DISCLAIMER

The information in this document has been funded wholly or in part by the U.S. Environmental Protection Agency (EPA). It has been subjected to the Agency’s peer and administrative review, and has been approved for publication as an EPA document. Mention of trade names or commercial products does not constitute endorsement or recommendation for use.

Although a reasonable effort has been made to assure that the results obtained are correct, the computer programs described in this manual are experimental. Therefore the author and the U.S. Environmental Protection Agency are not responsible and assume no liability whatsoever for any results or any use made of the results obtained from these programs, nor for any damages or litigation that result from the use of these programs for any purpose.
FOREWORD

The U.S. Environmental Protection Agency is charged by Congress with protecting the Nation’s land, air, and water resources. Under a mandate of national environmental laws, the Agency strives to formulate and implement actions leading to a compatible balance between human activities and the ability of natural systems to support and nurture life. To meet this mandate, EPA’s research program is providing data and technical support for solving environmental problems today and building a science knowledge base necessary to manage our ecological resources wisely, understand how pollutants affect our health, and prevent or reduce environmental risks in the future.

The National Risk Management Research Laboratory is the Agency’s center for investigation of technological and management approaches for reducing risks from threats to human health and the environment. The focus of the Laboratory’s research program is on methods for the prevention and control of pollution to the air, land, water, and subsurface resources; protection of water quality in public water systems; remediation of contaminated sites and ground water; and prevention and control of indoor air pollution. The goal of this research effort is to catalyze development and implementation of innovative, cost-effective environmental technologies; develop scientific and engineering information needed by EPA to support regulatory and policy decisions; and provide technical support and information transfer to ensure effective implementation of environmental regulations and strategies.

Water quality impairment due to runoff from urban and developing areas continues to be a major threat to the ecological health of our nation’s waterways. The EPA Stormwater Management Model is a computer program that can assess the impacts of such runoff and evaluate the effectiveness of mitigation strategies. The modernized and updated version of the model described in this document will make it a more accessible and valuable tool for researchers and practitioners engaged in water resources and water quality planning and management.

Sally C. Gutierrez, Acting Director
National Risk Management Research Laboratory
ACKNOWLEDGEMENTS

The development of SWMM 5 was pursued under a Cooperative Research and Development Agreement between the Water Supply and Water Resources Division of the U.S. Environmental Protection Agency and the consulting engineering firm of Camp Dresser & McKee Inc.. The project team consisted of the following individuals:

US EPA         CDM
Lewis Rossman   Robert Dickinson
Trent Schade    Carl Chan
Daniel Sullivan (retired) Edward Burgess

The team would like to acknowledge the assistance provided by Wayne Huber (Oregon State University), Dennis Lai (US EPA), and Michael Gregory (CDM). We also want to acknowledge the contributions made by the following individuals to previous versions of SWMM that we drew heavily upon in this new version: John Aldrich, Douglas Ammon, Carl W. Chen, Brett Cunningham, Robert Dickinson, James Heaney, Wayne Huber, Miguel Medina, Russell Mein, Charles Moore, Stephan Nix, Alan Peltz, Don Polmann, Larry Roesner, Charles Rowney, and Robert Shubinsky. Finally, we wish to thank Wayne Huber, Thomas Barnwell (US EPA), Richard Field (US EPA), Harry Torno (US EPA retired) and William James (University of Guelph) for their continuing efforts to support and maintain the program over the past several decades.
CHAPTER 4 - SWMM’S MAIN WINDOW .............................................. 61
4.1 Overview ................................................................................................. 61
4.2 Main Menu ............................................................................................... 62
4.3 Toolbars ..................................................................................................... 65
4.4 Status Bar .................................................................................................. 67
4.5 Study Area Map ......................................................................................... 68
4.6 Data Browser ............................................................................................. 68
4.7 Map Browser .............................................................................................. 69
4.8 Property Editor .......................................................................................... 70
4.9 Setting Program Preferences ...................................................................... 71

CHAPTER 5 - WORKING WITH PROJECTS .................................... 73
5.1 Creating a New Project ............................................................................. 73
5.2 Opening an Existing Project ..................................................................... 73
5.3 Saving a Project ......................................................................................... 73
5.4 Setting Project Defaults ........................................................................... 74
5.5 Calibration Data ....................................................................................... 76
5.6 Viewing All Project Data ........................................................................... 77

CHAPTER 6 - WORKING WITH OBJECTS ..................................... 79
6.1 Types of Objects ....................................................................................... 79
6.2 Adding Objects ......................................................................................... 79
6.3 Selecting and Moving Objects ................................................................. 82
6.4 Editing Objects ......................................................................................... 82
6.5 Converting an Object ............................................................................... 83
6.6 Copying and Pasting Objects ................................................................... 84
6.7 Shaping and Reversing Links ................................................................. 84
6.8 Shaping a Subcatchment ......................................................................... 85
6.9 Deleting an Object........................................................................................................... 85
6.10 Editing or Deleting a Group of Objects............................................................................ 85

CHAPTER 7 - WORKING WITH THE MAP .......................................................... 87
7.1 Selecting a Map Theme................................................................................................. 87
7.2 Setting the Map’s Dimensions .................................................................................... 87
7.3 Utilizing a Backdrop Image ......................................................................................... 88
7.4 Zooming the Map......................................................................................................... 91
7.5 Panning the Map.......................................................................................................... 91
7.6 Viewing at Full Extent................................................................................................ 92
7.7 Finding an Object.......................................................................................................... 92
7.8 Submitting a Map Query ............................................................................................ 93
7.9 Using the Map Legends............................................................................................... 94
7.10 Using the Overview Map.......................................................................................... 95
7.11 Setting Map Display Options ................................................................................... 96
7.12 Exporting the Map..................................................................................................... 99

CHAPTER 8 - RUNNING A SIMULATION...................................................... 101
8.1 Setting Simulation Options ......................................................................................... 101
8.2 Starting a Simulation................................................................................................. 106
8.3 Troubleshooting Results............................................................................................ 107

CHAPTER 9 - VIEWING RESULTS.............................................................. 109
9.1 Viewing a Status Report.............................................................................................. 109
9.2 Variables That Can be Viewed.................................................................................. 109
9.3 Viewing Results on the Map .................................................................................... 110
9.4 Viewing Results with a Graph................................................................................... 110
9.5 Customizing a Graph’s Appearance.......................................................................... 115
9.6 Viewing Results with a Table................................................................................... 119
# Viewing a Statistics Report

## Chapter 10 - Printing and Copying

10.1 Selecting a Printer

10.2 Setting the Page Format

10.3 Print Preview

10.4 Printing the Current View

10.5 Copying to the Clipboard or to a File

## Chapter 11 - Files Used by SWMM

11.1 Project Files

11.2 Report and Output Files

11.3 Rainfall Files

11.4 Climate Files

11.5 Calibration Files

11.6 Time Series Files

11.7 Interface Files

## Appendix A - Useful Tables

A.1 Units of Measurement

A.2 Soil Characteristics

A.3 Soil Group Definitions

A.4 SCS Curve Numbers

A.5 Depression Storage

A.6 Manning’s n – Overland Flow

A.7 Manning’s n – Closed Conduits

A.8 Manning’s n – Open Channels

A.9 Water Quality Characteristics of Urban Runoff
APPENDIX B - VISUAL OBJECT PROPERTIES .................. 145

B.1 Rain Gage Properties ................................................................. 145
B.2 Subcatchment Properties .......................................................... 146
B.3 Junction Properties ................................................................. 147
B.4 Outfall Properties ....................................................................... 148
B.5 Flow Divider Properties ............................................................. 149
B.6 Storage Unit Properties ............................................................ 150
B.7 Conduit Properties ....................................................................... 151
B.8 Pump Properties ........................................................................... 152
B.9 Orifice Properties .......................................................................... 152
B.10 Weir Properties ........................................................................... 153
B.11 Outlet Properties .......................................................................... 154
B.12 Map Label Properties ............................................................... 154

APPENDIX C - SPECIALIZED PROPERTY EDITORS ...... 155

C.1 Aquifer Editor .............................................................................. 155
C.2 Climatology Editor ........................................................................ 156
C.3 Control Rules Editor ....................................................................... 161
C.4 Cross-Section Editor ....................................................................... 164
C.5 Curve Editor .................................................................................. 165
C.6 Groundwater Flow Editor ............................................................. 166
C.7 Infiltration Editor ............................................................................ 167
C.8 Infows Editor ................................................................................ 170
C.9 Initial Buildup Editor ..................................................................... 173
C.10 Land Use Editor ............................................................................ 174
C.11 Land Use Assignment Editor ....................................................... 177
C.12 Pollutant Editor ............................................................................ 178
C13. Snow Pack Editor........................................................................................................... 179
C.14 Time Pattern Editor ........................................................................................................ 183
C.15 Time Series Editor ......................................................................................................... 184
C.16 Title/Notes Editor ......................................................................................................... 186
C.17 Transect Editor ............................................................................................................... 186
C.18 Treatment Editor ........................................................................................................... 187
C.19 Unit Hydrograph Editor .......................................................................................... 188

A P P E N D I X  D - C O M M A N D  L I N E  S W M M .................................................191

A P P E N D I X  E - E R R O R  M E S S A G E S ...........................................................229
1.1 What is SWMM

The EPA Storm Water Management Model (SWMM) is a dynamic rainfall-runoff simulation model used for single event or long-term (continuous) simulation of runoff quantity and quality from primarily urban areas. The runoff component of SWMM operates on a collection of subcatchment areas that receive precipitation and generate runoff and pollutant loads. The routing portion of SWMM transports this runoff through a system of pipes, channels, storage/treatment devices, pumps, and regulators. SWMM tracks the quantity and quality of runoff generated within each subcatchment, and the flow rate, flow depth, and quality of water in each pipe and channel during a simulation period comprised of multiple time steps.

SWMM was first developed in 1971 and has undergone several major upgrades since then. It continues to be widely used throughout the world for planning, analysis and design related to storm water runoff, combined sewers, sanitary sewers, and other drainage systems in urban areas, with many applications in non-urban areas as well. The current edition, Version 5, is a complete re-write of the previous release. Running under Windows, SWMM 5 provides an integrated environment for editing study area input data, running hydrologic, hydraulic and water quality simulations, and viewing the results in a variety of formats. These include color-coded drainage area and conveyance system maps, time series graphs and tables, profile plots, and statistical frequency analyses.

This latest re-write of SWMM was produced by the Water Supply and Water Resources Division of the U.S. Environmental Protection Agency's National Risk Management Research Laboratory with assistance from the consulting firm of CDM, Inc.

1.2 Modeling Capabilities

SWMM accounts for various hydrologic processes that produce runoff from urban areas. These include:

- time-varying rainfall
- evaporation of standing surface water
- snow accumulation and melting
- rainfall interception from depression storage
- infiltration of rainfall into unsaturated soil layers

---

- percolation of infiltrated water into groundwater layers
- interflow between groundwater and the drainage system
- nonlinear reservoir routing of overland flow

Spatial variability in all of these processes is achieved by dividing a study area into a collection of smaller, homogeneous subcatchment areas, each containing its own fraction of pervious and impervious sub-areas. Overland flow can be routed between sub-areas, between subcatchments, or between entry points of a drainage system.

SWMM also contains a flexible set of hydraulic modeling capabilities used to route runoff and external inflows through the drainage system network of pipes, channels, storage/treatment units and diversion structures. These include the ability to:
- handle networks of unlimited size
- use a wide variety of standard closed and open conduit shapes as well as natural channels
- model special elements such as storage/treatment units, flow dividers, pumps, weirs, and orifices
- apply external flows and water quality inputs from surface runoff, groundwater interflow, rainfall-dependent infiltration/inflow, dry weather sanitary flow, and user-defined inflows
- utilize either kinematic wave or full dynamic wave flow routing methods
- model various flow regimes, such as backwater, surcharging, reverse flow, and surface ponding
- apply user-defined dynamic control rules to simulate the operation of pumps, orifice openings, and weir crest levels.

In addition to modeling the generation and transport of runoff flows, SWMM can also estimate the production of pollutant loads associated with this runoff. The following processes can be modeled for any number of user-defined water quality constituents:
- dry-weather pollutant buildup over different land uses
- pollutant washoff from specific land uses during storm events
- direct contribution of rainfall deposition
- reduction in dry-weather buildup due to street cleaning
- reduction in washoff load due to BMPs
- entry of dry weather sanitary flows and user-specified external inflows at any point in the drainage system
- routing of water quality constituents through the drainage system
- reduction in constituent concentration through treatment in storage units or by natural processes in pipes and channels.
1.3 Typical Applications of SWMM

Since its inception, SWMM has been used in thousands of sewer and stormwater studies throughout the world. Typical applications include:

- design and sizing of drainage system components for flood control
- sizing of detention facilities and their appurtenances for flood control and water quality protection
- flood plain mapping of natural channel systems
- designing control strategies for minimizing combined sewer overflows
- evaluating the impact of inflow and infiltration on sanitary sewer overflows
- generating non-point source pollutant loadings for waste load allocation studies
- evaluating the effectiveness of BMPs for reducing wet weather pollutant loadings.

1.4 Installing EPA SWMM

EPA SWMM Version 5 is designed to run under the Windows 98/NT/ME/2000/XP operating system of an IBM/Intel-compatible personal computer. It is distributed as a single file, epaswmm5_setup.exe, which contains a self-extracting setup program. To install EPA SWMM:

1. Select Run from the Windows Start menu.
2. Enter the full path and name of the epaswmm5_setup.exe file or click the Browse button to locate it on your computer.
3. Click the OK button type to begin the setup process.

The setup program will ask you to choose a folder (directory) where the SWMM program files will be placed. The default folder is c:\Program Files\EPA SWMM 5.0. After the files are installed your Start Menu will have a new item named EPA SWMM 5.0. To launch SWMM, simply select this item off of the Start Menu, then select EPA SWMM 5.0 from the submenu that appears. (The name of the executable file that runs SWMM under Windows is epaswmm5.exe.)

If SWMM is being installed within a multi-user network environment, then the network administrator may wish to install a shortcut to SWMM 5 on each user’s desktop whose target entry includes the name of the SWMM 5 executable followed by /s <userfolder>, where <userfolder> is the name of the folder where the user’s personal SWMM settings will be stored. An example might be:

“c:\Program Files\EPA SWMM 5.0\epaswmm5.exe” /s “My Folders\SWMM5\”.

This will allow SWMM to save the user’s personal program settings to a different location than where SWMM was installed so it will not overwrite any previously saved settings from other users.
To remove EPA SWMM from your computer, do the following:
1. Select **Settings** from the Windows Start menu.
2. Select **Control Panel** from the Settings menu.
3. Double-click on the **Add/Remove Programs** item.
4. Select **EPA SWMM 5.0** from the list of programs that appears.
5. Click the **Add/Remove** button.

### 1.5 Steps in Using SWMM

One typically carries out the following steps when using SWMM to model stormwater runoff over a study area:
1. Specify a default set of options and object properties to use (see Section 4.2).
2. Draw a network representation of the physical components of the study area (see Section 5.2).
3. Edit the properties of the objects that make up the system (see Section 5.4).
4. Select a set of analysis options (see Section 7.1).
5. Run a simulation (see Section 7.2).
6. View the results of the simulation (see Chapter 8).

Alternatively, a modeler may convert an input file from an older version of EPA SWMM instead of developing a new model as in Steps 1 through 4.

### 1.6 About This Manual

Chapter 2 presents a short tutorial to help get started using EPA SWMM. It shows how to add objects to a SWMM project, how to edit the properties of these objects, how to run a single event simulation for both hydrology and water quality, and how to run a long-term continuous simulation.

Chapter 3 provides background material on how SWMM models stormwater runoff within a drainage area. It discusses the behavior of the physical components that comprise a stormwater drainage area and collection system as well as how additional modeling information, such as rainfall quantity, dry weather sanitary inflows, and operational control, are handled. It also provides an overview of how the numerical simulation of system hydrology, hydraulics and water quality behavior is carried out.

Chapter 4 shows how the EPA SWMM graphical user interface is organized. It describes the functions of the various menu options and toolbar buttons, and how the three main windows – the Study Area Map, the Browser panel, and the Property Editor—are used.

Chapter 5 discusses the project files that store all of the information contained in a SWMM model of a drainage system. It shows how to create, open, and save these files as
well as how to set default project options. It also discusses how to register calibration data that are used to compare simulation results against actual measurements.

Chapter 6 describes how one goes about building a network model of a drainage system with EPA SWMM. It shows how to create the various physical objects (subcatchment areas, drainage pipes and channels, pumps, weirs, storage units, etc.) that make up a system, how to edit the properties of these objects, and how to describe the way that externally imposed inflows, boundary conditions and operational controls change over time.

Chapter 7 explains how to use the study area map that provides a graphical view of the system being modeled. It shows how to view different design and computed parameters in color-coded fashion on the map, how to re-scale, zoom, and pan the map, how to locate objects on the map, how to utilize a backdrop image, and what options are available to customize the appearance of the map.

Chapter 8 shows how to run a simulation of a SWMM model. It describes the various options that control how the analysis is made and offers some troubleshooting tips to use when examining simulation results.

Chapter 9 discusses the various ways in which the results of an analysis can be viewed. These include different views of the study area map, various kinds of graphs and tables, and several different types of special reports.

Chapter 10 explains how to print and copy the results discussed in Chapter 9.

Chapter 11 describes how EPA SWMM can use different types of interface files to make simulations runs more efficient.

The manual also contains several appendixes:
Appendix A - provides several useful tables of parameter values, including a table of units of expression for all design and computed parameters.
Appendix B - lists the editable properties of all visual objects that can be displayed on the study area map and be selected for editing using point and click.
Appendix C - describes the specialized editors available for setting the properties of non-visual objects.
Appendix D - provides instructions for running the command line version of SWMM and includes a detailed description of the format of a project file.
Appendix E - lists all of the error messages and their meaning that SWMM can produce.
CHAPTER 2 - QUICK START TUTORIAL

This chapter provides a tutorial on how to use EPA SWMM. If you are not familiar with the elements that comprise a drainage system, and how these are represented in a SWMM model, you might want to review the material in Chapter 3 first.

2.1 Example Study Area

In this tutorial we will model the drainage system serving a 12-acre residential area. The system layout is shown in Figure 2-1 and consists of subcatchment areas 3 S1 through S3, storm sewer conduits C1 through C4, and conduit junctions J1 through J4. The system discharges to a creek at the point labeled Out1. We will first go through the steps of creating the objects shown in this diagram on SWMM’s study area map and setting the various properties of these objects. Then we will simulate the water quantity and quality response to a 3-inch, 6-hour rainfall event, as well as a continuous, multi-year rainfall record.

![Figure 2-1. Example study area.](image)

2.2 Project Setup

Our first task is to create a new SWMM project and make sure that certain default options are selected. Using these defaults will simplify the data entry tasks later on.

1. Launch EPA SWMM if it is not already running and select **File >> New** from the Main Menu bar to create a new project.

2. Select **Project >> Defaults** to open the Project Defaults dialog.

---

3 A subcatchment is an area of land containing a mix of pervious and impervious surfaces whose runoff drains to a common outlet point, which could be either a node of the drainage network or another subcatchment.
3. On the ID Labels page of the dialog, set the ID Prefixes as shown in Figure 2-2. This will make SWMM automatically label new objects with consecutive numbers following the designated prefix.

![Project Defaults dialog](image)

Figure 2-2. Default ID labeling for tutorial example.

4. On the Subcatchments page of the dialog set the following default values:
   - **Area**: 4
   - **Width**: 400
   - **% Slope**: 0.5
   - **% Imperv.**: 50
   - **N-Imperv.**: 0.01
   - **N-Perv.**: 0.10
   - **Dstore-Imperv.**: 0.05
   - **Dstore-Perv**: 0.05
   - **%Zero-Imperv.**: 25
   - **Infil. Model**
     - **Method**: Green-Ampt
     - **Suction Head**: 3.5
     - **Conductivity**: 0.5
     - **Initial Deficit**: 0.26
5. On the Nodes/Links page set the following default values:

- **Node Invert**: 0
- **Node Max. Depth**: 4
- **Flow Units**: CFS
- **Conduit Length**: 400
- **Conduit Geometry**: <click to edit>
  - **Barrels**: 1
  - **Shape**: Circular
  - **Max. Depth**: 1.0
- **Conduit Roughness**: 0.01
- **Routing Model**: Kinematic Wave

6. Click **OK** to accept these choices and close the dialog. If you wanted to save these choices for all future new projects you could check the **Save** box at the bottom of the form before accepting it.

Next we will set some map display options so that ID labels and symbols will be displayed as we add objects to the study area map, and links will have direction arrows.

1. Select **View >> Map Options** to bring up the Map Options dialog (see Figure 2-3).
2. Select the Subcatchments page, set the Fill Style to Diagonal and the Symbol Size to 5.
3. Then select the Nodes page and set the Node Size to 5.
4. Select the Annotation page and check off the boxes that will display ID labels for Areas, Nodes, and Links. Leave the others un-checked.
5. Finally, select the Flow Arrows page, select the Filled arrow style, and set the arrow size to 7.
6. Click the **OK** button to accept these choices and close the dialog.
Before placing objects on the map we should set its dimensions.

