precision of 4 would be presented as 3.1416.

Specifying a negative number for display precision results in a displayed value that is rounded to the nearest power of 10. -1 rounds to 10, -2 rounds to 100, -3 rounds to 1000, and so on. For example, the number 1,234 with a display precision of -2 would be displayed as 1,200.

Note: The display precision affects only the way the numbers are displayed, not their actual values. The internal values stored by the software are still carried out to their maximum decimal precision—they are just displayed differently. This concept is important concept to keep in mind, especially when checking calculations by hand or working with values that are lower than the usual decimal precision. For example, be aware that a 0.75-in diameter pipe with a display precision of 0 will result in a displayed value of 1 in, even though the calculations will be performed based on the true 0.75-in diameter.

Scientific Notation

Scientific notation displays a value as a real number multiplied by some power of 10. It is displayed as an integer or real value followed by the letter “e” and a positive or negative integer.

For example, 12,345 could be written in scientific notation as 1.2345 e^4 , and 0.12345 could be written as 1.2345 e^{-1} .

Scientific notation follows the same display precision rules outlined previously. To turn scientific notation on or off, just check or uncheck the box labeled **Scientific notation**.

Minimum and Maximum Allowable Values

These options are available for selected input data and control the range of input values that the program will accept. For example, the user may want to set minimum or maximum values for roughness coefficients, slope, and so forth in order to limit these values to a realistic range, thus helping to prevent typographic errors during data entry.

B

Scenario Management

B.1 Overview

The Scenario Manager is included in many of the Bentley Systems software. This feature can dramatically increase your productivity in the “What If?” areas of modeling, including calibration, operations analysis, and planning. By investing a little time to understand scenario management, you can avoid unnecessary editing and data duplication. Tutorial 2 with Chapter 3 demonstrates how to manage and compare scenarios using SewerCAD and SewerGEMS, which can also be used for WaterGEMS, HAMMER, and PondPack.

In contrast to the old methods of scenario management (editing or copying data), automated scenario management using inheritance gives you these advantages:

- A single project file makes it possible to generate an unlimited number of “What If?” conditions without becoming overwhelmed with numerous modeling files and separate results.
- Because the software maintains the data for all the scenarios in a single project, it can provide you with powerful automated tools for directly comparing scenario results. Any set of results is immediately available at any time.
- The Scenario/Alternative relationship empowers you to mix and match groups of data from existing scenarios without having to declare any data again.
- With inheritance, you do not have to re-enter data if it remains unchanged in a new alternative or scenario, avoiding redundant copies of the same data. Inheritance also enables you to correct a data input error in a parent scenario and automatically update the corrected attribute in all child scenarios.

These advantages, while obvious, may not seem compelling for small projects. It is as projects grow to hundreds or thousands of network elements that the advantages of true scenario inheritance become clear. On a large project, being able to maintain a collection of base and modified alternatives accurately and

efficiently can be the difference between evaluating optional improvements and being forced to ignore them.

B.2 About This Appendix

The depth of scenario management as implemented by Bentley Systems is probably far beyond what you have ever seen before. With that in mind, this appendix is intended as an introduction to the philosophy and terminology upon which scenario management is based.

This appendix is not intended as a step-by-step guide to using the software. If you are a moderately experienced Windows software user, you should have no difficulty learning and exploring the scenario management interface.

In addition to the tutorials with this book, excellent tutorials and context-sensitive online help are also available within the software itself. These learning tools will help you with all aspects of the software, and should certainly not be ignored if you are having difficulty. For more information, just click the **Help** button, which is available from anywhere within the program. In addition, you can contact Bentley Systems or visit www.bentley.com for the schedule of training seminars held around the world.

B.3 Developing Scenarios

Let us begin by understanding the approaches that have historically been used to attempt “What If?” analyses. Traditionally, there have only been two possible ways of analyzing the effects of change on a software model:

- Change the model, recalculate, and review the results
- Create a copy of the model, edit that copy, calculate, and review the results

Although either of these methods may be adequate for a relatively small system, the data duplication, editing, and re-editing becomes very time-consuming and error-prone as the size of the system — and the number of possible conditions — increase. Additionally, comparing conditions requires manual data manipulation, because all output must be stored in physically separate data files.

The scenario-management feature developed by Bentley Systems logically organizes scenarios to meet the following objectives:

- Minimize the number of project files the modeler needs to maintain (one, ideally).
- Maximize the usefulness of scenarios through easy access to things such as input and output data, and direct comparisons.

- Maximize the number of scenarios you can simulate by mixing and matching data from existing scenarios (data reuse).
- Minimize the amount of data that needs to be duplicated to consider conditions that have a lot in common.

A single project file enables you to generate an unlimited number of “What If?” conditions, edit only the data that needs to be changed, and quickly generate direct comparisons of input and results for desired scenarios.

B.4 The Scenario Cycle

The process of working with scenarios is similar to the process of manually copying and editing data, but without the disadvantages of data duplication and troublesome file management. This process allows you to cycle through any number of changes to the model, without fear of overwriting critical data or duplicating important information (see Figure B.1). Of course, it is possible to directly change data for any scenario, but an “audit trail” of scenarios can be useful for retracing the steps of a calibration series or for understanding a group of master plan updates.

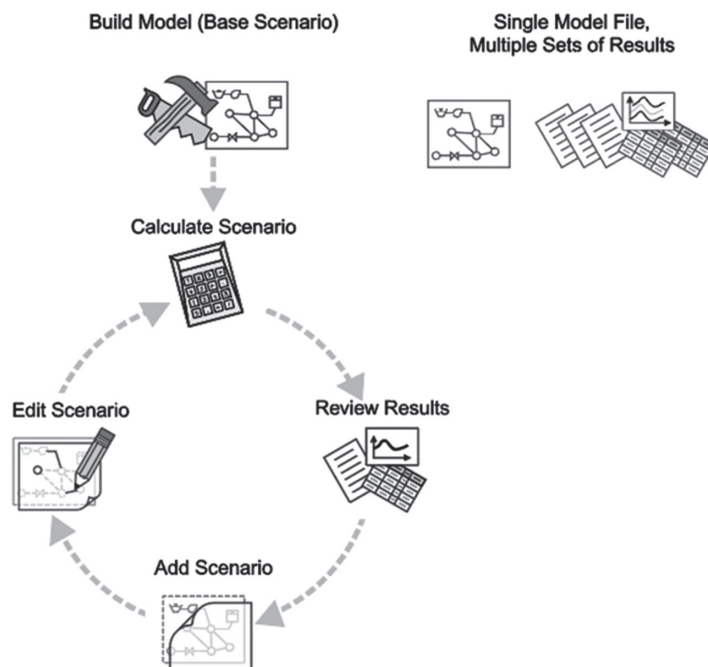


Figure B.1: Self-Contained Scenarios

Scenario Anatomy: Attributes and Alternatives

Before exploring scenario management further, a few key terms should be defined:

- An **attribute** is a fundamental property of an object, often a single numeric quantity. For example, the attributes of a pipe include diameter, length, and roughness.
- An **alternative** holds a family of related attributes so pieces of data that you are most likely to change together are grouped for easy referencing and editing. For example, a physical properties alternative groups physical data for the network's elements, such as elevations, sizes, and roughness coefficients.
- A **scenario** has a list of referenced alternatives (which hold the attributes), and combines these alternatives to form an overall set of system conditions that can be analyzed. This referencing of alternatives enables you to easily generate system conditions that mix and match groups of data that have been previously created. Note that scenarios do not actually hold any attribute data — the referenced alternatives do.

A Familiar Parallel

Although the structure of scenarios may seem a bit difficult at first, anyone who has eaten at a restaurant should be able to relate fairly easily. A meal (scenario) is comprised of several courses (alternatives), which might include a salad, an entrée, and a dessert. Each course has its own attributes. For example, the entrée may have a meat, a vegetable, and a starch. Examining the choices, we could present a menu as in Figure B.2.

The restaurant does not have to create a new recipe for every possible meal (combination of courses) that could be ordered. They can just assemble any meal based on what the customer orders for each alternative course. Salad 1, Entrée 1, and Dessert 2 might then be combined to define a complete meal.

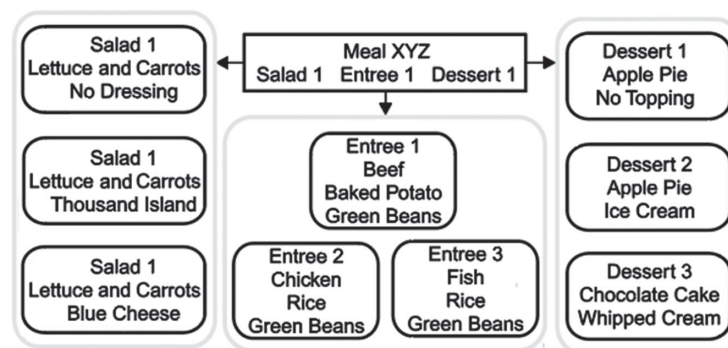


Figure B.2: A Restaurant Meal “Scenario”

Generalizing this concept, we see in Figure B.3 that any scenario simply references one alternative from each category to create a “big picture” that can be analyzed. Note that different types of alternatives may have different numbers and types of attributes, and any category can have an unlimited number of alternatives to choose from.