1. Select **View >> Dimensions** to bring up the Map Dimensions dialog.

2. You can leave the dimensions at their default values for this example.

Finally, look in the status bar at the bottom of the main window and check that the Auto-Length feature is off. If it is on, then right-click over the status bar and select “Auto-Length Off” from the popup menu that appears.

### 2.3 Drawing Objects

We are now ready to begin adding components to the Study Area Map\(^4\). We will start with the subcatchments.

---

\(^4\) Drawing objects on the map is just one way of creating a project. For large projects it might be more convenient to first construct a SWMM project file external to the program. The project file is a text file that describes each object in a specified format as described in Appendix D of this manual. Data extracted from various sources, such as CAD drawings or GIS files, can be used to create the project file.
1. Begin by clicking the button on the Object Toolbar. (If the toolbar is not visible then select View >> Toolbars >> Object). Notice how the mouse cursor changes shape to a pencil.

2. Move the mouse to the map location where one of the corners of subcatchment S1 lies and left-click the mouse.

3. Do the same for the next two corners and then right-click the mouse (or hit the Enter key) to close up the rectangle that represents subcatchment S1. You can press the Esc key if instead you wanted to cancel your partially drawn subcatchment and start over again. Don't worry if the shape or position of the object isn't quite right. We will go back later and show how to fix this.

4. Repeat this process for subcatchments S2 and S3.

Observe how sequential ID labels are generated automatically as we add objects to the map.

Next we will add in the junction nodes and the outfall node that comprise part of the drainage network.

1. To begin adding junctions, click the button on the Object Toolbar.

2. Move the mouse to the position of junction J1 and left-click it. Do the same for junctions J2 through J4.

3. To add the outfall node, click the button on the Object Toolbar, move the mouse to the outfall's location on the map, and left-click. Note how the outfall was automatically given the name Out1.

At this point your map should look something like that shown in Figure 2.4.

![Figure 2-4. Subcatchments and nodes for example study area.](image)

Now we will add the storm sewer conduits that connect our drainage system nodes to one another. (You must have created a link's end nodes as described previously before you can create the link.) We will begin with conduit C1, which connects junction J1 to J2.

---

5 If you right-click (or press Enter) after adding the first point of a subcatchment's outline, the subcatchment will be shown as just a single point.
1. Click the button on the Object Toolbar. The mouse cursor changes shape to a crosshair.

2. Click the mouse on junction J1. Note how the mouse cursor changes shape to a pencil.

3. Move the mouse over to junction J2 (note how an outline of the conduit is drawn as you move the mouse) and left-click to create the conduit. You could have cancelled the operation by either right clicking or by hitting the <Esc> key.

4. Repeat this procedure for conduits C2 through C4.

Although all of our conduits were drawn as straight lines, it is possible to draw a curved link by left-clicking at intermediate points where the direction of the link changes before clicking on the end node.

To complete the construction of our study area schematic we need to add a rain gage.

1. Click the Rain Gage button on the Object Toolbar.

2. Move the mouse over the Study Area Map to where the gage should be located and left-click the mouse.

At this point we have completed drawing the example study area. Your system should look like the one in Figure 2.1. If a rain gage, subcatchment or node is out of position you can move it by doing the following:

1. If the button is not already depressed, click it to place the map in Object Selection mode.

2. Click on the object to be moved.

3. Drag the object with the left mouse button held down to its new position.

To re-shape a subcatchment's outline:

1. With the map in Object Selection mode, click on the subcatchment's centroid (indicated by a solid square within the subcatchment) to select it.

2. Then click the button on the Map Toolbar to put the map into Vertex Selection mode.

3. Select a vertex point on the subcatchment outline by clicking on it (note how the selected vertex is indicated by a filled solid square).

4. Drag the vertex to its new position with the left mouse button held down.

5. If need be, vertices can be added or deleted from the outline by right-clicking the mouse and selecting the appropriate option from the popup menu that appears.

6. When finished, click the button to return to Object Selection mode.

This same procedure can also be used to re-shape a link.
2.4 Setting Object Properties

As visual objects are added to our project, SWMM assigns them a default set of properties. To change the value of a specific property for an object we must select the object into the Property Editor (see Figure 2-5). There are several different ways to do this. If the Editor is already visible, then you can simply click on the object or select it from the Data page of the Browser Panel of the main window. If the Editor is not visible then you can make it appear by one of the following actions:

- double-click the object on the map,
- or right-click on the object and select Properties from the pop-up menu that appears,
- or select the object from the Data page of the Browser panel and then click the Browser’s button.

![Subcatchment S1](image)

Figure 2-5. Property Editor window.

Whenever the Property Editor has the focus you can press the F1 key to obtain a more detailed description of the properties listed.

Two key properties of our subcatchments that need to be set are the rain gage that supplies rainfall data to the subcatchment and the node of the drainage system that receives runoff from the subcatchment. Since all of our subcatchments utilize the same rain gage, Gage1, we can use a shortcut method to set this property for all subcatchments at once:

1. From the main menu select **Edit >>Select All**.
2. Then select **Edit >> Group Edit** to make a Group Editor dialog appear (see Figure 2-6).
3. Select **Subcatchments** as the class of object to edit, **Rain Gage** as the property to edit, and type in **Gage1** as the new value.
4. Click **OK** to change the rain gage of all subcatchments to **Gage1**. A confirmation dialog will appear noting that 3 subcatchments have changed. Select “**No**” when asked to continue editing.

![Group Editor dialog](image)

**Figure 2-6. Group Editor dialog.**

Because the outlet nodes vary by subcatchment, we must set them individually as follows:

1. Double click on subcatchment **S1** or select it from the Data Browser and click the Browser's **button to bring up the Property Editor.

2. Type **J1** in the Outlet field and press **Enter**. Note how a dotted line is drawn between the subcatchment and the node.

3. Click on subcatchment **S2** and enter **J2** as its Outlet.

4. Click on subcatchment **S3** and enter **J3** as its Outlet.

We also wish to represent area **S3** as being less developed than the others. Select **S3** into the Property Editor and set its % Imperviousness to **25**.

The junctions and outfall of our drainage system need to have invert elevations assigned to them. As we did with the subcatchments, select each junction individually into the Property Editor and set its Invert Elevation to the value shown below.

<table>
<thead>
<tr>
<th>Node</th>
<th>Invert</th>
</tr>
</thead>
<tbody>
<tr>
<td>J1</td>
<td>96</td>
</tr>
<tr>
<td>J2</td>
<td>90</td>
</tr>
<tr>
<td>J3</td>
<td>93</td>
</tr>
<tr>
<td>J4</td>
<td>88</td>
</tr>
<tr>
<td>Out1</td>
<td>85</td>
</tr>
</tbody>
</table>

---

6 An alternative way to move from one object of a given type to the next in order (or to the previous one) in the Property Editor is to hit the Page Down (or Page Up) key.
Only one of the conduits in our example system has a non-default property value. This is conduit C4, the outlet pipe, whose diameter should be 1.5 instead of 1 ft. To change its diameter:

1. Select conduit C4 into the Property Editor (by either double-clicking anywhere on the conduit itself or by selecting it in the Data Browser and clicking the button).
2. Select the Shape field and click the ellipsis button (or press <Enter>).
3. In the Cross-Section Editor dialog that appears (see Figure 2-7), set the Max. Depth value to 1.5 and then click the OK button.

![Cross-Section Editor dialog](image)

**Figure 2-7. Cross-Section Editor dialog.**

In order to provide a source of rainfall input to our project we need to set the rain gage’s properties. Select Gage1 into the Property Editor and set the following properties:

- **Rain Format**: INTENSITY
- **Rain Interval**: 1:00
- **Data Source**: TIMESERIES
- **Series Name**: TS1

As mentioned earlier, we want to simulate the response of our study area to a 3-inch, 6-hour design storm. A time series named **TS1** will contain the hourly rainfall intensities that make up this storm. Thus we need to create a time series object and populate it with data. To do this:

1. From the Data Browser select the Time Series category of objects.
2. Click the + button on the Browser to bring up the Time Series Editor dialog (see Figure 2-8).
3. Enter **TS1** in the Time Series Name field.

7 The Time Series Editor can also be launched directly from the Rain Gage Property Editor by selecting the editor's Series Name field and double clicking on it.
4. Enter the values shown in Figure 2.8 into the Time and Value columns of the data entry grid (leave the Date column blank\(^8\)).

5. You can click the View button on the dialog to see a graph of the time series values. Click the OK button to accept the new time series.

![Time Series Editor dialog](image)

**Figure 2-8. Time Series Editor dialog.**

Having completed the initial design of our example project it is a good idea to give it a title and save our work to a file at this point. To do this:

1. Select the Title/Notes category from the Data Browser and click the button.

2. In the Project Title/Notes dialog that appears (see Figure 2-9), enter “Tutorial Example” as the title of our project and click the OK button to close the dialog.

3. From the File menu select the Save As option.

4. In the Save As dialog that appears, select a folder and file name under which to save this project. We suggest naming the file tutorial.inp. (An extension of .inp will be added to the file name if one is not supplied.)

\(^8\) Leaving off the dates for a time series means that SWMM will interpret the time values as hours from the start of the simulation. Otherwise, the time series follows the date/time values specified by the user.
5. Click Save to save the project to file.

The project data are saved to the file in a readable text format. You can view what the file looks like by selecting Project >> Details from the main menu. To open our project at some later time, you would select the Open command from the File menu.

![Title/Notes Editor](image)

**Figure 2-9. Title/Notes Editor.**

### 2.5 Running a Simulation

#### Setting Simulation Options

Before analyzing the performance of our example drainage system we need to set some options that determine how the analysis will be carried out. To do this:

1. From the Data Browser, select the Options category and click the button.
2. On the General page of the Simulation Options dialog that appears (see Figure 2-10), select Kinematic Wave as the flow routing method. The flow units should already be set to CFS and the infiltration method to Green-Ampt. The Allow Ponding option should be unchecked.
3. On the Dates page of the dialog, set the End Analysis time to 12:00:00.
4. On the Time Steps page, set the Routing Time Step to 00:01:00.
5. Click OK to close the Simulation Options dialog.

#### Running a Simulation

We are now ready to run the simulation. To do so, select Project >> Run Simulation (or click the button). If there was a problem with the simulation, a Status Report will appear describing what errors occurred. Upon successfully completing a run, there are numerous ways in which to view the results of the simulation. We will illustrate just a few here.
Viewing the Status Report

The Status Report contains useful summary information about the results of a simulation run. To view the report, select **Report >> Status**. A portion of the report for the system just analyzed is shown in Figure 2-11. The full report indicates the following:

- The quality of the simulation is quite good, with negligible mass balance continuity errors for both runoff and routing (-0.23% and 0.03%, respectively, if all data were entered correctly).
- Of the 3 inches of rain that fell on the study area, 1.75 infiltrated into the ground and essentially the remainder became runoff.
- The Node Depth Summary table (not shown in Figure 2-11) indicates there was internal flooding in the system at node \( J2 \). \(^9\)

\(^9\) In SWMM, flooding will occur whenever the water surface at a node exceeds the maximum defined depth. Normally such water will be lost from the system. The option also exists to have this water pond atop the node and be re-introduced into the drainage system when capacity exists to do so.
The Conduit Flow Summary table (also not shown in Figure 2-11) shows that Conduit C2, just downstream of node J2, was surcharged and therefore appears to be slightly undersized.

---

**EPA STORM WATER MANAGEMENT MODEL - VERSION 5.0**

-----------------------------------------------

**Tutorial Example**

**************

**Analysis Options**

**************

Flow Units ............... CFS
Infiltration Method ...... GREEN_AMPT
Flow Routing Method ...... KW
Starting Date ............ JUN-27-2002 00:00:00
Ending Date .............. JUN-27-2002 12:00:00
Wet Time Step ............ 00:15:00
Dry Time Step ............. 01:00:00
Routing Time Step ........ 00:01:00
Report Time Step ........ 00:15:00

<table>
<thead>
<tr>
<th>Runoff Quantity Continuity</th>
<th>Volume</th>
<th>Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Precipitation ......</td>
<td>3.000</td>
<td>3.000</td>
</tr>
<tr>
<td>Evaporation Loss ..........</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Infiltration Loss ..........</td>
<td>1.750</td>
<td>1.750</td>
</tr>
<tr>
<td>Surface Runoff ...........</td>
<td>1.241</td>
<td>1.241</td>
</tr>
<tr>
<td>Final Surface Storage ....</td>
<td>0.016</td>
<td>0.016</td>
</tr>
<tr>
<td>Continuity Error (%) .....</td>
<td>-0.228</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flow Routing Continuity</th>
<th>Volume</th>
<th>Volume</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dry Weather Inflow ......</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Wet Weather Inflow ......</td>
<td>1.246</td>
<td>0.406</td>
</tr>
<tr>
<td>Groundwater Inflow ......</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>RDII Inflow .............</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>External Inflow ..........</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Internal Flooding .......</td>
<td>0.054</td>
<td>0.018</td>
</tr>
<tr>
<td>External Outflow ........</td>
<td>1.192</td>
<td>0.388</td>
</tr>
<tr>
<td>Evaporation Loss ..........</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Initial Stored Volume ....</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Final Stored Volume ......</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Continuity Error (%) .....</td>
<td>0.029</td>
<td></td>
</tr>
</tbody>
</table>

---

**Figure 2-11. Portion of the Status Report for initial simulation run.**
Viewing Results on the Map

Simulation results (as well as some design parameters, such as subcatchment area, node invert elevation, link maximum depth) can be viewed in color-coded fashion on the study area map. To view a particular variable in this fashion:

1. Select the Map page of the Browser panel.
2. Select the variables to view for Subcatchments, Nodes, and Links from the dropdown combo boxes labeled Subcatch View, Node View, and Link View, respectively. In Figure 2-12, subcatchment runoff and link flow have been selected for viewing.
3. The color-coding used for a particular variable is displayed with a legend on the study area map. To toggle the display of a legend, select View >> Legends.
4. To move a legend to another location, drag it with the left mouse button held down.
5. To change the color-coding and the breakpoint values for different colors, select View >> Legends >> Modify and then the pertinent class of object (or if the legend is already visible, simply right-click on it). To view numerical values for the variables being displayed on the map, select View >> Map Options and then select the Annotation page of the Map Options dialog. Use the check boxes for Areas (i.e., sub-catchments), Nodes, and Links in the Values panel to specify what kind of annotation to add.
6. The Date / Time / Elapsed Time controls on the Map Browser can be used to move through the simulation results in time. Figure 2-12 depicts results at 1 hour and 15 minutes into the simulation.
7. To animate the map display through time, select View >> Toolbars >> Animator and use the controls on the Animator Toolbar to control the animation. For example, pressing the button will run the animation forward in time.

Viewing a Time Series Plot

To generate a time series plot of a simulation result:

1. Select Report >> Graph >> Time Series or simply click on the Standard Toolbar and select Time Series from the pull-down menu that appears.
2. A Time Series Plot dialog will appear. It is used to select the objects and variables to be plotted.

For our example, the Time Series Plot dialog can be used to graph the flow in conduits C1 and C2 as follows (refer to Figure 2-14):

1. Select Links as the Object Category.
2. Select Flow as the Variable to plot.
3. Click on conduit C1 (either on the map or in the Data Browser) and then click in the dialog to add it to the list of links plotted. Do the same for conduit C2.
4. Press OK to create the plot, which should look like the graph in Figure 2-15.
Figure 2-12. Example of viewing color-coded results on the Study Area Map.

Figure 2-13. The Animator Toolbar.
Figure 2-14. Time Series Plot dialog.

Figure 2-15. Time Series plot of results from initial simulation run.
After a plot is created you can:

- customize its appearance by selecting **Report >> Options** or right clicking on the plot,
- copy it to the clipboard and paste it into another application by selecting **Edit >> Copy To** or clicking on the Standard Toolbar
- print it by selecting **File >> Print** or **File >> Print Preview** (use **File >> Page Setup** first to set margins, orientation, etc.).

**Viewing a Profile Plot**

SWMM can generate profile plots showing how water surface depth varies across a path of connected nodes and links. Let's create such a plot for the conduits connecting junction **J1** to the outfall **Out1** of our example drainage system. To do this:

1. Select **Report >> Graph >> Profile** or simply click on the Standard Toolbar and select **Profile** from the pull-down menu that appears.
2. Either enter **J1** in the Start Node field of the Profile Plot dialog that appears (see Figure 2-16) or select it on the map or from the Data Browser and click the button next to the field.

![Profile Plot dialog](image)

**Figure 2-16. Profile Plot dialog.**
3. Do the same for node **Out1** in the End Node field of the dialog.

4. Click the **Find Path** button. An ordered list of the links forming a connected path between the specified Start and End nodes will be displayed in the Links in Profile box. You can edit the entries in this box if need be.

5. Click the **OK** button to create the plot, showing the water surface profile as it exists at the simulation time currently selected in the Map Browser (see Figure 2-17).

![Profile - Node J1 - Out1](image)

**Figure 2-17. Example of a Profile plot.**

As you move through time using the Map Browser or with the Animator control, the water depth profile on the plot will be updated. Observe how node **J2** becomes flooded between hours 2 and 3 of the storm event. A Profile Plot’s appearance can be customized and it can be copied or printed using the same procedures as for a Time Series Plot.

**Running a Full Dynamic Wave Analysis**

In the analysis just run we chose to use the Kinematic Wave method of routing flows through our drainage system. This is an efficient but simplified approach that cannot deal with such phenomena as backwater effects, pressurized flow, flow reversal, and non-dendritic layouts. SWMM also includes a Dynamic Wave routing procedure that can represent these conditions. This procedure, however, requires more computation time, due to the need for smaller time steps to maintain numerical stability.

Most of the effects mentioned above would not apply to our example. However we had one conduit, **C2**, which flowed full and caused its upstream junction to flood. It could be
that this pipe is actually being pressurized and could therefore convey more flow than was computed using Kinematic Wave routing. We would now like to see what would happen if we apply Dynamic Wave routing instead.

To run the analysis with Dynamic Wave routing:

1. From the Data Browser, select the **Options** category and click the button.
2. On the General page of the Simulation Options dialog that appears, select Dynamic Wave as the flow routing method.
3. On the Dynamic Wave page of the dialog, use the settings shown in Figure 2-1810.
4. Click **OK** to close the form and select **Project >> Run Simulation** (or click the button) to re-run the analysis.

![Simulation Options](image)

**Figure 2-18. Dynamic Wave simulation options.**

10 Normally when running a Dynamic Wave analysis, one would also want to reduce the routing time step (on the Time Steps page of the dialog). In this example, we will continue to use a 1-minute time step.
If you look at the Status Report for this run, you will see that there is no longer any junction flooding and that the peak flow carried by conduit C2 has been increased from 3.44 cfs to 4.04 cfs.

2.6 Simulating Water Quality

In the next phase of this tutorial we will add water quality analysis to our example project. SWMM has the ability to analyze the buildup, washoff, transport and treatment of any number of water quality constituents. The steps needed to accomplish this are:

1. Identify the pollutants to be analyzed.
2. Define the categories of land uses that generate these pollutants.
3. Set the parameters of buildup and washoff functions that determine the quality of runoff from each land use.
4. Assign a mixture of land uses to each subcatchment area
5. Define pollutant removal functions for nodes within the drainage system that contain treatment facilities.

We will now apply each of these steps, with the exception of number 5, to our example project.

We will define two runoff pollutants; total suspended solids (TSS), measured as mg/L, and total Lead, measured in ug/L. In addition, we will specify that the concentration of Lead in runoff is a fixed fraction (0.25) of the TSS concentration. To add these pollutants to our project:

1. Under the Quality category in the Data Browser, select the Pollutants sub-category beneath it.
2. Click the button to add a new pollutant to the project.
3. In the Pollutant Editor dialog that appears (see Figure 2-19), enter TSS for the pollutant name and leave the other data fields at their default settings.
4. Click the OK button to close the Editor.
5. Click the button on the Data Browser again to add our next pollutant.
6. In the Pollutant Editor, enter Lead for the pollutant name, select ug/L for the concentration units, enter TSS as the name of the Co-Pollutant, and enter 0.25 as the Co-Fraction value.
7. Click the OK button to close the Editor.

In SWMM, pollutants associated with runoff are generated by specific land uses assigned to subcatchments. In our example, we will define two categories of land uses: Residential and Undeveloped. To add these land uses to the project:

11 Aside from surface runoff, SWMM allows pollutants to be introduced into the nodes of a drainage system through user-defined time series of direct inflows, dry weather inflows, groundwater interflow, and rainfall derived inflow/infiltration
1. Under the **Quality** category in the Data Browser, select the **Land Uses** sub-category and click the + button.

2. In the Land Use Editor dialog that appears (see Figure 2-20), enter **Residential** in the Name field and then click the OK button.

3. Repeat steps 1 and 2 to create the **Undeveloped** land use category.

![Figure 2-19. Pollutant Editor dialog.](image)

![Figure 2-20. Land Use Editor dialog.](image)

Next we need to define buildup and washoff functions for TSS in each of our land use categories. Functions for Lead are not needed since its runoff concentration was defined to be a fixed fraction of the TSS concentration. Normally, defining these functions requires site-specific calibration.

In this example we will assume that suspended solids in Residential areas builds up at a constant rate of 1 pound per acre per day until a limit of 50 lbs per acre is reached. For the Undeveloped area we will assume that buildup is only half as much. For the washoff function, we will assume a constant event mean concentration of 100 mg/L for Residential land and 50 mg/L for Undeveloped land. When runoff occurs, these concentrations will be maintained until the available buildup is exhausted. To define these functions for the Residential land use:
1. Select the **Residential** land use category from the Data Browser and click the button.

2. In the Land Use Editor dialog, move to the Buildup page (see Figure 2-21).

3. Select **TSS** as the pollutant and **POW** (for Power function) as the function type.

4. Assign the function a maximum buildup of **50**, a rate constant of **1.0**, a power of **1** and select **AREA** as the normalizer.

5. Move to the Washoff page of the dialog and select **TSS** as the pollutant, **EMC** as the function type, and enter **100** for the coefficient. Fill the other fields with **0**.

6. Click the **OK** button to accept your entries.

Now do the same for the Undeveloped land use category, except use a maximum buildup of **25**, a buildup rate constant of **0.5**, a buildup power of **1**, and a washoff **EMC** of **50**.

![Land Use Editor dialog](image)

**Figure 2-21. Defining a TSS buildup function for Residential land use.**

The final step in our water quality example is to assign a mixture of land uses to each subcatchment area:

1. Select subcatchment **S1** into the Property Editor.

2. Select the Land Uses property and click the ellipsis button (or press `<Enter>`).

3. In the Land Use Assignment dialog that appears, enter **75** for the % Residential and **25** for the % Undeveloped (see Figure 2-22). Then click the **OK** button to close the dialog.
4. Repeat the same three steps for subcatchment S2.
5. Repeat the same for subcatchment S3, except assign the land uses as 25% Residential and 75% Undeveloped.

![Land Use Assignment dialog.](image)

**Figure 2-22. Land Use Assignment dialog.**

Before we simulate the runoff quantities of TSS and Lead from our study area, an initial buildup of TSS should be defined so it can be washed off during our single rainfall event. We can either specify the number of antecedent dry days prior to the simulation or directly specify the initial buildup mass on each subcatchment. We will use the former method:

1. From the **Options** category of the Data Browser, select the **Dates** sub-category and click the button.
2. In the Simulation Options dialog that appears, enter 5 into the Antecedent Dry Days field.
3. Leave the other simulation options the same as they were for the dynamic wave flow routing we just completed.
4. Click the **OK** button to close the dialog.

Now run the simulation by selecting **Project >> Run Simulation** or by clicking on the Standard Toolbar.

When the run is completed, view its Status Report. Note that two new sections have been added for Runoff Quality Continuity and Quality Routing Continuity. From the Runoff Quality Continuity table we see that there was an initial buildup of 47.5 lbs of TSS on the study area and an additional 1.9 lbs of buildup added during the dry periods of the simulation. Almost 48 lbs were washed off during the rainfall event. The quantity of Lead washed off is a fixed percentage (25% times 0.001 to convert from mg to ug) of the TSS as was specified.
If you plot the runoff concentration of TSS for subcatchment S1 and S3 together on the same time series graph, as in Figure 2-23, you will see the difference in concentrations resulting from the different mix of land uses in these two areas. You can also see that the duration over which pollutants are washed off is much shorter than the duration of the entire runoff hydrograph (i.e., 1 hour versus about 6 hours). This results from having exhausted the available buildup of TSS over this period of time.

![Graph - Subcatchment TSS](image)

**Figure 2-23.** TSS concentration of runoff from selected subcatchments.

2.7 Running a Continuous Simulation

As a final exercise in this tutorial we will demonstrate how to run a long-term continuous simulation using a historical rainfall record and how to perform a statistical frequency analysis on the results. The rainfall record will come from a file named `sta310301.dat` that was included with the example data sets provided with EPA SWMM. It contains several years of hourly rainfall beginning in January 1998. The data are stored in the National Climatic Data Center's DSI 3240 format, which SWMM can automatically recognize.

To run a continuous simulation with this rainfall record:

1. Select the rain gage **Gage1** into the Property Editor.
2. Change the selection of Data Source to **FILE**.
3. Select the File Name data field and click the ellipsis button (or press the <Enter> key) to bring up a standard Windows File Selection dialog.
4. Navigate to the folder where the SWMM example files were stored, select the file named `sta310301.dat`, and click **Open** to select the file and close the dialog.
5. In the Station No. field of the Property Editor enter **310301**.

6. Select the **Options** category in the Data Browser and click the button to bring up the Simulation Options form.

7. On the General page of the form, select **Kinematic Wave** as the Routing Method (this will help speed up the computations).

8. On the Date page of the form, set both the Start Analysis and Start Reporting dates to **01/01/1998**, and set the End Analysis date to **01/01/2000**.

9. On the Time Steps page of the form, set the Routing Time Step to **00:05:00**.

10. Close the Simulation Options form by clicking the **OK** button and start the simulation by selecting **Project >> Run Simulation** (or by clicking on the Standard Toolbar).

After our continuous simulation is complete we can perform a statistical frequency analysis on any of the variables produced as output. For example, to determine the distribution of rainfall volumes within each storm event over the two-year period simulated:

1. Select **Report >> Statistics** or click the button on the Standard Toolbar.

2. In the Statistics Selection dialog that appears, enter the values shown in Figure 2-24.

3. Click the **OK** button to close the form.

The results of this request will be a Statistics Report form (see Figure 2-25) containing three tabbed pages: a Summary page, an Events page containing a rank-ordered listing of each event, and a Histogram page containing a plot of the occurrence frequency versus event magnitude.

The summary page shows that there were a total of 213 rainfall events. The Events page shows that the largest rainfall event had a volume of 3.35 inches and occurred over a 24-hour period. There were no events that matched the 3-inch, 6-hour design storm event used in our previous single-event analysis that had produced some internal flooding. In fact, the status report for this continuous simulation indicates that there were no flooding or surcharge occurrences over the simulation period.
Figure 2-24. Statistics Selection dialog.

Figure 2-25. Statistical Analysis report.
We have only touched the surface of SWMM's capabilities. Some additional features of the program that you will find useful include:

- utilizing additional types of drainage elements, such as storage units, flow dividers, pumps, and regulators, to model more complex types of systems
- using control rules to simulate real-time operation of pumps and regulators
- employing different types of externally-imposed inflows at drainage system nodes, such as direct time series inflows, dry weather inflows, and rainfall-derived infiltration/inflow
- modeling groundwater interflow between aquifers beneath subcatchment areas and drainage system nodes
- modeling snow fall accumulation and melting within subcatchments
- adding calibration data to a project so that simulated results can be compared with measured values
- utilizing a background street, site plan, or topo map to assist in laying out a system's drainage elements and to help relate simulated results to real-world locations.

You can find more information on these and other features in the remaining chapters of this manual.
CHAPTER 3 - SWMM's CONCEPTUAL MODEL

This chapter discusses how SWMM models the objects and operational parameters that constitute a stormwater drainage system. Details about how this information is entered into the program are presented in later chapters. An overview is also given on the computational methods that SWMM uses to simulate the hydrology, hydraulics and water quality transport behavior of a drainage system.

3.1 Introduction

SWMM conceptualizes a drainage system as a series of water and material flows between several major environmental compartments. These compartments and the SWMM objects they contain include:

- The Atmosphere compartment, from which precipitation falls and pollutants are deposited onto the land surface compartment. SWMM uses Rain Gage objects to represent rainfall inputs to the system.
- The Land Surface compartment, which is represented through one or more Subcatchment objects. It receives precipitation from the Atmospheric compartment in the form of rain or snow; it sends outflow in the form of infiltration to the Groundwater compartment and also as surface runoff and pollutant loadings to the Transport compartment.
- The Groundwater compartment receives infiltration from the Land Surface compartment and transfers a portion of this inflow to the Transport compartment. This compartment is modeled using Aquifer objects.
- The Transport compartment contains a network of conveyance elements (channels, pipes, pumps, and regulators) and storage/treatment units that transport water to outfalls or to treatment facilities. Inflows to this compartment can come from surface runoff, groundwater interflow, sanitary dry weather flow, or from user-defined hydrographs. The components of the Transport compartment are modeled with Node and Link objects.

Not all compartments need appear in a particular SWMM model. For example, one could model just the transport compartment, using pre-defined hydrographs as inputs.

3.2 Visual Objects

Figure 3-1 depicts how a collection of SWMM’s visual objects might be arranged together to represent a stormwater drainage system. These objects can be displayed on a map in the SWMM workspace. The following sections describe each of these objects.

35
3.2.1 Rain Gages

Rain Gages supply precipitation data for one or more subcatchment areas in a study region. The rainfall data can be either a user-defined time series or come from an external file. Several different popular rainfall file formats currently in use are supported, as well as a standard user-defined format.

The principal input properties of rain gages include:
- rainfall data type (e.g., intensity, volume, or cumulative volume)
- recording time interval (e.g., hourly, 15-minute, etc.)
- source of rainfall data (input time series or external file)
- name of rainfall data source

3.2.2 Subcatchments

Subcatchments are hydrologic units of land whose topography and drainage system elements direct surface runoff to a single discharge point. The user is responsible for dividing a study area into an appropriate number of subcatchments, and for identifying the outlet point of each subcatchment. Discharge outlet points can be either nodes of the drainage system or other subcatchments.

Subcatchments can be divided into pervious and impervious subareas. Surface runoff can infiltrate into the upper soil zone of the pervious subarea, but not through the impervious subarea. Impervious areas are themselves divided into two subareas - one that contains depression storage and another that does not. Runoff flow from one subarea in a subcatchment can be routed to the other subarea, or both subareas can drain to the subcatchment outlet.
Infiltration of rainfall from the pervious area of a subcatchment into the unsaturated upper soil zone can be described using three different models:

- Horton infiltration
- Green-Ampt infiltration
- SCS Curve Number infiltration

To model the accumulation, re-distribution, and melting of precipitation that falls as snow on a subcatchment, it must be assigned a Snow Pack object. To model groundwater flow between an aquifer underneath the subcatchment and a node of the drainage system, the subcatchment must be assigned a set of Groundwater parameters. Pollutant buildup and washoff from subcatchments are associated with the Land Uses assigned to the subcatchment.

The other principal input parameters for subcatchments include:

- assigned rain gage
- outlet node or subcatchment
- assigned land uses
- tributary surface area
- imperviousness
- slope
- characteristic width of overland flow
- Manning’s n for overland flow on both pervious and impervious areas
- depression storage in both pervious and impervious areas
- percent of impervious area with no depression storage.

### 3.2.3 Junction Nodes

Junctions are drainage system nodes where links join together. Physically they can represent the confluence of natural surface channels, manholes in a sewer system, or pipe connection fittings. External inflows can enter the system at junctions. Excess water at a junction can become partially pressurized while connecting conduits are surcharged and can either be lost from the system or be allowed to pond atop the junction and subsequently drain back into the junction.

The principal input parameters for a junction are:

- invert elevation
- height to ground surface
- ponded surface area when flooded (optional)
- external inflow data (optional).

### 3.2.4 Outfall Nodes

Outfalls are terminal nodes of the drainage system used to define final downstream boundaries under Dynamic Wave flow routing. For other types of flow routing they behave as a junction. Only a single link can be connected to an outfall node.
The boundary conditions at an outfall can be described by any one of the following stage relationships:

- the critical or normal flow depth in the connecting conduit
- a fixed stage elevation
- a tidal stage described in a table of tide height versus hour of the day
- a user-defined time series of stage versus time.

The principal input parameters for outfalls include:

- invert elevation
- boundary condition type and stage description
- presence of a flap gate to prevent backflow through the outfall.

### 3.2.5 Flow Divider Nodes

Flow Dividers are drainage system nodes that divert inflows to a specific conduit in a prescribed manner. A flow divider can have no more than two conduit links on its discharge side. Flow dividers are only active under Kinematic Wave routing and are treated as simple junctions under Dynamic Wave routing.

There are four types of flow dividers, defined by the manner in which inflows are diverted:

- **Cutoff Divider:** diverts all inflow above a defined cutoff value.
- **Overflow Divider:** diverts all inflow above the flow capacity of the non-diverted conduit.
- **Tabular Divider:** uses a table that expresses diverted flow as a function of total inflow.
- **Weir Divider:** uses a weir equation to compute diverted flow.

The flow diverted through a weir divider is computed by the following equation

\[ Q_{\text{div}} = C_w (fH_w)^{1.5} \]

where \( Q_{\text{div}} \) = diverted flow, \( C_w \) = weir coefficient, \( H_w \) = weir height and \( f \) is computed as

\[ f = \frac{Q_{\text{in}} - Q_{\min}}{Q_{\max} - Q_{\min}} \]

where \( Q_{\text{in}} \) is the inflow to the divider, \( Q_{\min} \) is the flow at which diversion begins, and \( Q_{\max} = C_w H_w^{1.5} \). The user-specified parameters for the weir divider are \( Q_{\min}, H_w, \) and \( C_w \).

The principal input parameters for a flow divider are:

- junction parameters (see above)
- name of the link receiving the diverted flow
- method used for computing the amount of diverted flow.
3.2.6 Storage Units

Storage Units are drainage system nodes that provide storage volume. Physically they could represent storage facilities as small as a catchbasin or as large as a lake. The volumetric properties of a storage unit are described by a function or table of surface area versus height.

The principal input parameters for storage units include:
- invert elevation
- maximum depth
- depth-surface area data
- evaporation potential
- ponded surface area when flooded (optional)
- external inflow data (optional).

3.2.7 Conduits

Conduits are pipes or channels that move water from one node to another in the conveyance system. Their cross-sectional shapes can be selected from a variety of standard open and closed geometries as listed in Table 3-1. Irregular natural cross-section shapes are also supported.

SWMM uses the Manning equation to express the relationship between flow rate \( Q \), cross-sectional area \( A \), hydraulic radius \( R \), and slope \( S \) in open channels and partially full closed conduits. For standard U.S. units,

\[
Q = \frac{1.49}{n} AR^{2/3} \sqrt{S}
\]

where \( n \) is the Manning roughness coefficient. For Steady Flow and Kinematic Wave flow routing, \( S \) is interpreted as the conduit slope. For Dynamic Wave flow routing it is the friction slope (i.e., head loss per unit length).

The principal input parameters for conduits are:
- names of the inlet and outlet nodes
- offset heights of the conduit above the inlet and outlet node inverts
- conduit length
- Manning's roughness
- cross-sectional geometry
- entrance/exit losses
- presence of a flap gate to prevent reverse flow.
Table 3-1. Available cross section shapes for conduits

<table>
<thead>
<tr>
<th>Name</th>
<th>Parameters</th>
<th>Shape</th>
<th>Name</th>
<th>Parameters</th>
<th>Shape</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular</td>
<td>Depth</td>
<td></td>
<td>Filled Circular</td>
<td>Depth, Filled Depth</td>
<td></td>
</tr>
<tr>
<td>Rectangular -</td>
<td>Depth, Width</td>
<td></td>
<td>Rectangular –</td>
<td>Depth, Width</td>
<td></td>
</tr>
<tr>
<td>Closed</td>
<td></td>
<td></td>
<td>Open</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Trapezoidal</td>
<td>Depth, Top Width, Side Slopes</td>
<td></td>
<td>Triangular</td>
<td>Depth, Top Width</td>
<td></td>
</tr>
<tr>
<td>Horizontal</td>
<td>Depth</td>
<td></td>
<td>Vertical Ellipse</td>
<td>Depth</td>
<td></td>
</tr>
<tr>
<td>Ellipse</td>
<td></td>
<td></td>
<td>Parabolic</td>
<td>Depth, Top Width</td>
<td></td>
</tr>
<tr>
<td>Arch</td>
<td>Depth</td>
<td></td>
<td>Rectangular-</td>
<td>Depth, Top Width</td>
<td></td>
</tr>
<tr>
<td>Power</td>
<td>Depth, Top Width, Exponent</td>
<td></td>
<td>Triangular</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rectangular-</td>
<td>Depth, Width</td>
<td></td>
<td>Modified</td>
<td>Depth, Width</td>
<td></td>
</tr>
<tr>
<td>Round</td>
<td></td>
<td></td>
<td>Baskethandle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Egg</td>
<td>Depth</td>
<td></td>
<td>Horseshoe</td>
<td>Depth</td>
<td></td>
</tr>
<tr>
<td>Gothic</td>
<td>Depth</td>
<td></td>
<td>Catenary</td>
<td>Depth</td>
<td></td>
</tr>
<tr>
<td>Semi-</td>
<td>Depth</td>
<td></td>
<td>Baskethandle</td>
<td>Depth</td>
<td></td>
</tr>
<tr>
<td>Elliptical</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Semi-</td>
<td>Depth</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Circular</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3.2.8 Pumps

Pumps are links used to lift water to higher elevations. A pump curve describes the relation between a pump's flow rate and conditions at its inlet and outlet nodes. Four different types of pump curves are supported:

**Type 1**
an off-line pump with a wet well where flow increases incrementally with available wet well volume

**Type 2**
an in-line pump where flow increases incrementally with inlet node depth

**Type 3**
an in-line pump where flow varies continuously with head difference between the inlet and outlet nodes

**Type 4**
a variable speed in-line pump where flow varies continuously with inlet node depth

The on/off status of pumps can be controlled dynamically through user-defined Control Rules.

The principal input parameters for a pump include:
- names of its inlet and outlet nodes
- name of its pump curve
- initial on/off status.
3.2.9 Flow Regulators

Flow Regulators are structures or devices used to control and divert flows within a conveyance system. They are typically used to:
- control releases from storage facilities
- prevent unacceptable surcharging
- divert flow to treatment facilities and interceptors

SWMM can model the following types of flow regulators:
- Orifices
- Weirs
- Outlets.

Orifices

Orifices are used to model outlet and diversion structures in drainage systems, which are typically openings in the wall of a manhole, storage facility, or control gate. They are internally represented in SWMM as a link connecting two nodes. An orifice can have either a circular or rectangular shape, be located either at the bottom or along the side of the upstream node, and have a flap gate to prevent backflow.

Orifices can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit node, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

The flow through a fully submerged orifice is computed as

\[ Q = CA\sqrt{2gh} \]

where \( Q \) = flow rate, \( C \) = discharge coefficient, \( A \) = area of orifice opening, \( g \) = acceleration of gravity, and \( h \) = head difference across the orifice. The area of an orifice's opening can be controlled dynamically through user-defined Control Rules.

The principal input parameters for an orifice include:
- names of its inlet and outlet nodes
- configuration (bottom or side)
- shape (circular or rectangular)
- height above the inlet node invert
- discharge coefficient.

Weirs

Weirs, like orifices, are used to model outlet and diversion structures in a drainage system. Weirs are typically located in a manhole, along the side of a channel, or within a storage unit. They are internally represented in SWMM as a link connecting two nodes, where the weir itself is placed at the upstream node. A flap gate can be included to prevent backflow.

Four varieties of weirs are available, each incorporating a different formula for computing flow across the weir as listed in Table 3-2.
Table 3-2. Available types of weirs.

<table>
<thead>
<tr>
<th>Weir Type</th>
<th>Cross Section Shape</th>
<th>Flow Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transverse</td>
<td>Rectangular</td>
<td>$C_w L h^{3/2}$</td>
</tr>
<tr>
<td>Side flow</td>
<td>Rectangular</td>
<td>$C_w L h^{5/3}$</td>
</tr>
<tr>
<td>V-notch</td>
<td>Triangular</td>
<td>$C_w S h^{5/2}$</td>
</tr>
<tr>
<td>Trapezoidal</td>
<td>Trapezoidal</td>
<td>$C_w L h^{3/2} + C_{ws} S h^{5/2}$</td>
</tr>
</tbody>
</table>

$C_w =$ weir discharge coefficient, $L =$ weir length, $S =$ side slope of V-notch or trapezoidal weir, $h =$ head difference across the weir, $C_{ws} =$ discharge coefficient through sides of trapezoidal weir.

Weirs can be used as storage unit outlets under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks that are analyzed with Dynamic Wave flow routing.

The height of the weir crest above the inlet node invert can be controlled dynamically through user-defined Control Rules. This feature can be used to model inflatable dams.

The principal input parameters for a weir include:
- names of its inlet and outlet nodes
- shape and geometry
- crest height above the inlet node invert
- discharge coefficient.

Outlets

Outlets are flow control devices that are typically used to control outflows from storage units. They are used to model special head-discharge relationships that cannot be characterized by pumps, orifices, or weirs. Outlets are internally represented in SWMM as a link connecting two nodes. An outlet can also have a flap gate that restricts flow to only one direction.

Outlets attached to storage units are active under all types of flow routing. If not attached to a storage unit, they can only be used in drainage networks analyzed with Dynamic Wave flow routing.

A user-defined function or table of flow versus head difference determines the flow through an outlet.

The principal input parameters for an outlet include:
- names of its inlet and outlet nodes
- height above the inlet node invert
- function or table containing its head-discharge relationship.
3.2.10 Map Labels

Map Labels are optional text labels added to SWMM's Study Area Map to help identify particular objects or regions of the map. The labels can be drawn in any Windows font, freely edited and be dragged to any position on the map.