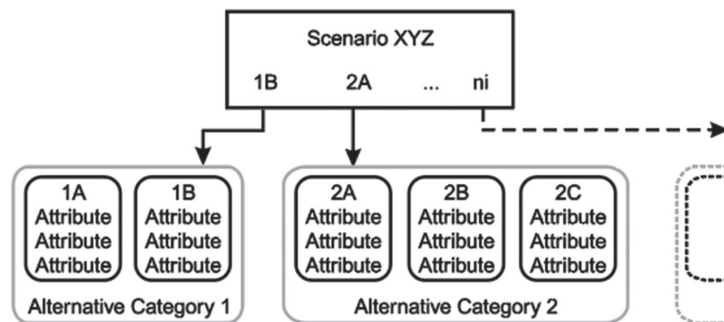


Figure B.3: Generic Scenario Anatomy

B.5 Scenario Behavior: Inheritance

The separation of scenarios into distinct alternatives (groups of data) meets one of the basic goals of scenario management: maximizing the number of scenarios you can develop by mixing and matching existing alternatives. Two other primary goals have also been addressed: a single project file is used, and easy access to input data and calculated results is provided in numerous formats through the intuitive graphical interface.

But what about the other objective: minimizing the amount of data that needs to be duplicated to consider conditions that have a lot of common input? Surely an entire set of pipe diameters should not be re-entered if only one or two change?

The solution is a familiar concept to most people: *inheritance*.

In the natural world, a child inherits characteristics from a parent. This may include such traits as eye color, hair color, and bone structure. There are two significant differences between the genetic inheritance that most of us know and the way inheritance is implemented in software:

- Overriding inheritance
- Dynamic inheritance

Overriding Inheritance

Overriding inheritance is the software equivalent of cosmetics. A child can override inherited characteristics at any time by specifying a new value for that

characteristic. These overriding values do not affect the parent, and are therefore considered “local” to the child. Local values can also be removed at any time, reverting the characteristic to its inherited state. The child has no choice in the value of his inherited attributes, only in local attributes.

For example, suppose a child has inherited the attribute of blue eyes from his parent. Now the child puts on a pair of greentinted contact lenses to hide his natural eye color. When the contact lenses are on, we say his natural eye color is “overridden” locally, and his eye color is green. When the child removes the tinted lenses, his eye color instantly reverts to blue, as inherited from his parent.

Dynamic Inheritance

Dynamic inheritance does not have a parallel in the genetic world. When a parent’s characteristic is changed, existing children also reflect the change. Using the eye-color example, this would be the equivalent of the parent changing eye color from blue to brown, and the children’s eyes instantly inheriting the brown color also. Of course, if the child has already overridden a characteristic locally, as with the green lenses, his eyes will remain green until the lenses are removed. At this point, his eye color will revert to the inherited color, now brown.

This dynamic inheritance has remarkable benefits for applying wide-scale changes to a model, fixing an error, and so on. If rippling changes are *not* desired, the child can override all of the parent’s values, or a copy of the parent can be made instead of a child.

When Are Values Local, and When Are They Inherited?

Any changes that are made to the model belong to the currently active scenario and the alternatives that it references. If the alternatives happen to have children, those children will also inherit the changes unless they have specifically overridden that attribute. Figure B.4 demonstrates the effects of a change to a mid-level alternative. Inherited values are shown as gray text, local values are shown as black text.



Figure B.4: A Mid-level Hierarchy Alternative Change

Minimizing Effort through Attribute Inheritance

About Scenario Management

Inheritance has an application every time you hear the phrase “just like x except for y ”. Rather than specifying all of the data from x again to form this new condition, we can simply create a child from x and change y appropriately. Now we have both conditions, with no duplicated effort.

We can even apply this inheritance to our restaurant analogy as follows. Inherited values are shown as gray text, local values are shown as black text.

Salad Alternative Hierarchy	Attribute: Vegetables	Attribute: Dressing
Salad 1	Lettuce and Carrots	No Dressing
└─ Salad 2	Lettuce and Carrots	Thousand Island
└─ Salad 3	Lettuce and Carrots	Blue Cheese

- “Salad 2 is just like Salad 1, except for the dressing.”
- “Salad 3 is just like Salad 1, except for the dressing.”

Note: Salad 3 could inherit from Salad 2, if we prefer: “Salad 3 is just like Salad 2, except for the dressing.”

Meal Scenario Hierarchy	Attribute: Meat	Attribute: Starch	Attribute: Vegetable
Entrée 1	Beef	Baked Potato	Green Beans
└─ Entrée 2	Chicken	Rice	Green Beans
└─ Entrée 3	Fish	Rice	Green Beans

- “Entrée 2 is just like Entrée 1, except for the meat and the starch.”
- “Entrée 3 is just like Entrée 2, except for the meat.”

Note: If the vegetable of the day changes (say from green beans to peas), only Entrée 1 needs to be updated, and the other entrées will automatically inherit the vegetable attribute of “Peas” instead of “Green Beans”.

Dessert Alternative Hierarchy	Attribute: Bakery Item	Attribute: Topping
Dessert 1	Apple Pie	No Topping
└─ Dessert 2	Apple Pie	Ice Cream
Dessert 3	Chocolate Cake	Whipped Cream

- “Dessert 2 is just like Dessert 1, except for the topping.”

Note: Dessert 3 is a parent alternative and has nothing in common with the other desserts, so it can be created as a “root” or “base” alternative. It does not inherit its attribute data from any other alternative.