3.3 Non-Visual Objects

In addition to physical objects that can be displayed visually on a map, SWMM utilizes several classes of non-visual data objects to describe additional characteristics and processes within a study area.

3.3.1 Climatology

Temperature

Air temperature data are used when simulating snowfall and snowmelt processes during runoff calculations. If these processes are not being simulated then temperature data are not required. Air temperature data can be supplied to SWMM from one of the following sources:

- a user-defined time series of point values (values at intermediate times are interpolated)
- an external climate file containing daily minimum and maximum values (SWMM fits a sinusoidal curve through these values depending on the day of the year).

For user-defined time series, temperatures are in degrees F for US units and degrees C for metric units. The external climate file can also be used to supply evaporation and wind speed as well.

Evaporation

Evaporation can occur for standing water on subcatchment surfaces, for subsurface water in groundwater aquifers, and for water held in storage units. Evaporation rates can be stated as:

- a single constant value
- a set of monthly average values
- a user-defined time series of daily values
- daily values read from an external climate file.

If a climate file is used, then a set of monthly pan coefficients should also be supplied to convert the pan evaporation data to free water-surface values.

Wind Speed

Wind speed is an optional climatic variable that is only used for snowmelt calculations. SWMM can use either a set of monthly average speeds or wind speed data contained in the same climate file used for daily minimum/maximum temperatures.
Snowmelt

Snowmelt parameters are climatic variables that apply across the entire study area when simulating snowfall and snowmelt. They include:

- the air temperature at which precipitation falls as snow
- heat exchange properties of the snow surface
- study area elevation, latitude, and longitude correction

Areal Depletion

Areal depletion refers to the tendency of accumulated snow to melt non-uniformly over the surface of a subcatchment. As the melting process proceeds, the area covered by snow gets reduced. This behavior is described by an Areal Depletion Curve that plots the fraction of total area that remains snow covered against the ratio of the actual snow depth to the depth at which there is 100% snow cover. A typical ADC for a natural area is shown in Figure 3-2. Two such curves can be supplied to SWMM, one for impervious areas and another for pervious areas.

![Figure 3-2. Areal Depletion curve for a natural area.](image)

3.3.2 Snow Packs

Snow Pack objects contain parameters that characterize the buildup, removal, and melting of snow over three types of sub-areas within a subcatchment:

- The Plowable snow pack area consists of a user-defined fraction of the total impervious area. It is meant to represent such areas as streets and parking lots where plowing and snow removal can be done.
- The Impervious snow pack area covers the remaining impervious area of a subcatchment.
- The Pervious snow pack area encompasses the entire pervious area of a subcatchment.
Each of these three areas is characterized by the following parameters:
- Minimum and maximum snow melt coefficients
- Minimum air temperature for snow melt to occur
- Snow depth above which 100% areal coverage occurs
- Initial snow depth
- Initial and maximum free water content in the pack.

In addition, a set of snow removal parameters can be assigned to the Plowable area. These parameters consist of the depth at which snow removal begins and the fractions of snow moved onto various other areas.

Subcatchments are assigned a snow pack object through their Snow Pack property. A single snow pack object can be applied to any number of subcatchments. Assigning a snow pack to a subcatchment simply establishes the melt parameters and initial snow conditions for that subcatchment. Internally, SWMM creates a "physical" snow pack for each subcatchment, which tracks snow accumulation and melting for that particular subcatchment based on its snow pack parameters, its amount of pervious and impervious area, and the precipitation history it sees.

3.3.3 Aquifers

Aquifers are sub-surface groundwater areas used to model the vertical movement of water infiltrating from the subcatchments that lie above them. They also permit the infiltration of groundwater into the drainage system, or exfiltration of surface water from the drainage system, depending on the hydraulic gradient that exists. The same aquifer object can be shared by several subcatchments. Aquifers are only required in models that need to explicitly account for the exchange of groundwater with the drainage system or to establish baseflow and recession curves in natural channels and non-urban systems.

Aquifers are represented using two zones – an unsaturated zone and a saturated zone. Their behavior is characterized using such parameters as soil porosity, hydraulic conductivity, evapotranspiration depth, bottom elevation, and loss rate to deep groundwater. In addition, the initial water table elevation and initial moisture content of the unsaturated zone must be supplied.

Aquifers are connected to subcatchments and to drainage system nodes as defined in a subcatchment's Groundwater Flow property. This property also contains parameters that govern the rate of groundwater flow between the aquifer's saturated zone and the drainage system node.

3.3.4 Unit Hydrographs

Unit Hydrographs (UHs) estimate rainfall-derived infiltration/inflow (RDII) into a sewer system. A UH set contains up to three such hydrographs, one for a short-term response, one for an intermediate-term response, and one for a long-term response. A UH group can have up to 12 UH sets, one for each month of the year. Each UH group is considered as a separate object by SWMM, and is assigned its own unique name along with the name of the rain gage that supplies rainfall data to it.
Each unit hydrograph, as shown in Figure 3-3, is defined by three parameters:

- R: the fraction of rainfall volume that enters the sewer system
- T: the time from the onset of rainfall to the peak of the UH in hours
- K: the ratio of time to recession of the UH to the time to peak

![Figure 3-3. An RDII unit hydrograph](image)

To generate RDII into a drainage system node, the node must identify (through its Inflows property) the UH group and the area of the surrounding sewershed that contributes RDII flow.

An alternative to using unit hydrographs to define RDII flow is to create an external RDII interface file, which contains RDII time series data.

### 3.3.5 Transects

Transects refer to the geometric data that describe how bottom elevation varies with horizontal distance over the cross section of a natural channel or irregular-shaped conduit. Figure 3-4 displays an example of a transect for a natural channel.

![Figure 3-4. Example of a natural channel transect.](image)
Each transect must be given a unique name. Conduits refer to that name to represent their shape. A special Transect Editor is available for editing the station-elevation data of a transect. SWMM internally converts these data into tables of area, top width, and hydraulic radius versus channel depth. In addition, as shown in the diagram above, each transect can have a left and right overbank section whose Manning's roughness can be different from that of the main channel. This feature can provide more realistic estimates of channel conveyance under high flow conditions.

3.3.6 External Inflows

In addition to inflows originating from subcatchment runoff and groundwater, drainage system nodes can receive three other types of external inflows:

- **Direct Inflows** - These are user-defined time series of inflows added directly into a node. They can be used to perform flow and water quality routing in the absence of any runoff computations (as in a study area where no subcatchments are defined).

- **Dry Weather Inflows** - These are continuous inflows that typically reflect the contribution from sanitary sewage in sewer systems or base flows in pipes and stream channels. They are represented by an average inflow rate that can be periodically adjusted on a monthly, daily, and hourly basis by applying Time Pattern multipliers to this average value.

- **Rainfall-Derived Infiltration/Inflow (RDII)** - These are stormwater flows that enter sanitary or combined sewers due to "inflow" from direct connections of downspouts, sump pumps, foundation drains, etc. as well as "infiltration" of subsurface water through cracked pipes, leaky joints, poor manhole connections, etc. RDII can be computed for a given rainfall record based on set of triangular unit hydrographs (UH) that determine a short-term, intermediate-term, and long-term inflow response for each time period of rainfall. Any number of UH sets can be supplied for different sewershed areas and different months of the year. RDII flows can also be specified in an external RDII interface file.

Direct, Dry Weather, and RDII inflows are properties associated with each type of drainage system node (junctions, outfalls, flow dividers, and storage units) and can be specified when nodes are edited. It is also possible to make the outflows generated from an upstream drainage system be the inflows to a downstream system by using interface files. See Section 11.8 for further details.

3.3.7 Control Rules

Control Rules determine how pumps and regulators in the drainage system will be adjusted over the course of a simulation. Some examples of these rules are:

Simple time-based pump control:

```
RULE R1
IF SIMULATION TIME > 8
THEN PUMP 12 STATUS = ON
ELSE PUMP 12 STATUS = OFF
```
Multiple-condition orifice gate control:

RULE R2A
IF NODE 23 DEPTH > 12
AND LINK 165 FLOW > 100
THEN ORIFICE R55 SETTING = 0.5

RULE R2B
IF NODE 23 DEPTH > 12
AND LINK 165 FLOW > 200
THEN ORIFICE R55 SETTING = 1.0

RULE R2C
IF NODE 23 DEPTH <= 12
OR LINK 165 FLOW <= 100
THEN ORIFICE R55 SETTING = 0

Pump station operation:

RULE R3A
IF NODE N1 DEPTH > 5
THEN PUMP N1A STATUS = ON

RULE R3B
IF NODE N1 DEPTH > 7
THEN PUMP N1B STATUS = ON

RULE R3C
IF NODE N1 DEPTH <= 3
THEN PUMP N1A STATUS = OFF
AND PUMP N1B STATUS = OFF

Appendix C.3 describes the control rule format in more detail and the special Editor used to edit them.

3.3.8 Pollutants

SWMM can simulate the generation, inflow and transport of any number of user-defined pollutants. Required information for each pollutant includes:

- pollutant name
- concentration units (i.e., milligrams/liter, micrograms/liter, or counts/liter)
- concentration in rainfall
- concentration in groundwater
- concentration in direct infiltration/inflow
- first-order decay coefficient.

Co-pollutants can also be defined in SWMM. For example, pollutant X can have a co-pollutant Y, meaning that the runoff concentration of X will have some fixed fraction of the runoff concentration of Y added to it.
Pollutant buildup and washoff from subcatchment areas are determined by the land uses assigned to those areas. Input loadings of pollutants to the drainage system can also originate from external time series inflows as well as from dry weather inflows.

3.3.9 Land Uses

Land Uses are categories of development activities or land surface characteristics assigned to subcatchments. Examples of land use activities are residential, commercial, industrial, and undeveloped. Land surface characteristics might include rooftops, lawns, paved roads, undisturbed soils, etc. Land uses are used solely to account for spatial variation in pollutant buildup and washoff rates within subcatchments.

The SWMM user has many options for defining land uses and assigning them to subcatchment areas. One approach is to assign a mix of land uses for each subcatchment, which results in all land uses within the subcatchment having the same pervious and impervious characteristics. Another approach is to create subcatchments that have a single land use classification along with a distinct set of pervious and impervious characteristics that reflects the classification.

The following processes can be defined for each land use category:
- pollutant buildup
- pollutant washoff
- street cleaning.

Pollutant Buildup

Pollutant buildup that accumulates within a land use category is described (or “normalized”) by either a mass per unit of subcatchment area or per unit of curb length. Mass is expressed in pounds for US units and kilograms for metric units. The amount of buildup is a function of the number of preceding dry weather days and can be computed using one of the following functions:

- **Power Function**: Pollutant buildup \( B \) accumulates proportionally to time \( t \) raised to some power, until a maximum limit is achieved,

\[
B = \text{Min}(C_1, C_2 t^{C_3})
\]

where \( C_1 \) = maximum buildup possible (mass per unit of area or curb length), \( C_2 \) = buildup rate constant, and \( C_3 \) = time exponent.

- **Exponential Function**: Buildup follows an exponential growth curve that approaches a maximum limit asymptotically,

\[
B = C_1 (1 - e^{-C_2 t})
\]

where \( C_1 \) = maximum buildup possible (mass per unit of area or curb length) and \( C_2 \) = buildup rate constant (1/days).
Saturation Function: Buildup begins at a linear rate that continuously declines with time until a saturation value is reached,

\[ B = \frac{C_1 t}{C_2 + t} \]

where \( C_1 \) = maximum buildup possible (mass per unit area or curb length) and \( C_2 \) = half-saturation constant (days to reach half of the maximum buildup).

Pollutant Washoff

Pollutant washoff from a given land use category occurs during wet weather periods and can be described in one of the following ways:

- **Exponential Washoff**: The washoff load (W) in units of mass per hour is proportional to the product of runoff raised to some power and to the amount of buildup remaining,

\[ W = C_1 q^{C_2} B \]

where \( C_1 \) = washoff coefficient, \( C_2 \) = washoff exponent, \( q \) = runoff rate per unit area (inches/hour or mm/hour), and \( B \) = pollutant buildup in mass (lbs or kg) per unit area or curb length. Washoff mass units are the same as used to express the pollutant's concentration (milligrams, micrograms, or counts).

- **Rating Curve Washoff**: The rate of washoff \( W \) in mass per second is proportional to the runoff rate raised to some power,

\[ W = C_1 Q^{C_2} \]

where \( C_1 \) = washoff coefficient, \( C_2 \) = washoff exponent, and \( Q \) = runoff rate in user-defined flow units.

- **Event Mean Concentration**: This is a special case of Rating Curve Washoff where the exponent is 1.0 and the coefficient \( C_1 \) represents the washoff pollutant concentration in mass per liter (Note: the conversion between user-defined flow units used for runoff and liters is handled internally by SWMM).

Note that in each case buildup is continuously depleted as washoff proceeds, and washoff ceases when there is no more buildup available.

Washoff loads for a given pollutant and land use category can be reduced by a fixed percentage by specifying a BMP Removal Efficiency that reflects the effectiveness of any BMP controls associated with the land use. It is also possible to use the Event Mean Concentration option by itself, without having to model any pollutant buildup at all.
Street Sweeping

Street sweeping can be used on each land use category to periodically reduce the accumulated buildup of specific pollutants. The parameters that describe street sweeping include:

- days between sweeping
- days since the last sweeping at the start of the simulation
- the fraction of buildup of all pollutants that is available for removal by sweeping
- the fraction of available buildup for each pollutant removed by sweeping

Note that these parameters can be different for each land use, and the last parameter can vary also with pollutant.

3.3.10 Treatment

Removal of pollutants from the flow streams entering any drainage system node is modeled by assigning a set of treatment functions to the node. A treatment function can be any well-formed mathematical expression involving:

- the pollutant concentration of the mixture of all flow streams entering the node (use the pollutant name to represent a concentration)
- the removals of other pollutants (use $R_*$ prefixed to the pollutant name to represent removal)
- any of the following process variables:
  - $FLOW$ for flow rate into node (in user-defined flow units)
  - $DEPTH$ for water depth above node invert (ft or m)
  - $AREA$ for node surface area (ft$^2$ or m$^2$)
  - $DT$ for routing time step (sec)
  - $HRT$ for hydraulic residence time (hours)

The result of the treatment function can be either a concentration (denoted by the letter $C$) or a fractional removal (denoted by $R$). For example, a first-order decay expression for BOD exiting from a storage node might be expressed as:

$$\frac{C}{C_0} = e^{-0.05 \times HRT}$$

or the removal of some trace pollutant that is proportional to the removal of total suspended solids (TSS) could be expressed as:

$$R = 0.75 \times R_{\text{TSS}}$$

3.3.11 Curves

Curve objects are used to describe a functional relationship between two quantities. The following types of curves are available in SWMM:
- Storage - describes how the surface area of a Storage Unit node varies with water depth.
- Diversion - relates diverted outflow to total inflow for a Flow Divider node.
- Tidal - describes how the stage at an Outfall node changes by hour of the day.
- Pump - relates flow through a Pump link to the depth or volume at the upstream node or to the head delivered by the pump.
- Rating - relates flow through an Outlet link to the head difference across the outlet.

Each curve must be given a unique name and can be assigned any number of data pairs.

3.3.12 Time Series

Time Series objects are used to describe how certain object properties vary with time. Time series can be used to describe:
- temperature data
- evaporation data
- rainfall data
- water stage at outfall nodes
- external inflow hydrographs at drainage system nodes
- external inflow pollutographs at drainage system nodes.

Each time series must be given a unique name and can be assigned any number of time-value data pairs. Time can be specified either as hours from the start of a simulation or as an absolute date and time-of-day.

⚠️ For rainfall time series, it is only necessary to enter periods with non-zero rainfall amounts. SWMM interprets the rainfall value as a constant value lasting over the recording interval specified for the rain gage that utilizes the time series. For all other types of time series, SWMM uses interpolation to estimate values at times that fall in between the recorded values.

⚠️ For times that fall outside the range of the time series, SWMM will use a value of 0 for rainfall and external inflow time series, and either the first or last series value for temperature, evaporation, and water stage time series.

3.3.13 Time Patterns

Time Patterns allow external Dry Weather Flow (DWF) to vary in a periodic fashion. They consist of a set of adjustment factors applied as multipliers to a baseline DWF flow rate or pollutant concentration. The different types of time patterns include:
- Monthly - one multiplier for each month of the year
- Daily - one multiplier for each day of the week
- Hourly - one multiplier for each hour from 12 AM to 11 PM
- Weekend - hourly multipliers for weekend days

Each Time Pattern must have a unique name and there is no limit on the number of patterns that can be created. Each dry weather inflow (either flow or quality) can have up to four patterns associated with it, one for each type listed above.

### 3.4 Computational Methods

SWMM is a physically based, discrete-time simulation model. It employs principles of conservation of mass, energy, and momentum wherever appropriate. This section briefly describes the methods SWMM uses to model stormwater runoff quantity and quality through the following physical processes:
- Surface Runoff
- Infiltration
- Groundwater
- Snow Melt
- Flow Routing
- Surface Ponding
- Water Quality Routing

#### 3.4.1 Surface Runoff

The conceptual view of surface runoff used by SWMM is illustrated in Figure 3-5 below. Each subcatchment surface is treated as a nonlinear reservoir. Inflow comes from precipitation and any designated upstream subcatchments. There are several outflows, including infiltration, evaporation, and surface runoff. The capacity of this "reservoir" is the maximum depression storage, which is the maximum surface storage provided by ponding, surface wetting, and interception. Surface runoff per unit area, Q, occurs only when the depth of water in the "reservoir" exceeds the maximum depression storage, dp, in which case the outflow is given by Manning's equation. Depth of water over the subcatchment (d in feet) is continuously updated with time (t in seconds) by solving numerically a water balance equation over the subcatchment.

![Figure 3-5. Conceptual view of surface runoff.](image)
Infiltration

Infiltration is the process of rainfall penetrating the ground surface into the unsaturated soil zone of pervious subcatchments areas. SWMM offers three choices for modeling infiltration:

**Horton's Equation**
This method is based on empirical observations showing that infiltration decreases exponentially from an initial maximum rate to some minimum rate over the course of a long rainfall event. Input parameters required by this method include the maximum and minimum infiltration rates, a decay coefficient that describes how fast the rate decreases over time, and a time it takes a fully saturated soil to completely dry.

**Green-Ampt Method**
This method for modeling infiltration assumes that a sharp wetting front exists in the soil column, separating soil with some initial moisture content below from saturated soil above. The input parameters required are the initial moisture deficit of the soil, the soil's hydraulic conductivity, and the suction head at the wetting front.

**Curve Number Method**
This approach is adopted from the NRCS (SCS) Curve Number method for estimating runoff. It assumes that the total infiltration capacity of a soil can be found from the soil's tabulated Curve Number. During a rain event this capacity is depleted as a function of cumulative rainfall and remaining capacity. The input parameters for this method are the curve number, the soil's hydraulic conductivity (used to estimate a minimum separation time for distinct rain events), and a time it takes a fully saturated soil to completely dry.

### 3.4.2 Groundwater

Figure 3-6 is a definitional sketch of the two-zone groundwater model that is used in SWMM. The upper zone is unsaturated with a variable moisture content of $\theta$. The lower zone is fully saturated and therefore its moisture content is fixed at the soil porosity $\phi$. The fluxes shown in the figure, expressed as volume per unit area per unit time, consist of the following:

- $f_I$: infiltration from the surface
- $f_{EU}$: evapotranspiration from the upper zone which is a fixed fraction of the un-used surface evaporation
- $f_U$: percolation from the upper to lower zone which depends on the upper zone moisture content $\theta$ and depth $d_U$
- $f_{EL}$: evapotranspiration from the lower zone, which is a function of the depth of the upper zone $d_U$
- $f_L$: percolation from the lower zone to deep groundwater which depends on the lower zone depth $d_L$
- $f_G$: lateral groundwater interflow to the drainage system, which depends on the lower zone depth $d_L$ as well as the depth in the receiving channel or node.
After computing the water fluxes that exist during a given time step, a mass balance is written for the change in water volume stored in each zone so that a new water table depth and unsaturated zone moisture content can be computed for the next time step.

3.4.3 Snowmelt

The snowmelt routine in SWMM is a part of the runoff modeling process. It updates the state of the snow packs associated with each subcatchment by accounting for snow accumulation, snow redistribution by areal depletion and removal operations, and snow melt via heat budget accounting. Any snowmelt coming off the pack is treated as an additional rainfall input onto the subcatchment.

At each runoff time step the following computations are made:

1. Air temperature and melt coefficients are updated according to the calendar date.
2. Any precipitation that falls as snow is added to the snow pack.
3. Any excess snow depth on the plowable area of the pack is redistributed according to the removal parameters established for the pack.
4. Areal coverages of snow on the impervious and pervious areas of the pack are reduced according to the Areal Depletion Curves defined for the study area.
5. The amount of snow in the pack that melts to liquid water is found using:
   - a heat budget equation for periods with rainfall, where melt rate increases with increasing air temperature, wind speed, and rainfall intensity
   - a degree-day equation for periods with no rainfall, where melt rate equals the product of a melt coefficient and the difference between the air temperature and the pack's base melt temperature.
6. If no melting occurs, the pack temperature is adjusted up or down based on the product of the difference between current and past air temperatures and an adjusted melt coefficient. If melting occurs, the temperature of the pack is increased by the
equivalent heat content of the melted snow, up to the base melt temperature. Any remaining melt liquid beyond this is available to runoff from the pack.