Minimizing Effort through Scenario Inheritance

Just as a child alternative can inherit attributes from its parent, a child scenario can inherit which alternatives it references from its parent. This is essentially still the phrase “just like x except for y ”, but on a larger scale.

Carrying through on the meal example, consider a situation where you go out to dinner with three friends. The first friend places his order, and the second friend orders the same thing except for the dessert. The third friend orders something totally different, and you order the same meal as hers except for the salad.

The four meal “scenarios” could then be presented as follows (inherited values are shown as gray text, local values are shown as black text):

Meal Scenario Hierarchy	Salad Alternative	Entree Alternative	Dessert Alternative
Meal 1	Salad 1	Entree 2	Dessert 3
└ Meal 2	Salad 1	Entree 2	Dessert 1
Meal 3	Salad 3	Entree 3	Dessert 2
└ Meal 2	Salad 2	Entree 3	Dessert 2

- “Meal 2 is just like Meal 1, except for the dessert.” The salad and entrée alternatives are inherited from Meal 1.
- “Meal 3 is nothing like Meal 1 or Meal 2.” A totally new “base” or “root” is created.
- “Meal 4 is just like Meal 3, except for the salad.” The entrée and dessert alternatives are inherited from Meal 3.

B.6 A Water Distribution Example

This section presents a fairly simple water distribution system: a single reservoir supplies water by gravity to three junction nodes (see Figure B.5).

Although true water distribution scenarios include such alternative categories as initial settings, operational controls, water quality, and fire flow, we are going to focus on the two most commonly changed sets of alternatives: demands and physical properties. Within these alternatives, we are going to concentrate on junction baseline demands and pipe diameters.

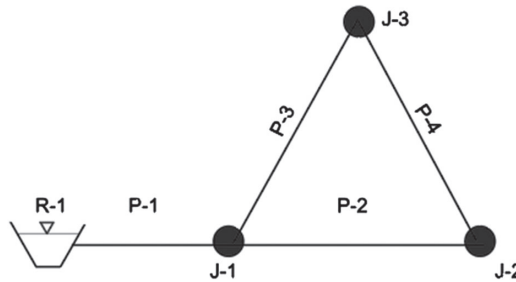


Figure B.5: Example Water Distribution System

Building the Model (Average Day Conditions)

During model construction, probably only one alternative from each category is going to be considered. This model is built with average demand calculations and preliminary pipe diameter estimates. At this point we can name our scenario and alternatives, and the hierarchies look like the following (showing only the items of interest):

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	500 gpm	100 gpm

Physical Alternative Hierarchy	P-1	P-2	P-3
Preliminary Pipes	8 inches	6 inches	6 inches

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes

Analyzing Different Demands (Maximum Day Conditions)

In our example, the local planning board also requires analysis of maximum day demands, so a new demand alternative is required. No variation in demand is expected at J-2, which is an industrial site. As a result, the new demand alternative can inherit J-2's demand from "Average Day" while the other two demands are overridden.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	500 gpm	100 gpm
Maximum Day	200 gpm	500 gpm	200 gpm

Now we can create a child scenario from “Average Day” that inherits the physical alternative, but overrides the selected demand alternative. As a result, we get the following scenario hierarchy:

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
└─Max. Day	Maximum Day	Preliminary Pipes

Because no physical data (pipe diameters) have been changed, the physical alternative hierarchy remains the same as before.

Another Set of Demands (Peak Hour Conditions)

Based on pressure requirements, the system is adequate to supply maximum day demands. Another local regulation requires analysis of peak hour demands, with slightly lower allowable pressures. Because the peak hour demands also share the industrial load from the “Average Day” condition, “Peak Hour” can be inherited from “Average Day.” In this instance, “Peak Hour” could inherit just as easily from “Maximum Day.”

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	500 gpm	100 gpm
└─Maximum Day	200 gpm	500 gpm	200 gpm
└─Peak Hour	250 gpm	500 gpm	250 gpm

Another scenario is also created to reference these new demands, as shown here:

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
└─Max. Day	Maximum Day	Preliminary Pipes
└─Peak	Peak Hour	Preliminary Pipes

Note again that the physical data was not changed, so the physical alternatives remain the same.

Correcting an Error

This analysis results in acceptable pressures, until it is discovered that the industrial demand is not actually 500 gpm — it is 1,500 gpm! Because of the inheritance within the demand alternatives, however, only the “Average Day”

Scenario Management

A Water Distribution Example

demand for J-2 needs to be updated. The changes will ripple through to the children. After the single change is made, the demand hierarchy is as follows:

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	1500 gpm	100 gpm
└─Maximum Day	200 gpm	1500 gpm	200 gpm
└─Peak Hour	250 gpm	1500 gpm	250 gpm

Notice that no changes need to be made to the scenarios to reflect these corrections. The three scenarios can now be calculated as a batch to update the results.

When these results are reviewed, it is determined that the system does *not* have the ability to adequately supply the system as it was originally thought. The pressure at J-2 is too low under peak hour demand conditions.