7. The available snowmelt is then reduced by the amount of free water holding capacity remaining in the pack. The remaining melt is treated the same as an additional rainfall input onto the subcatchment.

3.4.4 Flow Routing

Flow routing within a conduit link in SWMM is governed by the conservation of mass and momentum equations for gradually varied, unsteady flow (i.e., the Saint Venant flow equations). The SWMM user has a choice on the level of sophistication used to solve these equations:

- **Steady Flow Routing**
- **Kinematic Wave Routing**
- **Dynamic Wave Routing**

**Steady Flow Routing**

Steady Flow routing represents the simplest type of routing possible (actually no routing) by assuming that within each computational time step flow is uniform and steady. Thus it simply translates inflow hydrographs at the upstream end of the conduit to the downstream end, with no delay or change in shape. The Manning equation is used to relate flow rate to flow area (or depth).

This type of routing cannot account for channel storage, backwater effects, entrance/exit losses, flow reversal or pressurized flow. It can only be used with dendritic conveyance networks, where each node has only a single outflow link (unless the node is a divider in which case two outflow links are required). This form of routing is insensitive to the time step employed and is really only appropriate for preliminary analysis using long-term continuous simulations.

**Kinematic Wave Routing**

This routing method solves the continuity equation along with a simplified form of the momentum equation in each conduit. The latter requires that the slope of the water surface equal the slope of the conduit.

The maximum flow that can be conveyed through a conduit is the full-flow Manning equation value. Any flow in excess of this entering the inlet node is either lost from the system or can pond atop the inlet node and be re-introduced into the conduit as capacity becomes available.

Kinematic wave routing allows flow and area to vary both spatially and temporally within a conduit. This can result in attenuated and delayed outflow hydrographs as inflow is routed through the channel. However this form of routing cannot account for backwater effects, entrance/exit losses, flow reversal, or pressurized flow, and is also restricted to dendritic network layouts. It can usually maintain numerical stability with moderately large time steps, on the order of 5 to 15 minutes. If the aforementioned effects are not
expected to be significant then this alternative can be an accurate and efficient routing method, especially for long-term simulations.

**Dynamic Wave Routing**

Dynamic Wave routing solves the complete one-dimensional Saint Venant flow equations and therefore produces the most theoretically accurate results. These equations consist of the continuity and momentum equations for conduits and a volume continuity equation at nodes.

With this form of routing it is possible to represent pressurized flow when a closed conduit becomes full, such that flows can exceed the full-flow Manning equation value. Flooding occurs when the water depth at a node exceeds the maximum available depth, and the excess flow is either lost from the system or can pond atop the node and re-enter the drainage system.

Dynamic wave routing can account for channel storage, backwater, entrance/exit losses, flow reversal, and pressurized flow. Because it couples together the solution for both water levels at nodes and flow in conduits it can be applied to any general network layout, even those containing multiple downstream diversions and loops. It is the method of choice for systems subjected to significant backwater effects due to downstream flow restrictions and with flow regulation via weirs and orifices. This generality comes at a price of having to use much smaller time steps, on the order of a minute or less (SWMM will automatically reduce the user-defined maximum time step as needed to maintain numerical stability).

**3.4.5 Surface Ponding**

Normally in flow routing, when the flow into a junction exceeds the capacity of the system to transport it further downstream, the excess volume overflows the system and is lost. An option exists to have instead the excess volume be stored atop the junction, in a ponded fashion, and be reintroduced into the system as capacity permits. Under Steady and Kinematic Wave flow routing, the ponded water is stored simply as an excess volume. For Dynamic Wave routing, which is influenced by the water depths maintained at nodes, the excess volume is assumed to pond over the node with a constant surface area. This amount of surface area is an input parameter supplied for the junction.

Alternatively, the user may wish to represent the surface overflow system explicitly. In open channel systems this can include road overflows at bridges or culvert crossings as well as additional floodplain storage areas. In closed conduit systems, surface overflows may be conveyed down streets, alleys, or other surface routes to the next available stormwater inlet or open channel. Overflows may also be impounded in surface depressions such as parking lots, back yards or other areas.

**3.4.6 Water Quality Routing**

Water quality routing within conduit links assumes that the conduit behaves as a continuously stirred tank reactor (CSTR). Although a plug flow reactor assumption might be more realistic, the differences will be small if the travel time through the conduit is on the same order as the routing time step. The concentration of a constituent exiting the
conduit at the end of a time step is found by integrating the conservation of mass equation, using average values for quantities that might change over the time step such as flow rate and conduit volume.

Water quality modeling within storage unit nodes follows the same approach used for conduits. For other types of nodes that have no volume, the quality of water exiting the node is simply the mixture concentration of all water entering the node.
This chapter discusses the essential features of SWMM’s workspace. It describes the main menu bar, the tool and status bars, and the three windows used most often – the Study Area Map, the Browser, and the Property Editor. It also shows how to set program preferences.

4.1 Overview

The EPA SWMM main window is pictured below. It consists of the following user interface elements: a Main Menu, several Toolbars, a Status Bar, the Study Area Map window, a Browser panel, and a Property Editor window. A description of each of these elements is provided in the sections that follow.
4.2 Main Menu

The Main Menu located across the top of the EPA SWMM main window contains a collection of menus used to control the program. These include:
- File Menu
- Edit Menu
- View Menu
- Project Menu
- Report Menu
- Window Menu
- Help Menu

File Menu

The File Menu contains commands for opening and saving data files and for printing:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Creates a new SWMM project</td>
</tr>
<tr>
<td>Open</td>
<td>Opens an existing project</td>
</tr>
<tr>
<td>Reopen</td>
<td>Reopens a recently used project</td>
</tr>
<tr>
<td>Save</td>
<td>Saves the current project</td>
</tr>
<tr>
<td>Save As</td>
<td>Saves the current project under a different name</td>
</tr>
<tr>
<td>Export</td>
<td>Exports study area map to a file in a variety of formats</td>
</tr>
<tr>
<td>Page Setup</td>
<td>Sets page margins and orientation for printing</td>
</tr>
<tr>
<td>Print Preview</td>
<td>Previews a printout of the currently active view (map, report, graph, or table)</td>
</tr>
<tr>
<td>Print</td>
<td>Prints the current view</td>
</tr>
<tr>
<td>Preferences</td>
<td>Sets program preferences</td>
</tr>
<tr>
<td>Exit</td>
<td>Exits EPA SWMM</td>
</tr>
</tbody>
</table>
### Edit Menu

The Edit Menu contains commands for editing and copying:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Copy To</td>
<td>Copies the currently active view (map, report, graph or table) to the clipboard or to a file</td>
</tr>
<tr>
<td>Select Object</td>
<td>Enables the user to select an object on the map</td>
</tr>
<tr>
<td>Select Vertex</td>
<td>Enables the user to select the vertex of a subcatchment or link</td>
</tr>
<tr>
<td>Select Region</td>
<td>Enables the user to delineate a region on the map for selecting multiple objects</td>
</tr>
<tr>
<td>Select All</td>
<td>Selects all objects when the map is the active window or all cells of a table when a tabular report is the active window</td>
</tr>
<tr>
<td>Find</td>
<td>Locates a specific object on the map</td>
</tr>
<tr>
<td>Group Edit</td>
<td>Edits a property for the group of objects that fall within the outlined region of the map</td>
</tr>
<tr>
<td>Group Delete</td>
<td>Deletes a group of objects that fall within the outlined region of the map</td>
</tr>
</tbody>
</table>

### View Menu

The View Menu contains commands for viewing the Study Area Map:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensions</td>
<td>Sets reference coordinates and map units for the study area map</td>
</tr>
<tr>
<td>Backdrop</td>
<td>Allows a backdrop image to be added, positioned, and viewed</td>
</tr>
<tr>
<td>Pan</td>
<td>Pans across the map</td>
</tr>
<tr>
<td>Zoom In</td>
<td>Zooms in on the map</td>
</tr>
<tr>
<td>Zoom Out</td>
<td>Zooms out on the map</td>
</tr>
<tr>
<td>Full Extent</td>
<td>Redraws the map at full extent</td>
</tr>
<tr>
<td>Query</td>
<td>Highlights objects on the map that meet specific criteria</td>
</tr>
<tr>
<td>Overview</td>
<td>Toggles the display of the Overview Map</td>
</tr>
<tr>
<td>Objects</td>
<td>Toggles display of classes of objects on the map</td>
</tr>
<tr>
<td>Legends</td>
<td>Controls display of the map legends</td>
</tr>
<tr>
<td>Toolbars</td>
<td>Toggles display of tool bars</td>
</tr>
<tr>
<td>Options</td>
<td>Sets map appearance options</td>
</tr>
</tbody>
</table>
Project Menu

The Project menu contains commands related to the current project being analyzed:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Summary</td>
<td>Lists the number of each type of object in the project</td>
</tr>
<tr>
<td>Details</td>
<td>Shows a detailed listing of all project data</td>
</tr>
<tr>
<td>Defaults</td>
<td>Edits a project’s default properties</td>
</tr>
<tr>
<td>Calibration Data</td>
<td>Registers files containing calibration data with the project</td>
</tr>
<tr>
<td>Run Simulation</td>
<td>Runs a simulation</td>
</tr>
</tbody>
</table>

Report Menu

The Report menu contains commands used to report analysis results in different formats:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status</td>
<td>Displays a status report for the most recent simulation run</td>
</tr>
<tr>
<td>Graph</td>
<td>Displays simulation results in graphical form</td>
</tr>
<tr>
<td>Table</td>
<td>Displays simulation results in tabular form</td>
</tr>
<tr>
<td>Statistics</td>
<td>Displays a statistical analysis of simulation results</td>
</tr>
<tr>
<td>Options</td>
<td>Controls the display style of the currently active graph</td>
</tr>
</tbody>
</table>

Window Menu

The Window Menu contains commands for arranging and selecting windows within the SWMM workspace:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cascade</td>
<td>Arranges windows in cascaded style, with the study area map filling the entire display area</td>
</tr>
<tr>
<td>Tile</td>
<td>Minimizes the study area map and tiles the remaining windows vertically in the display area</td>
</tr>
<tr>
<td>Close All</td>
<td>Closes all open windows except for the study area map</td>
</tr>
<tr>
<td>Window List</td>
<td>Lists all open windows; the currently selected window has the focus and is denoted with a check mark</td>
</tr>
</tbody>
</table>
Help Menu

The Help Menu contains commands for getting help in using EPA SWMM:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Help Topics</td>
<td>Displays the Help system's Table of Contents</td>
</tr>
<tr>
<td>How Do I</td>
<td>Displays a list of topics covering the most common operations</td>
</tr>
<tr>
<td>Measurement Units</td>
<td>Shows measurement units for all of SWMM’s parameters</td>
</tr>
<tr>
<td>Error Messages</td>
<td>Lists the meaning of all error messages</td>
</tr>
<tr>
<td>Tutorial</td>
<td>Presents a short tutorial introducing the user to EPA SWMM</td>
</tr>
<tr>
<td>About</td>
<td>Lists information about the version of EPA SWMM being used</td>
</tr>
</tbody>
</table>

4.3 Toolbars

Toolbars provide shortcuts to commonly used operations. There are four such toolbars:

- Standard Toolbar
- Map Toolbar
- Object Toolbar
- Animator Toolbar

Except for the Animator Toolbar, all toolbars can be docked underneath the Main Menu bar, docked on the right side of the Browser Panel or dragged to any location on the EPA SWMM workspace. When undocked, they can also be re-sized.

Toolbars can be made visible or invisible by selecting View >> Toolbars from the Main Menu.

Standard Toolbar

The Standard Toolbar contains buttons for the following commonly used commands:

- Creates a new project (File >> New)
- Opens an existing project (File >> Open)
- Saves the current project (File >> Save)
- Prints the currently active window (File >> Print)
- Copies selection to the clipboard or to a file (Edit >> Copy To)
- Finds a specific item on the map (Edit >> Find)
- Runs a simulation (Project >> Run Simulation)
- Makes a visual query of the study area map (View >> Query)
Creates a new graph of simulation results (**Report >> Graph**)

Creates a new table of simulation results (**Report >> Table**)

∑ Performs a statistical analysis of simulation results (**Report >> Statistics**)

Modifies display options for the currently active view (**View >> Options** or **Report >> Options**)

Arranges windows in cascaded style, with the study area map filling the entire display area (**Window >> Cascade**)

**Map Toolbar**

The Map Toolbar contains the following buttons for viewing the study area map:

- Selects an object on the map (**Edit >> Select Object**)
- Selects link or subcatchment vertex points (**Edit >> Select Vertex**)
- Selects a region on the map (**Edit >> Select Region**)
- Pans across the map (**View >> Pan**)
- Zooms in on the map (**View >> Zoom In**)
- Zooms out on the map (**View >> Zoom Out**)
- Draws map at full extent (**View >> Full Extent**)

**Object Toolbar**

The Object Toolbar contains buttons for adding objects to the study area map:

- Adds a rain gage to the map
- Adds a subcatchment to the map
- Adds a junction node to the map
- Adds an outfall node to the map
- Adds a flow divider node to the map
- Adds a storage unit node to the map
- Adds a conduit link to the map
- Adds a pump link to the map
- Adds an orifice link to the map
- Adds a weir link to the map
- Adds an outlet link to the map
- Adds a label to the map
Animator Toolbar

The Animator Toolbar contains controls for animating the Study Area Map and all Profile Plots through time (i.e., updating map color-coding and hydraulic grade line profile depths as the simulation time clock is automatically moved forward or back). The toolbar and the meaning of its controls are shown below.

The Animator toolbar is initially hidden on program startup. To make the toolbar visible, select View >> Toolbars >> Animator from the Main Menu.

4.4 Status Bar

The Status Bar appears at the bottom of SWMM's Main Window and is divided into five sections:

**Auto-Length**
Indicates whether the automatic computation of conduit lengths and subcatchment areas is turned on or off. Right-click over this section to toggle this feature on/off.

**Flow Units**
Displays the current flow units that are in effect.

**Run Status**
A faucet icon shows:
- no running water if simulation results are not available,
- running water when simulation results are available,
- a broken faucet when simulation results are available but may be invalid because project data have been modified.

**Zoom Level**
Displays the current zoom level for the map (100% is full-scale).

**XY Location**
Displays the map coordinates of the current position of the mouse pointer.
4.5 **Study Area Map**

The Study Area Map (shown below) provides a planar schematic diagram of the objects comprising a drainage system. Its pertinent features are as follows:

- The location of objects and the distances between them do not necessarily have to conform to their actual physical scale.
- Selected properties of these objects, such as water quality at nodes or flow velocity in links, can be displayed by using different colors. The color-coding is described in a Legend, which can be edited.
- New objects can be directly added to the map and existing objects can be selected for editing, deleting, and repositioning.
- A backdrop drawing (such as a street or topographic map) can be placed behind the network map for reference.
- The map can be zoomed to any scale and panned from one position to another.
- Nodes and links can be drawn at different sizes, flow direction arrows added, and object symbols, ID labels and numerical property values displayed.
- The map can be printed, copied onto the Windows clipboard, or exported as a DXF file or Windows metafile.

4.6 **Data Browser**

The Data Browser panel (shown below) appears when the Data tab on the left panel of the SWMM’s workspace is selected. It provides access to all of the data objects in a project.
The Categories list box displays the various categories of data objects available to a SWMM project. The lower list box lists the name of each individual object of the currently selected data category.

The buttons between the two list boxes of the Data Browser are used as follows:
- adds a new object
- deletes the selected object
- edits the selected object
- moves the selected object up one position
- moves the selected object down one position
- sorts the objects in ascending order

Selections made in the Data Browser are coordinated with objects highlighted on the Study Area Map, and vice versa. For example, selecting a conduit in the Data Browser will cause that conduit to be highlighted on the map, while selecting it on the map will cause it to become the selected object in the Data Browser.

4.7 Map Browser

The Map Browser panel (shown below) appears when the Map tab on the left panel of the SWMM’s workspace is selected. It controls the mapping themes and time periods to be viewed on the Study Area Map.

The selections available on the Map Browser are as follows:

Subcatch View - selects the theme to display for the subcatchment areas shown on the Study Area Map.

Node View - selects the theme to display for the drainage system nodes shown on the Study Area Map.

Link View - selects the theme to display for the drainage system links shown on the Study Area Map.

Date - selects the day for which simulation results will be viewed.

Time - selects the hour of the current day for which simulation results will be viewed.

Elapsed Time - selects the elapsed time from the start of the simulation for which results will be viewed.
4.8 Property Editor

The Property Editor (shown to the right) is used to edit the properties of data objects that can appear on the Study Area Map. It is invoked when one of these objects is selected (either on the Study Area Map or in the Data Browser) and double-clicked or when the Data Browser's Edit button is clicked.

Key features of the Property Editor include:

- The Editor is a grid with two columns - one for the property's name and the other for its value.
- The columns can be re-sized by re-sizing the header at the top of the Editor with the mouse.
- A hint area is displayed at the bottom of the Editor with an expanded description of the property being edited. The size of this area can be adjusted by dragging the splitter bar located just above it.
- The Editor window can be moved and re-sized via the normal Windows operations.
- An asterisk next to a property name means that it is a required property -- its value cannot be left blank.
- Depending on the property, the value field can be one of the following:
  - a text box in which you enter a value
  - a dropdown combo box from which you select a value from a list of choices
  - a dropdown combo box in which you can enter a value or select from a list of choices
  - an ellipsis button which you click to bring up a specialized editor.
- The property field in the Editor that currently has the focus will be highlighted with a white background.
- Both the mouse and the Up and Down arrow keys on the keyboard can be used to move between property fields.
- To begin editing the property with the focus, either begin typing a value or hit the Enter key.
- To have the program accept edits made in a property field, either press the Enter key or move to another property. To cancel the edits, press the Esc key.
- The Property Editor can be hidden by clicking the button in the upper right corner of its title bar.
4.9 Setting Program Preferences

Program preferences allow one to customize certain program features. To set program preferences, select **Preferences** from the **File** menu. A Preferences dialog form will appear containing two tabbed pages – one for General Preferences and one for Number Formats.
General Preferences

The following preferences can be set on the General Preferences page of the Preferences dialog:

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bold Fonts</td>
<td>Check to use bold fonts in all windows</td>
</tr>
<tr>
<td>Large Fonts</td>
<td>Check to use large size fonts in all windows</td>
</tr>
<tr>
<td>Blinking Map Highlighter</td>
<td>Check to make the selected object on the study area map blink on and off</td>
</tr>
<tr>
<td>Flyover Map Labeling</td>
<td>Check to display the ID label and current theme value in a hint-style box whenever the mouse is placed over an object on the study area map</td>
</tr>
<tr>
<td>Confirm Deletions</td>
<td>Check to display a confirmation dialog box before deleting any object</td>
</tr>
<tr>
<td>Automatic Backup File</td>
<td>Check to save a backup copy of a newly opened project to disk named with a .bak extension</td>
</tr>
<tr>
<td>Clear File List</td>
<td>Check to clear the list of most recently used files that appears when File &gt;&gt; Reopen is selected from the Main Menu</td>
</tr>
<tr>
<td>Temporary Directory</td>
<td>Name of the directory (folder) where EPA SWMM writes its temporary files</td>
</tr>
</tbody>
</table>

⚠️ The Temporary Directory must be a file directory (folder) where the user has write privileges and must have sufficient space to temporarily store files which can easily grow to several tens of megabytes for larger study areas and simulation runs. The original default is the folder where Windows writes its temporary files.

Number Format Preferences

The Number Formats page of the Preferences dialog controls the number of decimal places displayed when simulation results are reported. Use the dropdown list boxes to select a specific Subcatchment, Node or Link parameter, and then use the edit boxes next to them to select the number of decimal places to use when displaying computed results for the parameter. Note that the number of decimal places displayed for any particular input design parameter, such as slope, diameter, length, etc. is whatever the user enters.
CHAPTER 5 - WORKING WITH PROJECTS

Project files contain all of the information used to model a study area. They are usually named with a .INP extension. This section describes how to create, open, and save EPA SWMM projects as well as setting their default properties.

5.1 Creating a New Project

To create a new project:

1. Select File >> New from the Main Menu or click on the Standard Toolbar.
2. You will be prompted to save the existing project (if changes were made to it) before the new project is created.
3. A new, unnamed project is created with all options set to their default values.

A new project is automatically created whenever EPA SWMM first begins.

⚠️ If you are going to use a backdrop image with automatic area and length calculation, then it is recommended that you set the map dimensions immediately after creating the new project (see Setting the Map's Dimensions).

5.2 Opening an Existing Project

To open an existing project stored on disk:

1. Either select File >> Open from the Main Menu or click on the Standard Toolbar.
2. You will be prompted to save the current project (if changes were made to it).
3. Select the file to open from the Open File dialog form that will appear.
4. Click Open to open the selected file.