Analyzing Improvement Suggestions

To counter the head loss from the increased demand load, two possible improvements are suggested:

- A much larger diameter is proposed for P-1 (the pipe from the reservoir). This physical alternative is created as a child of the “Preliminary Pipes” alternative, inheriting all the diameters except P-1’s, which is overridden.
- Slightly larger diameters are proposed for all pipes. Because there are no commonalities between this recommendation and either of the other physical alternatives, this can be created as a base (root) alternative.

These changes are then incorporated to arrive at the following hierarchies:

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	1500 gpm	100 gpm
└─Maximum Day	200 gpm	1500 gpm	200 gpm
└─Peak Hour	250 gpm	1500 gpm	250 gpm

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	8 inches	6 inches	6 inches	6 inches
└─Larger P-1	18 inches	6 inches	6 inches	6 inches
Larger All Pipes	12 inches	12 inches	12 inches	12 inches

This time, the demand alternative hierarchy remains the same since no demands were changed. The two new scenarios (“Peak, Big P-1,” “Peak, All Big Pipes”) can be batch run to provide results for these proposed improvements.

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
Max. Day	Maximum Day	Preliminary Pipes
Peak	Peak Hour	Preliminary Pipes
Peak, Big P-1	Peak Hour	Larger P-1
Peak, All Big Pipes	Peak Hour	Larger All Pipes

Now features like Scenario Comparison Annotation (from the Scenario Manager) and comparison Graphs (for extended period simulations, from the element editor dialogs) can be used to directly determine which proposal results in the most improved pressures.

Finalizing the Project

It is decided that enlarging P-1 is the optimum solution, so new scenarios are created to check the results for average day and maximum day demands. Notice that this step does not require handling any new data. All of the information we want to model is present in the alternatives we already have!

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
Max. Day	Maximum Day	Preliminary Pipes
Max. Day, Big P-1	Maximum Day	Larger P-1
Peak	Peak Hour	Preliminary Pipes
Peak, Big P-1	Peak Hour	Larger P-1
Peak, All Big Pipes	Peak Hour	Larger All Pipes
Avg. Day, Big P-1	Average Day	Larger P-1

Also note that it would be equally effective in this case to inherit the “Avg. Day, Big P-1” scenario from “Avg. Day” (changing the physical alternative) or to inherit from “Peak, Big P-1” (changing the demand alternative). Likewise, “Max. Day, Big P-1” could inherit from either “Max. Day” or “Peak, Big P-1.”

Neither the demand nor physical alternative hierarchies were changed in order to run the last set of scenarios, so they remain as they were.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	1500 gpm	100 gpm
Maximum Day	200 gpm	1500 gpm	200 gpm
Peak Hour	250 gpm	1500 gpm	250 gpm

Scenario Management

A Water Distribution Example

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	8 inches	6 inches	6 inches	6 inches
└─ Larger P-1	18 inches	6 inches	6 inches	6 inches
Larger All Pipes	12 inches	12 inches	12 inches	12 inches

Index

A

Aboveground pond, 129
Abstractions, 39, 40
 depression storage, 39
 infiltration, 39–40
 initial rainfall, 45
 interception, 39
Adverse slope, 93
Affinity laws, 167
Air chamber. *See*
 Hydropneumatic tank
Air inlet valves, 205
Allowable spread length, 81
Antecedent moisture condition (AMC
 condition), 48
Area, 2–3
Attenuation, 130
Automated optimization, 176
 model calibration, 176–177
 system design, 177–178
Average-end-area method, 134–136

B

Babbitt method, 217
Backwater analysis, 93
Base flow, 65
Best management strategies
 (BMPs), 237
Broad-crested weir, 20
Burst diaphragm. *See* Rupture disk
Bursting disk. *See* Rupture disk

C

C coefficients, 57
Celerity, 242
Centrifugal pump, 167
Change in energy (ΔE), 6
Characteristic curve, 167
Check valves (CVs), 169
Chézy's equation, 10

Colebrook–White equation. *See*
 Darcy–Weisbach equation
Combination inlets on grade, 87–88
Combined sewers, 211
Composite stage *vs.* discharge
 relationships, 142
 structures in parallel, 142–145
 structures in series, 145
Computer applications, 23–24
Conic method, 137–139
Conservation
 of energy, 171
 of mass, 170
 of mass equation, 174
Constant horsepower pumps, 168
Continuity equation, 198
Continuous pattern, 163, 214
Contracted weir, 20
Contributing area, 215
Control valves, 169
Conventional method, 164
Convex routing method, 220–221
Cover considerations, 42–43
Crest, 19
Critical depth, 15–17, 98
Critical slope, 92
Crown elevation, 98
Culvert hydraulics, 113
 CulvertMaster and tutorials,
 122–123
 culvert systems, 113–114
 inlet control hydraulics, 114,
 118–122
 outlet control hydraulics, 114–116
CulvertMaster and tutorials,
 122–123
Culvert outlets, 141
Culvert systems, 113–114
Curb inlets on grade, 85–87
Curve number
 method, 45
 tables, 48–51