To open a project that was worked on recently:

1. Select File >> Reopen from the Main Menu.
2. Select a file from the list of recently used files to open.

5.3 Saving a Project

To save a project under its current name either select File >> Save from the Main Menu or click on the Standard Toolbar.
To save a project using a different name:

1. Select **File >> Save As** from the Main Menu.
2. A standard File Save dialog form will appear from which you can select the folder and name that the project should be saved under.

### 5.4 Setting Project Defaults

Each project has a set of default values that are used unless overridden by the SWMM user. These values fall into three categories:

- Default ID labels (labels used to identify nodes and links when they are first created)
- Default subcatchment properties (e.g., area, width, slope, etc.)
- Default node/link properties (e.g., node invert, conduit length, routing method).

To set default values for a project:

1. Select **Project >> Defaults** from the Main Menu.
2. A Project Defaults dialog will appear with three pages, one for each category listed above.
3. Check the box in the lower right of the dialog form if you want to save your choices for use in all new future projects as well.
4. Click **OK** to accept your choice of defaults.

The specific items for each category of defaults will be discussed next.

### Default ID Labels

The ID Labels page of the Project Defaults dialog form is used to determine how SWMM will assign default ID labels for the visual project components when they are first created. For each type of object you can enter a label prefix in the corresponding entry field or leave the field blank if an object's default name will simply be a number. In the last field you can enter an increment to be used when adding a numerical suffix to the default label. As an example, if C were used as a prefix for Conduits along with an increment of 5, then as conduits are created they receive default names of C5, C10, C15 and so on. An object's default name can be changed by using the Property Editor for visual objects or the object-specific editor for non-visual objects.
Default Subcatchment Properties

The Subcatchment page of the Project Defaults dialog sets default property values for newly created subcatchments. These properties include:
- Subcatchment Area
- Characteristic Width
- Slope
- % Impervious
- Impervious Area Manning’s N
- Pervious Area Manning’s N
- Impervious Area Depression Storage
- Pervious Area Depression Storage
- % of Impervious Area with No Depression Storage
- Infiltration Method

The default properties of a subcatchment can be modified later by using the Property Editor.

Default Node/Link Properties

The Nodes/Links page of the Project Defaults dialog sets default property values for newly created nodes and links. These properties include:
- Node Invert Elevation
- Node Maximum Depth
- Conduit Length
- Conduit Shape and Size
- Conduit Roughness
- Flow Units
- Routing Method

These default properties can be modified later by using the Property Editor.

⚠️ The choice of flow units determines whether US or metric units are used for all other quantities. Default values are not automatically adjusted when the unit system is changed from US to metric (or vice versa).

5.5 Calibration Data

SWMM can compare the results of a simulation with measured field data in its Time Series Plots, which are discussed in section 9.3. Before SWMM can use such calibration data they must be entered into a specially formatted text file and registered with the project.

Calibration Files

Calibration Files contain measurements of a single parameter at one or more locations that can be compared with simulated values in Time Series Plots. Separate files can be used for each of the following parameters:
- Subcatchment Runoff
- Subcatchment Pollutant Washoff
- Node Depth
- Node Inflow
- Node Water Quality
- Link Flow

The format of the file is described in Section 11.5.

Registering Calibration Data

To register calibration data residing in a Calibration File:

1. Select Project >> Calibration Data from the Main Menu.
2. In the Calibration Data dialog form shown below, click in the box next to the parameter (e.g., node depth, link flow, etc.) whose calibration data will be registered.
3. Either type in the name of a Calibration File for this parameter or click the Browse button to search for it.
4. Click the Edit button if you want to open the Calibration File in Windows NotePad for editing.
5. Repeat steps 2 - 4 for any other parameters that have calibration data.
6. Click OK to accept your selections.
5.6 Viewing All Project Data

A listing of all project data (with the exception of map coordinates) can be viewed in a non-editable window, formatted for input to SWMM's computational engine. This can be useful for checking data consistency and to make sure that no key components are missing. To view such a listing, select Project >> Details from the Main Menu. The format of the data in this listing is the same as that used when the file is saved to disk. It is described in detail in Appendix D.2.
CHAPTER 6 - WORKING WITH OBJECTS

SWMM uses various types of objects to model a drainage area and its conveyance system. This section describes how these objects can be created, selected, edited, deleted, and repositioned.

6.1 Types of Objects

SWMM contains both physical objects that can appear on its Study Area Map, and non-physical objects that encompass design, loading, and operational information. These objects, which are listed in the Data Browser and were described in Chapter 3, consist of the following:

- Project Title/Notes
- Simulation Options
- Climatology
- Rain Gages
- Subcatchments
- Aquifers
- Snow Packs
- RDII Hydrographs
- Nodes
- Links
- Transects
- Controls
- Pollutants
- Land Uses
- Curves
- Time Series
- Time Patterns
- Map Labels

6.2 Adding Objects

Visual objects are those that can appear on the Study Area Map and include Rain Gages, Subcatchments, Nodes, Links, and Map Labels. With the exception of Map Labels, there are two ways to add these objects into a project:

- selecting the object’s icon from the Object Toolbar and then clicking on the map,
- selecting the object’s category in the Data Browser and clicking the Browser’s button.

The first method makes the object appear on the map and is therefore recommended. With the second method, the object will not appear on the map until X,Y coordinates are entered manually by editing the object’s properties. What follows are more specific instructions for adding each type of object to a project.

Adding a Rain Gage

To add a Rain Gage using the Object Toolbar:

1. Click on the toolbar.
2. Move the mouse to the desired location on the map and click.
To add a Rain Gage using the Data Browser:
1. Select Rain Gages from the list of categories.
2. Click the button.
3. Enter the rain gage's X and Y coordinates in the Property Editor if you want
   it to appear on the study area map.

Adding a Subcatchment

To add a Subcatchment using the Object Toolbar:
1. Click on the toolbar.
2. Use the mouse to draw a polygon outline of the subcatchment on the map:
   ▪ left-click at each vertex
   ▪ right-click to close the polygon
   ▪ press the <Esc> key if you wish to cancel the action.

To add a Subcatchment using the Data Browser:
1. Select Subcatchments from the list of object categories.
2. Click the button.
3. Enter the X and Y coordinates of the subcatchment's centroid in the Property Editor if you want
   it to appear on the study area map.

Adding a Node

To add a Node using the Object Toolbar:
1. Click the button for the type of node to add (if its not already depressed):
   ○ for a junction
   ▽ for an outfall
   ◊ for a flow divider
   ▪ for a storage unit.
2. Move the mouse to the desired location on the map and click.

To add a Node using the Data Browser:
1. Select the type of node (Junction, Outfall, Flow Divider, or Storage Unit from the
   categories list of the Data Browser.
2. Click the button.
3. Enter the node's X and Y coordinates in the Property Editor if you want it to appear
   on the study area map.
Adding a Link

To add a Link using the Object Toolbar:
1. Click the button corresponding to the type of link to add (if its not already depressed):
   - for a Conduit
   - for a Pump
   - for an Orifice
   - for a Weir
   - for an Outlet.
2. On the study area map, click the mouse on the link's inlet (upstream) node.
3. Move the mouse in the direction of the link's outlet (downstream) node, clicking at all intermediate points needed to define the link alignment.
4. Click the mouse a final time over the link's outlet (downstream) node.

Pressing the right mouse button or the `<Esc>` key while drawing a link will cancel the operation.

To add a Link using the Data Browser:
1. Select the type of link to add from the categories listed in the Data Browser.
2. Click the button.
3. Enter the names of the inlet and outlet nodes of the link in the Property Editor.

Adding a Map Label

To add a text label to the Study Area Map:
1. Click the Text button on the Object Toolbar.
2. Click the mouse on the map where the top left corner of the label should appear.
3. Enter the text for the label.
4. Press the Enter key to accept the label or <Esc> to cancel.

Adding a Non-visual Object

To add an object belonging to a class that is not displayable on the Study Area Map (which includes Climatology, Aquifers, Snow Packs, Unit Hydrographs, Transects, Control Rules, Pollutants, Land Uses, Curves, Time Series, and Time Patterns):
1. Select the object's category from the list in the Data Browser
2. Click the button.
3. Edit the object's properties in the special editor dialog form that appears (see Appendix C for descriptions of these editors).

6.3 Selecting and Moving Objects

To select an object on the map:
1. Make sure that the map is in Selection mode (the mouse cursor has the shape of an arrow pointing up to the left). To switch to this mode, either click the Select Object button on the Map Toolbar or choose Edit >> Select Object from the Main Menu.
2. Click the mouse over the desired object on the map.

To select an object using the Data Browser:
1. Select the object’s category from the upper list in the Browser.
2. Select the object from the lower list in the Browser.

Rain gages, subcatchments, and nodes can be moved to another location on the Study Area Map. To move an object to another location:
1. Select the object on the map.
2. With the left mouse button held down over the object, drag it to its new location.
3. Release the mouse button.

The following alternative method can also be used:
1. Select the object to be moved from the Data Browser (it must be either a rain gage, subcatchment, node, or map label).
2. With the left mouse button held down, drag the item from the Items list box of the Data Browser to its new location on the map.
3. Release the mouse button.

Note that the second method can be used to place objects on the map that were imported from a project file that had no coordinate information included in it.

6.4 Editing Objects

To edit an object appearing on the Study Area Map:
1. Select the object on the map.
2. If the Property Editor is not visible either:
   - double click on the object
   - or right-click on the object and select Properties from the pop-up menu that appears
   - or click on in the Data Browser.
3. Edit the object’s properties in the Property Editor.

Appendix B lists the properties associated with each of SWMM’s visual objects.

To edit an object listed in the Data Browser:
1. Select the object in the Data Browser.
2. Either:
   - click on in the Data Browser,
   - or double-click the item in the Objects list,
   - or press the Enter key.

Depending on the class of object selected, a special property editor will appear in which the object’s properties can be modified. Appendix C describes all of the special property editors used with SWMM’s non-visual objects.

! The unit system in which object properties are expressed depends on the choice of units for flow rate. Using a flow rate expressed in cubic feet, gallons or acre-feet implies that US units will be used for all quantities. Using a flow rate expressed in liters or cubic meters means that SI metric units will be used. Flow units are selected either from the project’s default Node/Link properties (see Section 5.4) or from its General Simulation Options (see Section 8.1). The units used for all properties are listed in Appendix A.1.

6.5 Converting an Object

It is possible to convert a node or link from one type to another without having to first delete the object and add a new one in its place. An example would be converting a Junction node into an Outfall node, or converting an Orifice link into a Weir link. To convert a node or link to another type:
1. Right-click the object on the map.
2. Select Convert To from the popup menu that appears.
3. Select the new type of node or link to convert to from the sub-menu that appears.
4. Edit the object to provide any data that was not included with the previous type of object.

Only data that is common to both types of objects will be preserved after an object is converted to a different type. For nodes this includes its name, position, description, tag, external inflows, treatment functions, and invert elevation. For links it includes just its name, end nodes, description, and tag.
6.6 Copying and Pasting Objects

The properties of an object displayed on the Study Area Map can be copied and pasted into another object from the same category.

To copy the properties of an object to SWMM's internal clipboard:
1. Right-click the object on the map.
2. Select Copy from the pop-up menu that appears.

To paste copied properties into an object:
1. Right-click the object on the map.
2. Select Paste from the pop-up menu that appears.

Only data that can be shared between objects of the same type can be copied and pasted. Properties not copied include the object's name, coordinates, end nodes (for links), Tag property and any descriptive comment associated with the object. For Map Labels, only font properties are copied and pasted.

6.7 Shaping and Reversing Links

Links can be drawn as polylines containing any number of straight-line segments that define the alignment or curvature of the link. Once a link has been drawn on the map, interior points that define these line segments can be added, deleted, and moved. To edit the interior points of a link:

1. Select the link to edit on the map and put the map in Vertex Selection mode by either clicking on the Map Toolbar, selecting Edit >> Select Vertex from the Main Menu, or right clicking on the link and selecting Vertices from the popup menu.
2. The mouse pointer will change shape to an arrow tip, and any existing vertex points on the link will be displayed as small open squares. The currently selected vertex will be displayed as a filled square. To select a particular vertex, click the mouse over it.
3. To add a new vertex to the link, right-click the mouse and select Add Vertex from the popup menu (or simply press the Insert key on the keyboard).
4. To delete the currently selected vertex, right-click the mouse and select Delete Vertex from the popup menu (or simply press the Delete key on the keyboard).
5. To move a vertex to another location, drag it to its new position with the left mouse button held down.
6. While in Vertex Selection mode you can begin editing the vertices for another link by simply clicking on the link. To leave Vertex Selection mode, right-click on the map and select Quit Editing from the popup menu, or simply select one of the other buttons on the Map Toolbar.

A link can also have its direction reversed (i.e., its end nodes switched) by right clicking on it and selecting Reverse from the pop-up menu that appears. Normally, links should be oriented so that the upstream end is at a higher elevation than the downstream end.
6.8 Shaping a Subcatchment

Subcatchments are drawn on the Study Area Map as closed polygons. To edit or add vertices to the polygon, follow the same procedures used for links. To completely redraw the outline of the subcatchment, right-click on the subcatchment’s centroid symbol and select Redraw from the popup menu that appears. Then use the same procedure as for drawing a new subcatchment (see Section 6.2). If the subcatchment is originally drawn or is edited to have two or less vertices, then only its centroid symbol will be displayed on the Study Area Map.

6.9 Deleting an Object

To delete an object:
1. Select the object on the map or from the Data Browser.
2. Either click the button on the Data Browser or press the <Delete> key on the keyboard.

⚠️ You can require that all deletions be confirmed before they take effect. See the General Preferences page of the Program Preferences dialog box described in Section 4.9.

6.10 Editing or Deleting a Group of Objects

A group of objects located within an irregular region of the Study Area Map can have a common property edited or be deleted all together. To select such a group of objects:

1. Choose Edit >> Select Region from the Main Menu or click on the Map Toolbar.
2. Draw a polygon around the region of interest on the map by clicking the left mouse button at each successive vertex of the polygon.
3. Close the polygon by clicking the right button or by pressing the <Enter> key; cancel the selection by pressing the <Esc> key.

To select all objects in the project, whether in view or not, select Edit >> Select All from the Main Menu.

Once a group of objects has been selected, you can edit a common property shared among them:

1. Select Edit >> Group Edit from the Main Menu.
2. Use the Group Editor dialog that appears to select a property and specify its new value.

The Group Editor dialog, shown below, is used to modify a property for a selected group of objects. To use the dialog:
1. Select a class of object (Subcatchments, Junctions, or Conduits) to edit.

2. Check the "with Tag equal to" box if you want to add a filter that will limit the objects selected for editing to those with a specific Tag value.

3. Enter a Tag value to filter on if you have selected that option.

4. Select the property to edit.

5. In the New Value field, enter the value that should replace the existing value for all selected objects. If an ellipsis button appears next to the New Value field, then click the button to bring up a specialized editor for that property.

6. Click OK to execute the group edit.

To delete the objects located within a selected area of the map, select Edit >> Group Delete from the Main Menu. Then select the categories of objects you wish to delete from the dialog box that appears. Recall that deleting a node will also delete any links connected to the node.
EPA SWMM can display a map of the study area being modeled. This section describes how you can manipulate this map to enhance your visualization of the system.

7.1 Selecting a Map Theme

A map theme displays object properties in color-coded fashion on the Study Area Map. The dropdown list boxes on the Map Browser are used for selecting a theme to display for Subcatchments, Nodes and Links.

Methods for changing the color-coding associated with a theme are discussed in Section 7.9 below.

7.2 Setting the Map’s Dimensions

The physical dimensions of the map can be defined so that map coordinates can be properly scaled to the computer’s video display. To set the map's dimensions:

1. Select View >> Dimensions from the Main Menu.
2. Enter coordinates for the lower-left and upper-right corners of the map into the Map Dimensions dialog (see below) that appears or click the Auto-Size button to automatically set the dimensions based on the coordinates of the objects currently included in the map.
3. Select the distance units to use for these coordinates.
4. Click the OK button to resize the map.
If you are going to use a backdrop image with the automatic distance and area calculation feature, then it is recommended that you set the map dimensions immediately after creating a new project. Map distance units can be different from conduit length units. The latter (feet or meters) depend on whether flow rates are expressed in US or metric units. SWMM will automatically convert units if necessary.

### 7.3 Utilizing a Backdrop Image

SWMM can display a backdrop image behind the Study Area Map. The backdrop image might be a street map, utility map, topographic map, site development plan, or any other relevant picture or drawing. For example, using a street map would simplify the process of adding sewer lines to the project since one could essentially digitize the drainage system's nodes and links directly on top of it.

The backdrop image must be a Windows metafile, bitmap, or JPEG image created outside of SWMM. Once imported, its features cannot be edited, although its scale and viewing area will change as the map window is zoomed and panned. For this reason metafiles work better than bitmaps or JPEGs since they will not loose resolution when re-scaled. Most CAD and GIS programs have the ability to save their drawings and maps as metafiles.
Selecting **View >> Backdrop** from the Main Menu will display a sub-menu with the following commands:

- **Load** (loads a backdrop image file into the project)
- **Unload** (unloads the backdrop image from the project)
- **Align** (aligns the drainage system schematic with the backdrop)
- **Resize** (resizes the map dimensions of the backdrop)
- **Watermark** (toggles the backdrop image appearance between normal and lightened)

To load a backdrop image, select **View >> Backdrop >> Load** from the Main Menu. A Backdrop Image Selector dialog form will be displayed. The entries on this form are as follows:

**Backdrop Image File**

Enter the name of the file that contains the image. You can click the button to bring up a standard Windows file selection dialog from which you can search for the image file.

**World Coordinates File (optional)**

If a “world” file exists for the image, enter its name here, or click the button to search for it. A world file contains geo-referencing information for the image and can be created from the software that produced the image file or by using a text editor. It contains six lines with the following information:

- Line 1: real world width of a pixel in the horizontal direction.
- Line 2: X rotation parameter (not used).
- Line 3: Y rotation parameter (not used).
- Line 4: negative of the real world height of a pixel in the vertical direction.
- Line 5: real world X coordinate of the upper left corner of the image.
- Line 6: real world Y coordinate of the upper left corner of the image.

If no world file is specified, then the backdrop will be scaled to fit into the center of the map display window.
Scale Map to Backdrop Image
This option is only available when a world file has been specified. Selecting it forces the dimensions of the Study Area Map to coincide with those of the backdrop image. In addition, all existing objects on the map will have their coordinates adjusted so that they appear within the new map dimensions yet maintain their relative positions to one another. Selecting this option may then require that the backdrop be re-aligned so that its position relative to the drainage area objects is correct. How to do this is described below.

The backdrop image can be re-positioned relative to the drainage system by selecting View >> Backdrop >> Align. This allows the backdrop image to be moved across the drainage system (by moving the mouse with the left button held down) until one decides that it lines up properly.

The backdrop image can also be resized by selecting View >> Backdrop >> Resize. In this case the following Backdrop Dimensions dialog will appear.

![Backdrop Dimensions Dialog](image)

The dialog lets you manually enter the X,Y coordinates of the backdrop’s lower left and upper right corners. The Study Area Map’s dimensions are also displayed for reference. While the dialog is visible you can view map coordinates by moving the mouse over the map window and noting the X,Y values displayed in SWMM’s Status Panel (at the bottom of the main window).

Selecting the Resize Backdrop Image Only button will resize only the backdrop, and not the Study Area Map, according to the coordinates specified. Selecting the Scale Backdrop Image to Map button will position the backdrop image in the center of the Study Area Map.
Map and have it resized to fill the display window without changing its aspect ratio. The
map's lower left and upper right coordinates will be placed in the data entry fields for the
backdrop coordinates, and these fields will become disabled. Selecting Scale Map to
Backdrop Image makes the dimensions of the map coincide with the dimensions being
set for the backdrop image. Note that this option will change the coordinates of all objects
currently on the map so that their positions relative to one another remain unchanged.
Selecting this option may then require that the backdrop be re-aligned so that its position
relative to the drainage area objects is correct.

⚠️ Exercise caution when selecting the Scale Map to Backdrop Image option in
either the Backdrop Image Selector dialog or the Backdrop Dimensions dialog as it will modify the coordinates of all existing objects currently on the Study Area
Map. You might want to save your project before carrying out this step in case the
results are not what you expected.

The name of the backdrop image file and its map dimensions are saved along with the
rest of a project’s data whenever the project is saved to file.

For best results in using a backdrop image:

- Use a metafile, not a bitmap.
- If the image is loaded before any objects are added to the project then scale the map
to it.

### 7.4 Zooming the Map

To Zoom In on the Study Area Map:

1. Select View >> Zoom In from the Main Menu or click on the Map Toolbar.
2. To zoom in 100% (i.e., 2X), move the mouse to the center of the zoom area and click the left button.
3. To perform a custom zoom, move the mouse to the upper left corner of the zoom area
   and with the left button pressed down, draw a rectangular outline around the zoom
   area. Then release the left button.

To Zoom Out on the Study Area Map:

1. Select View >> Zoom Out from the Main Menu or click on the Map Toolbar.
2. Move the mouse to the center of the new zoom area and click the left button.
3. The map will be returned to its previous zoom level.

### 7.5 Panning the Map

To pan across the Study Area Map window:

1. Select View >> Pan from the Main Menu or click on the Map Toolbar.
2. With the left button held down over any point on the map, drag the mouse in the
direction you wish to pan in.
3. Release the mouse button to complete the pan.

To pan using the Overview Map (which is described in Section 7.10 below):
1. If not already visible, bring up the Overview Map by selecting View >> Overview
Map from the Main Menu.
2. If the Study Area Map has been zoomed in, an outline of the current viewing area
will appear on the Overview Map. Position the mouse within this outline on the
Overview Map.
3. With the left button held down, drag the outline to a new position.
4. Release the mouse button and the Study Area Map will be panned to an area
   corresponding to the outline on the Overview Map.

**7.6 Viewing at Full Extent**

To view the Study Area Map at full extent, either:
a) select View >> Full Extent from the Main Menu, or
b) press on the Map Toolbar.

**7.7 Finding an Object**

To find an object on the Study Area Map
whose name is known:
1. Select View >> Find from the Main Menu or click on the Standard Toolbar.
2. In the Map Finder dialog that appears, select the type of object to find and enter
   its name.
3. Click the Go button.

If the object exists, it will be highlighted on the map and in the Data Browser. If the map
is currently zoomed in and the object falls outside the current map boundaries, the map
will be panned so that the object comes into view.

⚠️ User-assigned object names in SWMM are not case sensitive. E.g., NODE123 is
equivalent to Node123.
After an object is found, the Map Finder dialog will also list:
- the outlet connections for a subcatchment
- the connecting links for a node
- the connecting nodes for a link.

7.8 Submitting a Map Query

A Map Query identifies objects on the study area map that meet a specific criterion (e.g., nodes which flood, links with velocity below 2 ft/sec, etc.). To submit a map query:

1. Select a time period in which to query the map from the Map Browser.
2. Select **View >> Query** or click on the Map Toolbar.
3. Fill in the following information in the Query dialog that appears:
   - Select whether to search for Subcatchments, Nodes or Links.
   - Select a parameter to query.
   - Select the appropriate operator: Above, Below, or Equals.
   - Enter a value to compare against.

4. Click the **Go** button. The number of objects that meet the criterion will be displayed in the Query dialog and each such object will be highlighted on the Study Area Map.
5. As a new time period is selected in the Browser, the query results are automatically updated.
6. You can submit another query using the dialog box or close it by clicking the button in the upper right corner.
After the Query box is closed the map will revert back to its original display.

7.9 Using the Map Legends

Map Legends associate a color with a range of values for the current theme being viewed. Separate legends exist for Subcatchments, Nodes, and Links. A Date/Time Legend is also available for displaying the date and clock time of the simulation period being viewed on the map.

To display or hide a map legend:
1. Select View >> Legends from the Main Menu or right-click on the map and select Legends from the pop-up menu that appears
2. Click on the type of legend whose display should be toggled on or off.
   A visible legend can also be hidden by double clicking on it.

To move a legend to another location press the left mouse button over the legend, drag the legend to its new location with the button held down, and then release the button.

To edit a legend, either select View >> Legends >> Modify from the Main Menu or right-click on the legend if it is visible. Then use the Legend Editor dialog that appears to modify the legend's colors and intervals.

The Legend Editor is used to set numerical ranges to which different colors are assigned for viewing a particular parameter on the network map. It works as follows:

- Numerical values, in increasing order, are entered in the edit boxes to define the ranges. Not all four boxes need to have values.
- To change a color, click on its color band in the Editor and then select a new color from the Color Dialog that will appear.
- Click the **Auto-Scale** button to automatically assign ranges based on the minimum and maximum values attained by the parameter in question at the current time period.
- The **Color Ramp** button is used to select from a list of built-in color schemes.
- The **Reverse Colors** button reverses the ordering of the current set of colors (the color in the lowest range becomes that of the highest range and so on).
- Check **Framed** if you want a frame drawn around the legend.

### 7.10 Using the Overview Map

The Overview Map, as pictured below, allows one to see where in terms of the overall system the main Study Area Map is currently focused. This zoom area is depicted by the rectangular outline displayed on the Overview Map. As you drag this rectangle to another position the view within the main map will be redrawn accordingly. The Overview Map can be toggled on and off by selecting **View >> Overview Map** from the Main Menu. The Overview Map window can also be dragged to any position as well as be re-sized.

![Overview Map Diagram]
7.11 Setting Map Display Options

The Map Options dialog (shown below) is used to change the appearance of the Study Area Map. There are several ways to invoke it:

- select View >> Options from the Main Menu or,
- click the Options button on the Standard Toolbar when the Study Area Map window has the focus or,
- right-click on any empty portion of the map and select Options from the popup menu that appears.

The dialog contains a separate page, selected from the panel on the left side of the form, for each of the following display option categories:

- **Subcatchments** (controls fill style, symbol size, and outline thickness of subcatchment areas)
- **Nodes** (controls size of nodes and making size be proportional to value)
- **Links** (controls thickness of links and making thickness be proportional to value)
- **Labels** (turns display of map labels on/off)
- **Annotation** (displays or hides node/link ID labels and parameter values)
- **Symbols** (turns display of storage unit, pump, and regulator symbols on/off)
- **Flow Arrows** (selects visibility and style of flow direction arrows)
- **Background** (changes color of map's background)
Subcatchment Options

The Subcatchments page of the Map Options dialog controls how subcatchment areas are displayed on the study area map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fill Style</td>
<td>Selects style used to fill interior of subcatchment area</td>
</tr>
<tr>
<td>Symbol Size</td>
<td>Sets the size of the symbol (in pixels) placed at the centroid of a subcatchment area</td>
</tr>
<tr>
<td>Outline Thickness</td>
<td>Sets the thickness of the line used to draw a subcatchment's boundary; set to zero if no boundary should be displayed</td>
</tr>
<tr>
<td>Display Link to Outlet</td>
<td>If checked then a dashed line is drawn between the subcatchment centroid and the subcatchment's outlet node (or outlet subcatchment)</td>
</tr>
</tbody>
</table>

Node Options

The Nodes page of the Map Options dialog controls how nodes are displayed on the study area map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Size</td>
<td>Selects node diameter in pixels</td>
</tr>
<tr>
<td>Proportional to Value</td>
<td>Select if node size should increase as the viewed parameter increases in value</td>
</tr>
<tr>
<td>Display Border</td>
<td>Select if a border should be drawn around each node (recommended for light-colored backgrounds)</td>
</tr>
</tbody>
</table>

Link Options

The Links page of the Map Options dialog controls how links are displayed on the map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Link Size</td>
<td>Sets thickness of links displayed on map (in pixels)</td>
</tr>
<tr>
<td>Proportional to Value</td>
<td>Select if link thickness should increase as the viewed parameter increases in value</td>
</tr>
<tr>
<td>Display Border</td>
<td>Check if a black border should be drawn around each link</td>
</tr>
</tbody>
</table>
Label Options

The Labels page of the Map Options dialog controls how user-created map labels are displayed on the study area map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use Transparent Text</td>
<td>Check to display label with a transparent background (otherwise an opaque background is used)</td>
</tr>
<tr>
<td>At Zoom Of</td>
<td>Selects minimum zoom at which labels should be displayed; labels will be hidden at zooms smaller than this</td>
</tr>
</tbody>
</table>

Annotation Options

The Annotation page of the Map Options dialog form determines what kind of annotation is provided alongside of the objects on the study area map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rain Gage IDs</td>
<td>Check to display rain gage ID names</td>
</tr>
<tr>
<td>Subcatch IDs</td>
<td>Check to display subcatchment ID names</td>
</tr>
<tr>
<td>Node IDs</td>
<td>Check to display node ID names</td>
</tr>
<tr>
<td>Link IDs</td>
<td>Check to display link ID names</td>
</tr>
<tr>
<td>Subcatch Values</td>
<td>Check to display value of current subcatchment variable</td>
</tr>
<tr>
<td>Node Values</td>
<td>Check to display value of current node variable</td>
</tr>
<tr>
<td>Link Values</td>
<td>Check to display value of current link variable</td>
</tr>
<tr>
<td>Use Transparent Text</td>
<td>Check to display text with a transparent background (otherwise an opaque background is used)</td>
</tr>
<tr>
<td>Font Size</td>
<td>Adjusts the size of the font used to display annotation</td>
</tr>
<tr>
<td>At Zoom Of</td>
<td>Selects minimum zoom at which annotation should be displayed; all annotation will be hidden at zooms smaller than this</td>
</tr>
</tbody>
</table>

Symbol Options

The Symbols page of the Map Options dialog determines which types of objects are represented with special symbols on the map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display Node Symbols</td>
<td>If checked then special node symbols will be used</td>
</tr>
<tr>
<td>Display Link Symbols</td>
<td>If checked then special link symbols will be used</td>
</tr>
<tr>
<td>At Zoom Of</td>
<td>Selects minimum zoom at which symbols should be displayed; symbols will be hidden at zooms smaller than this</td>
</tr>
</tbody>
</table>
Flow Arrow Options

The Flow Arrows page of the Map Options dialog controls how flow-direction arrows are displayed on the map.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arrow Style</td>
<td>Selects style (shape) of arrow to display (select None to hide arrows)</td>
</tr>
<tr>
<td>Arrow Size</td>
<td>Sets arrow size</td>
</tr>
<tr>
<td>At Zoom Of</td>
<td>Selects minimum zoom at which arrows should be displayed; arrows will be hidden at zooms smaller than this</td>
</tr>
</tbody>
</table>

⚠️ Flow direction arrows will only be displayed after a successful simulation has been made and a computed parameter has been selected for viewing. Otherwise the direction arrow will point from the user-designated start node to end node.

Background Options

The Background page of the Map Options dialog offers a selection of colors used to paint the map’s background with.

7.12 Exporting the Map

The full extent view of the study area map can be saved to file using either:

- Autodesk's DXF (Drawing Exchange Format) format,
- the Windows enhanced metafile (EMF) format,
- EPA SWMM's own ASCII text (.map) format.

The DXF format is readable by many Computer Aided Design (CAD) programs. Metafiles can be inserted into word processing documents and loaded into drawing programs for re-scaling and editing. Both formats are vector-based and will not lose resolution when they are displayed at different scales.

To export the map to a DXF, metafile, or text file:

1. Select **File >> Export >> Map.**
2. In the Map Export dialog that appears select the format that you want the map saved in.
3. If you select DXF format, you have a choice of how nodes will be represented in the DXF file. They can be drawn as filled circles, as open circles, or as filled squares. Not all DXF readers can recognize the format used in the DXF file to draw a filled circle. Also note that map annotation, such as node and link ID labels, will not be exported, but map label objects will be.
4. After choosing a format, click **OK** and enter a name for the file in the Save As dialog that appears.
CHAPTER 8 - RUNNING A SIMULATION

After a study area has been suitably described, its runoff response, flow routing and water quality behavior can be simulated. This section describes how to specify options to be used in the analysis, how to run the simulation and how to troubleshoot common problems that might occur.

8.1 Setting Simulation Options

SWMM has a number of options that control how the simulation of a stormwater drainage system is carried out. To set these options:

1. Select the Options category from the Data Browser and then click the button.
2. A Simulation Options dialog will appear where you can make selections for the following categories of options:
3. When finished with the dialog, click the **OK** button to accept your choices or the **Cancel** button to cancel them.

The following sections discuss each category of options.

**General Options**

The General page of the Simulation Options dialog sets values for the following options:

**Flow Units**

Six choices of flow units are available. Selecting a US flow unit means that all other quantities will be expressed in US units, while choosing a metric flow unit will force all quantities to be expressed in metric units (see Appendix A.1). Units on previously entered data are not automatically adjusted if the unit system is changed.

**Infiltration Model**

This option controls how infiltration of rainfall into the upper soil zone of subcatchments is modeled. The choices are:

- Horton
- Green-Ampt
- Curve Number

Changing this option will require re-entering values for the infiltration parameters in each subcatchment.

**Routing Method**

This option determines which method is used to route flows through the conveyance system. The choices are:

- Steady Flow
- Kinematic Wave
- Dynamic Wave

**Allow Ponding**

Checking this option will allow excess water to collect atop nodes and be re-introduced into the system as conditions permit. In order for ponding to actually occur at a particular node, a non-zero value for its Ponded Area attribute must be used.

**Report Control Actions**

Check this option if you want the simulation’s Status Report to list all control actions taken by the Control Rules associated with a project. This option should only be used for short-term simulation.
Date Options

The Dates page of the Simulation Options dialog determines the starting and ending
dates/times of a simulation.

Start Analysis On
Enter the date (month/day/year) and time of day when the simulation begins.

Start Reporting On
Enter the date and time of day when reporting of simulation results is to begin. This must
be on or after the simulation starting date and time.

End Analysis On
Enter the date and time when the simulation is to end.

Start Sweeping On
Enter the day of the year (month/day) when street sweeping operations begin. The default
is January 1.

End Sweeping On
Enter the day of the year (month/day) when street sweeping operations end. The default
is December 31.

Antecedent Dry Days
Enter the number of days with no rainfall prior to the start of the simulation. This value is
used to compute an initial buildup of pollutant load on the surface of subcatchments.

⚠️ If rainfall or climate data are read from external files, then the simulation dates
should be set to coincide with the dates recorded in these files.

Time Step Options

The Time Steps page of the Simulation Options dialog establishes the length of the time
steps used for runoff computation, routing computation and results reporting. Time steps
are specified in days and hours:minutes:seconds.

Runoff - Wet Weather Time Step
Enter the time step length used to compute runoff from subcatchments during periods of
rainfall or when ponded water still remains on the surface.

Runoff - Dry Weather Time Step
Enter the time step length used for runoff computations (consisting essentially of
pollutant buildup) during periods when there is no rainfall and no ponded water. This
must be greater or equal to the Wet Weather time step.

Routing Time Step
Enter the time step length used for routing flows and water quality constituents through
the conveyance system. Note that Dynamic Wave routing requires a much smaller time
step than the other methods of flow routing.
Reporting Time Step
Enter the time interval for reporting of computed results.

Dynamic Wave Options

The Dynamic Wave page of the Simulation Options dialog sets several parameters that control how the dynamic wave flow routing computations are made. These parameters have no effect for the other flow routing methods.

Compatibility Mode
Indicates which version of SWMM's dynamic wave solution method should be used.
- **SWMM 5** - uses Picard iterations (a method of successive approximations) to integrate the nodal continuity equation at each time step. The Preissmann slot method is used to handle surcharging.
- **SWMM 4** - uses modified Euler integration and a special iterative procedure to handle surcharging.
- **SWMM 3** - same as SWMM 4 except that different weights are used to compute an average conduit flow area and hydraulic radius based on values computed at either end.

Inertial Terms
Indicates how the inertial terms in the St. Venant momentum equation will be handled.
- **KEEP** maintains these terms at their full value under all conditions.
- **DAMPEN** reduces the terms as flow comes closer to being critical and ignores them when flow is supercritical.
- **IGNORE** drops the terms altogether from the momentum equation, producing what is essentially a Diffusion Wave solution.

Variable Time Step
Indicates whether or not a variable time step should be used. The variable step is computed for each time period so as to satisfy the Courant stability criterion for each conduit and to prevent an excessive change in water depth at each node.

Safety Factor
This is a safety factor, between 10 and 200%, which is applied to the variable time step computed from the Courant criterion. It only applies when the Variable Time Step option is checked.

Time Step for Conduit Lengthening
This is a time step, in seconds, used to artificially lengthen conduits so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer conduits will require lengthening. A value of 0 means that no conduits will be lengthened.

Minimum Surface Area
This is a minimum surface area used at nodes when computing changes in water depth. If 0 is entered, then the default value of 12.566 ft² (i.e., the area of a 4-ft diameter manhole) is used.
File Options

The Interface Files page of the Simulation Options dialog is used to specify which interface files will be used or saved during the simulation. (Interface files are described in Chapter 11.) The page contains a list box with three buttons underneath it. The list box lists the currently selected files, while the buttons are used as follows:

**Add** adds a new interface file specification to the list.

**Edit** edits the properties of the currently selected interface file.

**Delete** deletes the currently selected interface from the project (but not from your hard drive).

When the **Add** or **Edit** buttons are clicked, an Interface File Selection dialog appears where you can specify the type of interface file, whether it should be used or saved, and its name. The entries on this dialog are as follows:
File Type
Select the type of interface file to be specified.

Use / Save Buttons
Select whether the named interface file will be used to supply input to a simulation run or whether simulation results will be saved to it.

File Name
Enter the name of the interface file.

Browse Button
Click this button to launch a standard file selection dialog from which the path and name of the interface file can be selected.

8.2 Starting a Simulation

To start a simulation run either select Project >> Run Simulation from the Main Menu or click on the Standard Toolbar. A Run Status window will appear which displays the progress of the simulation.
To stop a run before its normal termination, click the **Stop** button on the Run Status window or press the `<Esc>` key. Simulation results up until the time when the run was stopped will be available for viewing. To minimize the SWMM program while a simulation is running, click the **Minimize** button on the Run Status window.

If the analysis runs successfully the **icon** will appear in the Run Status section of the Status Bar at the bottom of SWMM’s main window. Any error or warning messages will appear in a Status Report window. If you modify the project after a successful run has been made, the faucet icon changes to a broken faucet indicating that the current computed results no longer apply to the modified project.

### 8.3 Troubleshooting Results

When a run ends prematurely, the Run Status dialog will indicate the run was unsuccessful and direct the user to the Status Report for details. The Status Report will include an error statement, code, and description of the problem (e.g., ERROR 138: Node TG040 has initial depth greater than maximum depth). Consult Appendix E for a description of SWMM’s error messages. The following are the most common reasons for a run to end prematurely or to contain questionable results.

**Unknown ID Error Message**

This message typically appears when an object references another object that was never defined. An example would be a subcatchment whose outlet was designated as N29, but no such subcatchment or node with that name exists. Similar situations can exist for incorrect references made to Curves, Time Series, Time Patterns, Aquifers, Snow Packs, Transects, Pollutants, and Land Uses.

**File Errors**

File errors can occur when:
- a file cannot be located on the user's computer
- a file being used has the wrong format
- a file being written cannot be opened because the user does not have write privileges for the directory (folder) where the file is to be stored.

SWMM needs to have write privileges for a directory (folder) where temporary files are stored during a run. The original default is the directory where Windows writes its temporary files. If this directory does not exist or the user does not have write privileges to it, then a new directory must be assigned by using the Program Preferences dialog, which is discussed in Section 4.9.

**Drainage System Layout Errors**

A valid drainage system layout must obey the following conditions:
- The links must be oriented so that no closed loop exists in the network.
- An outfall node can have only one conduit link connected to it.
- A flow divider node must have exactly two outflow links.
- Under Kinematic Wave routing, a junction node can only have one outflow link and a regulator link cannot be the outflow link of a non-storage node.
- Under Dynamic Wave routing there must be at least one outfall node in the network.

An error message will be generated if any of these conditions are violated.

Excessive Continuity Errors

When a run completes successfully, the mass continuity errors for runoff, flow routing, and pollutant routing will be displayed in the Run Status window. These errors represent the percent difference between initial storage plus total inflow and final storage plus total outflow for the entire drainage system. If they exceed some reasonable level, such as 10 percent, then the validity of the analysis results must be questioned. The most common reason for an excessive continuity error is too large a computational time step.

Unstable Flow Routing Results

Due to the explicit nature of the numerical methods used for Dynamic Wave routing (and to a lesser extent, Kinematic Wave routing), the flows in some links or water depths at some nodes may fluctuate or oscillate significantly at certain periods of time as a result of numerical instabilities in the solution method. SWMM does not automatically identify when such conditions exist, so it is up to the user to verify the numerical stability of the model and to determine if the simulation results are valid for the modeling objectives. Time series plots at key locations in the network can help identify such situations as can a scatter plot between a link’s flow and the corresponding water depth at its upstream node (see Section 9.4, Viewing Results with a Graph).

Numerical instabilities can occur over short durations and may not be apparent when time series are plotted with a long time interval. When detecting such instabilities, it is recommended that a reporting time step of 1 minute or less be used, at least for an initial screening of results.

Numerical instabilities under Dynamic Wave flow routing can be reduced by:
- reducing the routing time step
- utilizing the variable time step option with a smaller time step factor
- selecting to ignore the inertial terms of the momentum equation
- selecting the option to lengthen short conduits.
CHAPTER 9 - VIEWING RESULTS

This chapter describes the different ways in which the results of a simulation can be viewed. These include a status report, various map views, graphs, tables, and a statistical frequency report.

9.1 Viewing a Status Report

A Status Report is available for viewing after each simulation. It contains:

- a list of any error conditions encountered during the run
- the mass continuity errors for runoff quantity and quality as well as for flow and water quality routing
- summary results tables for all drainage system nodes and links
- frequency distributions of time step size and iterations required when Dynamic Wave routing analyses are performed.

To view the Status Report, select Report >> Status from the Main Menu.