D

Dam embankments, 132
Darcy–Weisbach equation, 11–12,
 13, 24, 164
Darwin Designer, 177–178
Demand patterns, 162–163
Demands, 170
Depression storage, 39
Design storm hyetograph from SCS
 distributions, 35–38
Detention basins, 127
Detention pond design
 basic design considerations,
 127–130
 components of detention facilities,
 131–133
 modeling concepts, 130–131
 PondPack and tutorials, 155–156
 postdevelopment criteria, 128–129
 predevelopment criteria, 128–129
 recurrence frequency, 128–129
 storage and hydraulic relationships,
 133–145
 storage indication method,
 145–149
 stormwater detention analysis
 procedure, 127, 149–155
 types and configurations of
 stormwater detention facilities,
 129–130
Detention ponds. *See* Detention
 basins
Diffusion wave model, 243–244
Dimensionless unit hydrograph, 70
Direct-step method, 95
Direct runoff hydrograph, 65
Discrete convolution, 74–76
Discrete volume method (DVM), 175
Distributed parameter models, 243
Diurnal curve, 162, 214
Drawdown analysis, 93
Dry-weather loads. *See* Sanitary loads

E

Dry detention basins, 127
Dry pond, 129
Dynamic modeling, 236–237, 238
 initial and boundary conditions, 241–242
 numerical solution, 240–241
 problems, 248–260
 St. Venant equations, 239–240

E

Effective precipitation, 39
Effective rainfall hyetographs. *See* Runoff hyetographs
Elastic model, 198–199
Emergency spillways. *See* Overflow spillways
Employees, 215
Energy balance, 94–95
Energy equation, 7, 114
Energy gains, 8–9, 166–168
Energy grade, 8
Energy grade line (EGL), 4, 8
Energy losses, 8–9, 164
 friction losses, 164–165
 minor losses, 165–166
Energy principle, 6–9
Entrance minor loss, 116
Eulerian DVM, 175
Eulerian models, 174
Exit minor loss, 117–118
Extended-period simulations (EPS), 172, 220
 routing, 220–222
Extreme flow factors (EFF), 215–217

F

Federal Highway Administration (FHWA), 82
Federov method, 217
FlexTables, 24
Flow Control Valves (FCVs), 169
FlowMaster, 24
Flows, 170
 accumulation, 97
 area, 2
 characteristics, 5
 classification, 92–95
 conveyance, 3

 rapidly varied, 98
 rate, 1–2
 tracking, 173
 zone classification, 93
Free-surface flow. *See* Open-channel flow
Freeboard, 132
Frequency, 29–30
Friction losses, 9, 116, 164–165
 Chézy's equation, 10
 Darcy–Weisbach equation, 11–12
 Hazen–Williams equation, 10–11
 Manning's equation, 9–10
 typical roughness factors, 12
Friction slope, 8
Frontwater analysis, 93
Froude number, 15–16, 242
Full-flow conditions, 3

G

Gas vessel. *See* Hydropneumatic tank
Gradually varied flow, 91–92
 analysis, 118
 energy balance, 94–95
 flow classification, 92–94
Gutter, 81
 sections on grade, 82–83

H

HAMMER software, 200
 and tutorials, 207–208
Harmon method, 217
Hazen–Williams equation, 10–11, 13, 24, 164
Head gains, 7
Head losses, 7
Headwater depth (HW depth), 113–114
HEC-22 *Urban Drainage Design Manual*, 82
High-level outlets, 140
Horizontal slope, 93
Horton equation, 38, 66
Horton infiltration equation, 41–42
 infiltration rate, 43
 land use and cover considerations, 42–43
 runoff estimation, 43

Hydraulic(s), 211
 grade, 7–8
 jumps, 99
 model, 163
 radius, 2–3
 routing model, 243
 theory, 164
 time steps, 222
Hydraulic grade line (HGL), 4, 8, 164
Hydraulic transient, 195
 design strategies, 203–204
 general modeling equations, 199–200
 HAMMER and tutorials, 207–208
 minimizing transients in systems, 203
 modeling, 206–207
 problems, 208–210
 protection devices, 204–206
 speed and pressure of transients, 200–202
 theory, 198–199
 transient flow, 195–198
HYDRO-35, 32
Hydrograph(s), 64, 220
 discrete convolution, 74–76
 load, 213–214
 runoff hyetographs, 65–68, 76–78
 unit hydrographs, 68–73
Hydrograph routing, 133
 effects, 242–243
Hydrologic routing model, 243
Hydrologic time steps, 222
Hydrology, 29
 hydrographs, 64–76
 model, 97
 peak runoff flow rate, 51–63
 rainfall, 29–41
 unit hydrographs, 68–73
Hydropneumatic tank, 205

I

Infiltration, 39–40, 219
Inflow, 219
 hydrograph, 130
Initial rainfall abstractions, 45
Inlet control hydraulics, 114, 118–122

Inlet design overview, 81, 90–91
 inlet and gutter network problems
 using StormCAD, 91

Inlets in sag, 82, 88–89

Inlets on grade, 82, 84–85

combination, 87–88

curb, 85–87

Intensity–duration–frequency curves
 (IDF curves), 30–32, 98

Interception, 39

Interconnected pond modeling, 130

International System of Units (SI), 1

J

Jet, 17

K

Kinematic wave model, 244

Kutter's equation. *See* Chézy's
 equation

L

Lag, 130

Lagrangian TDM, 175

Laminar flow, 4–6

Land use, 42–43

Loading, 212

EFF, 215–217

hydrographs, 213–214

load types, 213

pattern loads, 214–215

sanitary unit loading, 215

variable PF, 217–219

Low-Impact Development (LID),
 236–238

Low-level outlets, 140

M

Manning's equation, 9–10, 24, 55

Mass conservation, 45

Metered demand, 161

Mild slope, 93

Minor losses, 165–166

Mixed flow profiles, 95–96

Model calibration, 176–177

Momentum equations, 198–199, 201

Muskingum–Cunge method, 246

Muskingum routing method,
 245–246

N

Nappe, 20

Navier–Stokes equations, 199

Network analysis, 171–172

Nonrevenue water and leaks,

163–164

Nonuniform flow, 4

Normal depth, 4, 98

NRCS curve number equation, 38

NRCS (SCS) curve number

method, 46–47

antecedent moisture condition,
 48

cover type, 48

curve number tables, 48–51

hydrologic condition, 48

soil group, 47

NRCS (SCS) peak flow

estimation, 60–63

NRCS synthetic unit

hydrographs, 70–73

O

Off-line pond, 129

Online pond, 129

Open-channel flow, 3, 14

critical flow, 14–17

specific energy, 14–17

Orifices, 17–19, 141

Oscillating pressure wave, 197

Outflow hydrograph, 130

Outlet control hydraulics, 114–116

Outlet structures, 132

Overflow spillways, 133, 141

P

Pattern loads, 214–215

Patterns, 162, 214

Peaking factor (PF), 216

variable, 217–219

Peak runoff flow rate, 51

NRCS (SCS) peak flow

estimation, 60–63

rational method, 56–60

time of concentration, 51–55

Pipe leakage, 163

Pipe networks, 170–171

Pipe outlets, 141

Pond bottoms, 132

PondPack

and tutorials, 155–156

for windows, 156

Precipitation, 40

Pressure-Reducing Valves (PRVs), 169

Pressure-Sustaining Valves

(PSVs), 169

Pressure Breaker Valves (PBVs), 169

Pressure flow, 3, 12, 14

Pressure of transients, 200–202

Pressure pipe friction losses, 12

Pressure piping systems, 161

automated optimization, 176–178

control valves, 169

energy gains, 166–168

energy losses, 164–166

network analysis, 171–172

pipe networks, 170–171

problems, 179–193

water demands, 161–164

WaterGEMS and tutorials,

178–179

water quality analysis, 172–176

Pressure wave celerity, 200

Prismatic channel, 2

Profile

classifications, 93–94

in StormCAD, 98–99

Proportional methods, 164

Pumps, 166

constant horsepower pumps, 168

variable-speed pumps, 167–168

R

Rainfall, 29

abstractions, 39, 40

characteristics, 29

data types, 30–38

Horton infiltration

equation, 41–43

hyetograph, 32

NRCS (SCS) curve number

method, 46–51

Rainfall (*continued*)
 return period and frequency, 29–30
 runoff coefficient, 44–45
 runoff volume, 38–41
 temporal distribution, 33
 watershed area, 38–39
 Rational method, 56–60
 Recurrence interval, 30
 Residents, 215
 Retention, 45
 pond, 129
 Return period, 29–30
 Reynolds number, 4–6
 Rigid-column model, 198
 Routing, 220, 242
 analysis, 130
 convex routing, 220–221
 of detention pond, 145–146
 hydrologic and hydraulic time steps, 222
 weighted translation routing, 221–222
 Runoff coefficient, 38, 44–45
 Runoff hydrograph, 51,
 65–68, 76–78
 Runoff volume, 38–43, 128
 Rupture disk, 205–206

S

Sanitary loads, 213
 hydrographs, 213
 Sanitary sewer design, 211–212
 elements, 212
 extended-period simulations, 220–222
 loading, 212–219
 problems, 224–233
 SewerCAD, 222–224
 Sanitary unit loading, 215
 unit sanitary loads from different sources, 216
 Sealing conditions, 96
 SewerCAD, 211, 222–224
 algorithm, 220–221
 SewerGEMS software, 98, 236,
 240–241, 244
 and tutorials, 246–280
 Sharp-crested weir, 20
 Side slopes, 132

Simplified routing models, 243
 diffusion wave model, 243–244
 kinematic wave model, 244
 Muskingum routing method, 245–246
 Skeletonization, 206–207
 Slope classification, 92–93
 Soil group, 47
 Specific energy, 14–15
 Speed of transients, 200–202
 Spread, 81
 Stage *vs.* discharge, 140
 orifices and weirs, 141
 overflow spillways, 141
 pipe or culvert outlets, 141
 rating table, 140
 Stage *vs.* storage volume, 133
 average-end-area method, 134–136
 conic method, 137–139
 Standard-step method, 95
 Steady-state network hydraulics, 171
 Steady flow, 3
 model, 236
 Steep slope, 93
 Stepwise pattern, 162–163, 214
 Storage and hydraulic relationships, 133
 composite stage *vs.* discharge relationships, 142–145
 stage *vs.* discharge, 140–141
 stage *vs.* storage volume, 133–139
 Storage indication curve, 149
 method, 145–149
 StormCAD, 91, 98–99
 analysis and design, 99
 profiles, 99–100
 Storm sewer analysis, 161
 Storm sewer applications, 96–97
 flow accumulation, 97
 hydrology model, 97
 sources of water, 97–98
 Stormwater detention, 127
 analysis procedure, 149–155
 facilities, 129–130
 Stormwater management, 237
 Streamlines, 4
 St. Venant equations, 239–240
 Subcritical flow, 15
 Submerged flow, 118, 120–122

Supercritical flow, 15
 Suppressed weir, 20
 Surge. *See* Hydraulic transients
 Surge relief valves, 205
 Surge tanks, 204–205
 Synthetic temporal distributions of rainfall, 33
 System curve, 167
 System design, 177–178
 System operating point, 167

T

10 States Standard for PE, 217–219
 Throttle Control Valves (TCVs), 169
 Time-driven method (TDM), 175
 Time of concentration, 51–55
 Time of concentration, 33
 Total hydrograph, 65
 Trace modeling, 173
 Transient flow, 195–198
 models, 199
 Transient protection devices, 198
 Transitional flow, 118
 Turbulent flow, 4–6
 Typical roughness coefficients, 13
 Typical roughness factors, 12

U

Uncontrolled outlets, 131
 Underground detention, 129
 Uniform emitter coefficient, 164
 Uniform flow, 4
 Uniform open-channel flow, 16–17
 Unit hydrographs, 68
 estimation methods, 64
 NRCS synthetic, 70–73
 visualizing and developing, 68–70
 Unsteady flow, 3
 model, 236–237
 Unsubmerged flow, 118–120
 Urbanization of land, 237
 User-specified tailwater, 98

V

V-notch weir coefficients of discharge, 21
 Variable-speed pumps, 167–168

Variable PF, 217–219
Velocity, 1–2
Vena contracta, 17, 19, 20
Volumetric flow rate, 1

W

“Water-hammer” effect, 24
Water demands, 161
 demand patterns, 162–163
 metered demand, 161
 nonrevenue water and leaks,
 163–164

Water Environment & Reuse
 Foundation, 237
WaterGEMS, 176–177, 206
 and tutorials, 178–179
Water hammer. *See* Hydraulic
 transients
Water quality analysis, 172
 age, 172–173
 constituents, 173–174
 DVM, 175
 initial conditions, 174
 numerical methods,
 174–175

TDM, 175–176
 trace, 172
Watershed area, 38–39, 57
Water sources, 97–98
Wave celerity, 15
Weighted runoff coefficient, 45
Weighted translation routing method,
 221–222
Weirs, 19–23, 141
Wet-weather loading, 213, 219
Wet detention basins, 127
Wet pond, 129
Wetted perimeter, 2–3

Written to meet the needs of both students and professionals, **Computer Applications in Hydraulic Engineering (CAiHE)** is the ideal text to work with design software in water resources engineering and hydraulic engineering courses, as well as a comprehensive learning tool for practicing engineers.

Linking hydraulic theory with real-world applications, **CAiHE** focuses on problem-solving skills and computer modeling of open channel and pressure pipe flow including culvert analysis and pumping systems, the design and analysis of storm and sanitary sewers, water distribution network design and analysis including transients and water quality, and stormwater management with Low Impact Development strategies.

Included in the text is how to access the academic license for working copies of Bentley System's OpenFlows Software. Users are introduced to the modeling power of the latest software with access to continuously updated tutorials found in the Bentley Systems website. The text explains the theory and equations on which the software is based, and each chapter includes several problems for the reader to practice their problem-solving and modeling skills.

Included in this 9th Edition:

- | | |
|-------------|-----------------|
| > WaterGEMS | > PondPack |
| > HAMMER | > StormCAD |
| > SewerGEMS | > FlowMaster |
| > SewerCAD | > CulvertMaster |

Bentley
Advancing Infrastructure



Category: Bentley Hydrology and Hydraulics
Authors: Brian E. Whitman and Thomas Walski
User Level: Upper-Level Undergraduate Student
Professional Experience Level: Beginner

www.bentley.com/books