9.2 Variables That Can be Viewed

Computed results for the following variables are available for viewing on the map and can be plotted, tabulated, and statistically analyzed:

<table>
<thead>
<tr>
<th>Subcatchment Variables</th>
<th>Link Variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>rainfall rate (in/hr or mm/hr)</td>
<td>flow rate (flow units)</td>
</tr>
<tr>
<td>snow depth (inches or millimeters)</td>
<td>average water depth (ft or m)</td>
</tr>
<tr>
<td>losses (infiltration + evaporation in in/hr or mm/hr)</td>
<td>flow velocity (ft/sec or m/sec)</td>
</tr>
<tr>
<td>runoff flow (flow units)</td>
<td>Froude number (dimensionless)</td>
</tr>
<tr>
<td>groundwater flow into the drainage network (flow units)</td>
<td>capacity (ratio of depth to full depth)</td>
</tr>
<tr>
<td>groundwater elevation (ft or m)</td>
<td>concentration of each pollutant (mass/liter)</td>
</tr>
<tr>
<td>washoff concentration of each pollutant (mass/liter)</td>
<td></td>
</tr>
</tbody>
</table>
### Node Variables
- water depth (ft or m above the node invert elevation)
- hydraulic head (ft or m, absolute elevation per vertical datum)
- water volume held in storage (including ponded water, ft³ or m³)
- lateral inflow (runoff + all other external inflows, in flow units)
- total inflow (lateral inflow + upstream inflows, in flow units)
- internal flooding (inflows lost from the system when the water depth exceeds the defined maximum node depth, flow units)
- concentration of each pollutant after any treatment (mass/liter)

### System-Wide Variables
- air temperature (degrees F or C)
- total rainfall (in/hr or mm/hr)
- total snow depth (inches or millimeters)
- average losses (in/hr or mm/hr)
- total runoff flow (flow units)
- total dry weather inflow (flow units)
- total groundwater inflow (flow units)
- total I&I inflow (flow units)
- total direct inflow (flow units)
- total external inflow (flow units)
- total internal flooding (flow units)
- total outflow from outfalls (flow units)
- total nodal storage volume (ft³ or m³)

### Viewing Results on the Map

There are several ways to view the values of certain input parameters and simulation results directly on the Study Area Map:

- For the current settings on the Map Browser, the subcatchments, nodes and links of the map will be colored according to their respective Map Legends. The map's color coding will be updated as a new time period is selected in the Map Browser.
- When the Flyover Map Labeling program preference is selected (see Section 4.9), moving the mouse over any map object will display its ID name and the value of its current theme parameter in a hint-style box.
- ID names and parameter values can be displayed next to all subcatchments, nodes and/or links by selecting the appropriate options on the Annotation page of the Map Options dialog (see Section 7.11).
- Subcatchments, nodes or links meeting a specific criterion can be identified by submitting a Map Query (see Section 7.8).
- You can animate the display of results on the network map either forward or backward in time by using the controls on the Animator Toolbar (see Section 4.3).
- The map can be printed, copied to the Windows clipboard, or saved as a DXF file or Windows metafile (see Section 7.12).

### Viewing Results with a Graph

Analysis results can be viewed using several different types of graphs. Graphs can be printed, copied to the Windows clipboard, or saved to a text file or to a Windows metafile. The following types of graphs can be created from available simulation results:
- Time Series Plot:

- Profile Plot:

- Scatter Plot:

The user can zoom in or out of any graph by holding down the <Shift> key while drawing a zoom rectangle with the mouse's left button held down. Drawing the rectangle from left to right zooms in, drawing from right to left zooms out. The plot can also be panned in any direction by holding down the <Ctrl> key and moving the mouse across the plot with the left button held down.

An opened graph will normally be redrawn when a new simulation is run. To prevent the automatic updating of a graph once a new set of results is computed the user can lock the...
current graph by clicking the icon in the upper left corner of the graph. To unlock the graph, click the icon again.

**Time Series Plots**

A Time Series Plot graphs the value of a particular variable at up to six locations against time. When only a single location is plotted, and that location has calibration data registered for the plotted variable, then the calibration data will be plotted along with the simulated results (see Section 5.5 for instructions on how to register calibration data with a project).

To create a Time Series Plot:

1. Select **Report >> Graph** from the Main Menu or click on the Standard Toolbar
2. Select **Time Series** from the sub-menu that appears.
3. A Time Series Plot dialog will appear. Use it to describe what objects and quantities should be plotted.

![Time Series Plot dialog](image)

The Time Series Plot dialog describes the objects and variable to be graphed in a time series plot. Time series for certain system-wide variables, such as total rainfall, total runoff, total flooding, etc., can also be plotted. Use the dialog as follows:

1. Select a **Start Date** and **End Date** for the plot (the default is the entire simulation period).
2. Choose whether to show time as **Elapsed Time** or as **Date/Time** values.
3. Choose an **Object Category** (Subcatchment, Node, Link, or System) for plotting.
4. If the object category is not System, identify the objects to plot by:
a. selecting the object either on the Study Area Map or in the Data Browser

b. clicking the + button on the dialog to add it to the plot,

c. repeating these steps for any additional objects of the same category.

5. Select a simulated variable to be plotted. The available choices depend on the category of object selected.

6. Click the OK button to create the plot.

A maximum of 6 objects can be selected for a single plot. Objects already selected can be deleted, moved up in the order or moved down in the order by clicking the -, +, and down buttons, respectively.

Profile Plots

A Profile Plot displays the variation in simulated water depth with distance over a connected path of drainage system links and nodes at a particular point in time. Once the plot has been created it will be automatically updated as a new time period is selected using the Map Browser and the Animator toolbar.

To create a Profile Plot:

1. Select Report >> Graph from the main menu or press on the Standard Toolbar
2. Select Profile from the sub-menu that appears.
3. A Profile Plot dialog will appear. Use it to identify the path along which the profile plot is to be drawn.

The Profile Plot dialog is used to specify a path of connected conveyance system links along which a water depth profile versus distance should be drawn. To define a path using the dialog:

1. Enter the ID of the upstream node of the first link in the path in the Start Node edit field (or click on the node on the Study Area Map and then on the + button next to the edit field).
2. Enter the ID of the downstream node of the last link in the path in the End Node edit field (or click the node on the map and then click the + button next to the edit field).
3. Click the Find Path button to have the program automatically identify the path of links between the start and end nodes. These will be listed in the Links in Profile box.
4. You can edit the entries in the Links in Profile if need be, using one link per line.
5. Click the OK button to view the profile plot.
To save the current set of links listed in the dialog for future use:
1. Click the **Save Current Profile** button.
2. Supply a name for the profile when prompted.

To use a previously saved profile:
1. Click the **Use Saved Profile** button.
2. Select the profile to use from the Profile Selection dialog that appears.

Scatter Plots

A Scatter Plot displays the relationship between a pair of variables, such as flow rate in a pipe versus water depth at a node. To create a Scatter Plot:
1. Select **Report >> Graph** from the main menu or press ➙ on the Standard Toolbar
2. Select **Scatter** from the sub-menu that appears.
3. Specify what time interval and what pair of objects and their variables to plot using the Scatter Plot dialog that appears.

The Scatter Plot dialog is used to select the objects and variables to be graphed against one another in a scatter plot. Use the dialog as follows:

1. Select a **Start Date** and **End Date** for the plot (the default is the entire simulation period).
2. Select the following choices for the **X-variable** (the quantity plotted along the horizontal axis):
   - **Object Category** (Subcatchment, Node or Link)
   - **Object ID** (enter a value or click on the object either on the Study Area Map or in the Data Browser and then click the ➔ button on the dialog)
   - **Variable** to plot (choices depend on the category of object selected).
3. Do the same for the **Y-variable** (the quantity plotted along the vertical axis).
4. Click the **OK** button to create the plot.

### 9.5 Customizing a Graph's Appearance

To customize the appearance of a graph:
1. Make the graph the active window (click on its title bar).
2. Select **Report >> Options** from the Main Menu, or click 🗃️ on the Standard Toolbar, or right-click on the graph.

3. Use the Graph Options dialog that appears to customize the appearance of a Time Series or Scatter Plot, or use the Profile Plot Options dialog for a Profile Plot.

**(Graph Options Dialog)**

The Graph Options dialog is used to customize the appearance of a time series plot or a scatter plot. To use the dialog:

1. Select from among the five tabbed pages that cover the following categories of options:
   - General
   - Horizontal Axis
   - Vertical Axis
   - Legend
   - Series

2. Check the **Default** box if you wish to use the current settings as defaults for all new graphs as well.

3. Select **OK** to accept your selections.
Graph Options - General

The following options can be set on the General page of the Graph Options dialog box:

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Panel Color</td>
<td>Color of the panel that contains the graph</td>
</tr>
<tr>
<td>Background Color</td>
<td>Color of graph’s plotting area</td>
</tr>
<tr>
<td>View in 3D</td>
<td>Check if graph should be drawn in 3D</td>
</tr>
<tr>
<td>3D Effect Percent</td>
<td>Degree to which 3D effect is drawn</td>
</tr>
<tr>
<td>Main Title</td>
<td>Text of graph’s main title</td>
</tr>
<tr>
<td>Font</td>
<td>Click to set the font used for the main title</td>
</tr>
</tbody>
</table>

Graph Options - Axes

The Horizontal Axis and Vertical Axis pages of the Graph Options dialog box adjust the way that the axes are drawn on a graph.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum</td>
<td>Sets minimum axis value (minimum data value is shown in parentheses). Can be left blank.</td>
</tr>
<tr>
<td>Maximum</td>
<td>Sets maximum axis value (maximum data value is shown in parentheses). Can be left blank.</td>
</tr>
<tr>
<td>Increment</td>
<td>Sets increment between axis labels. Can be left blank.</td>
</tr>
<tr>
<td>Auto Scale</td>
<td>If checked then Minimum, Maximum, and Increment settings are ignored.</td>
</tr>
<tr>
<td>Gridlines</td>
<td>Selects type of grid line to draw.</td>
</tr>
<tr>
<td>Axis Title</td>
<td>Text of axis title.</td>
</tr>
<tr>
<td>Font</td>
<td>Click to select a font for the axis title.</td>
</tr>
</tbody>
</table>

Graph Options - Legend

The Legend page of the Graph Options dialog box controls how the legend is displayed on the graph.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position</td>
<td>Selects where to place the legend.</td>
</tr>
<tr>
<td>Color</td>
<td>Selects color to use for legend background.</td>
</tr>
<tr>
<td>Symbol Width</td>
<td>Selects width to use (in pixels) to draw the symbol portion of the legend.</td>
</tr>
<tr>
<td>Framed</td>
<td>Places a frame around the legend.</td>
</tr>
<tr>
<td>Visible</td>
<td>Makes the legend visible.</td>
</tr>
</tbody>
</table>

Graph Options - Series

The Series page of the Graph Options dialog box controls how individual data series (or curves) are displayed on a graph. To use this page:

1. Select a data series to work with from the Series combo box.
2. Edit the title used to identify this series in the legend.
3. Click the Font button to change the font used for the legend. (Other legend properties are selected on the Legend page of the dialog.)
4. Select a property of the data series you would like to modify (not all properties are available for some types of graphs). The choices are:
   - Lines
   - Markers
   - Patterns
   - Labels

Profile Plot Options Dialog

The Profile Plot Options dialog is used to customize the appearance of a Profile Plot. The dialog contains two pages:

1. General Page - use this page of the dialog to set options for:
   - colors of various items on the plot (plot window panel, plot background, conduit interior, water depth, and node labels)
   - fill style for water depth
   - grid line styles.

2. Text Page - use this page to:
   - select display options for node labels
   - edit the main and axis titles, including their fonts.

![Profile Plot Options Dialog](image)

Check the Default box if you want these options to apply to all new profile plots when they are first created.
9.6 Viewing Results with a Table

Time series results for selected variables and objects can also be viewed in a tabular format. There are two types of formats available:

- **Table by Object** - tabulates the time series of several variables for a single object (e.g., flow and water depth for a conduit).

<table>
<thead>
<tr>
<th>Date</th>
<th>Time</th>
<th>Flow (CFS)</th>
<th>Depth (ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>01/01/1998</td>
<td>01:00:00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>02:00:00</td>
<td>2.58</td>
<td>0.40</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>03:00:00</td>
<td>4.87</td>
<td>0.55</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>04:00:00</td>
<td>5.41</td>
<td>0.58</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>05:00:00</td>
<td>5.23</td>
<td>0.57</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>06:00:00</td>
<td>1.74</td>
<td>0.33</td>
</tr>
</tbody>
</table>

- **Table by Variable** - tabulates the time series of a single variable for several objects of the same type (e.g., runoff for a group of subcatchments).

<table>
<thead>
<tr>
<th>Date</th>
<th>Time</th>
<th>Subcatch 2</th>
<th>Subcatch 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>01/01/1998</td>
<td>01:00:00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>02:00:00</td>
<td>1.24</td>
<td>1.81</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>03:00:00</td>
<td>2.56</td>
<td>3.82</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>04:00:00</td>
<td>4.52</td>
<td>6.56</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>05:00:00</td>
<td>2.51</td>
<td>3.59</td>
</tr>
<tr>
<td>01/01/1998</td>
<td>06:00:00</td>
<td>0.70</td>
<td>1.03</td>
</tr>
</tbody>
</table>

To create a tabular report:

1. Select **Report >> Table** from the Main Menu or click on the Standard Toolbar.
2. Choose the table format (either **By Object** or **By Variable**) from the sub-menu that appears.
3. Fill in the Table by Object or Table by Variable dialogs to specify what information the table should contain.

The Table by Object dialog is used when creating a time series table of several variables for a single object. Use the dialog as follows:
1. Select a **Start Date** and **End Date** for the table (the default is the entire simulation period).

2. Choose whether to show time as **Elapsed Time** or as **Date/Time** values.

3. Choose an **Object Category** (Subcatchment, Node, Link, or System).

4. Identify a specific object in the category by clicking the object either on the Study Area Map or in the Data Browser and then clicking the button on the dialog. Only a single object can be selected for this type of table.

5. Check off the variables to be tabulated for the selected object. The available choices depend on the category of object selected.

6. Click the **OK** button to create the table.

---

The Table by Variable dialog is used when creating a time series table of a single variable for one or more objects. Use the dialog as follows:

1. Select a **Start Date** and **End Date** for the table (the default is the entire simulation period).

2. Choose whether to show time as **Elapsed Time** or as **Date/Time** values.

3. Choose an **Object Category** (Subcatchment, Node or Link).

4. Select a simulated variable to be tabulated. The available choices depend on the category of object selected.

5. Identify one or more objects in the category by successively clicking the object either on the Study Area Map or in the Data Browser and then clicking the button on the dialog.
6. Click the **OK** button to create the table.

A maximum of 6 objects can be selected for a single table. Objects already selected can be deleted, moved up in the order or moved down in the order by clicking the , , and buttons, respectively.

9.7 **Viewing a Statistics Report**

A Statistics Report can be generated from the time series of simulation results. For a given object and variable this report will do the following:

- segregate the simulation period into a sequence of non-overlapping events, either by day, month, or by flow (or volume) above some minimum threshold value,
- compute a statistical value that characterizes each event, such as the mean, maximum, or total sum of the variable over the event’s time period,
- compute summary statistics for the entire set of event values (mean, standard deviation and skewness),
- perform a frequency analysis on the set of event values.

The frequency analysis of event values will determine the frequency at which a particular event value has occurred and will also estimate a return period for each event value. Statistical analyses of this nature are most suitable for long-term continuous simulation runs.

To generate a Statistics Report:
1. Select **Report >> Statistics** from the Main Menu or click $\sum$ on the Standard Toolbar.

2. Fill in the Statistics Selection dialog that appears, specifying the object, variable, and event definition to be analyzed.

The **Statistics Selection** dialog is used to define the type of statistical analysis to be made on a computed simulation result. It contains the following data fields:

![Statistics Selection dialog]

**Object Category**
Selects the category of object to analyze (Subcatchment, Node, Link or System).

**Object Name**
Specifies the ID name of the object to analyze.

**Variable Analyzed**
Specifies the name of the variable to be analyzed. The choices depend on the object category selected (e.g., rainfall, losses, or runoff for subcatchments; depth, inflow, or overflow for nodes; depth, flow, velocity, or capacity for links; water quality for all categories).

**Event Time Period**
Sets the length of the time period that defines an event. The choices are daily, monthly, or event-dependent. In the latter case, the event period depends on the number of consecutive reporting periods where simulation results are above the threshold values defined below.

**Event Thresholds**
- Rainfall: 0
- Event Volume: 0
- Inter-Event Hours: 6

---

122
Statistic
Selects the event statistic to be analyzed. The available choices depend on the choice of variable to be analyzed and include such quantities as mean value, peak value, event total, event duration, and time between events. For water quality variables the choices include mean concentration, peak concentration, mean loading, peak loading, and event total load.

Analysis Variable Threshold
Sets the minimum value of the variable being analyzed that must be exceeded for a time period to be included in an event.

Event Volume Threshold
Sets the minimum flow volume (or rainfall volume) that must be exceeded for a result to be counted as part of an event. Enter 0 if no volume threshold applies.

Inter-Event Hours
Sets the minimum number of hours that must occur between two separate events. Events with fewer hours are combined together. This value applies only to event-dependent time periods (not to daily or monthly event periods).

The Statistics Report consists of three tabbed pages that contain:
- a table of event summary statistics
- a table of rank-ordered event periods, including their date, duration, and magnitude
- a histogram plot of the chosen event statistic.

![Statistics - System Rainfall](image)
CHAPTER 10 - PRINTING AND COPYING

This chapter describes how to print, copy to the Windows clipboard, or copy to file the contents of the currently active window in the SWMM workspace. This can include the study area map, a graph, a table, or a report.

10.1 Selecting a Printer

To select a printer from among your installed Windows printers and set its properties:
1. Select File >> Page Setup from the Main Menu.
2. Click the Printer button on the Page Setup dialog that appears (see Figure 10-1).
3. Select a printer from the choices available in the combo box in the Print Setup dialog that appears.
4. Click the Properties button to select the appropriate printer properties (which vary with choice of printer).
5. Click OK on each dialog to accept your selections.

![Page Setup dialog]

Figure 10-1. The Margins page of the Page Setup dialog.
10.2 Setting the Page Format

To format the printed page:

1. Select **File >> Page Setup** from the main menu.

2. Use the Margins page of the Page Setup dialog form that appears (Figure 10-1) to:
   - Select a printer.
   - Select the paper orientation (Portrait or Landscape).
   - Set left, right, top, and bottom margins.

3. Use the Headers/Footers page of the dialog box (Figure 10-2) to:
   - Supply the text for a header that will appear on each page.
   - Indicate whether the header should be printed or not and how its text should be aligned.
   - Supply the text for a footer that will appear on each page.
   - Indicate whether the footer should be printed or not and how its text should be aligned.
   - Indicate whether pages should be numbered.

4. Click **OK** to accept your choices.

Figure 10-2. The Headers/Footers page of the Page Setup dialog.
10.3 **Print Preview**

To preview a printout, select **File >> Print Preview** from the Main Menu. A Preview form will appear which shows how each page being printed will appear. While in preview mode, the left mouse button will re-center and zoom in on the image and the right mouse button will re-center and zoom out.

10.4 **Printing the Current View**

To print the contents of the current window being viewed in the SWMM workspace either select **File >> Print** from the Main Menu or click 🖥️ on the Standard Toolbar. The following views can be printed:
- Study Area Map (at the current zoom level)
- Status Report.
- Graphs (Time Series, Profile, and Scatter plots)
- Tabular Reports
- Statistical Reports.

10.5 **Copying to the Clipboard or to a File**

SWMM can copy the text and graphics of the current window being viewed to the Windows clipboard or to a file. Views that can be copied in this fashion include the Study Area Map, graphs, tables, and reports. To copy the current view to the clipboard or to file:

1. If the current view is a table, select the cells of the table to copy by dragging the mouse over them or copy the entire table by selecting **Edit >> Select All** from the Main Menu.
2. Select **Edit >> Copy To** from the Main Menu or click the 🖥️ button on the Standard Toolbar.
3. Select choices from the Copy dialog (see Figure 10-3) that appears and click the **OK** button.
4. If copying to file, enter the name of the file in the **Save As** dialog that appears and click **OK**.

Use the Copy dialog as follows to define how you want your data copied and to where:

1. Select a destination for the material being copied (Clipboard or File)
2. Select a format to copy in:
   - Bitmap (graphics only)
   - Metafile (graphics only)
   - Data (text, selected cells in a table, or data used to construct a graph)
3. Click **OK** to accept your selections or **Cancel** to cancel the copy request.
Figure 10-3. Example of the Copy dialog.
CHAPTER 11 - FILES USED BY SWMM

This section describes the various files that SWMM can utilize. They include: the project file, the report and output files, rainfall files, the climate file, calibration data files, time series files, and interface files. The only file required to run SWMM is the project file; the others are optional.

11.1 Project Files

A SWMM project file is a plain text file that contains all of the data used to describe a study area and the options used to analyze it. The file is organized into sections, where each section generally corresponds to a particular category of object used by SWMM. The contents of the file can be viewed from within SWMM while it is open by selecting Project >> Details from the Main Menu. An existing project file can be opened by selecting File >> Open from the Main Menu and be saved by selecting File >> Save (or File >> Save As).

Normally a SWMM user would not edit the project file directly, since SWMM's graphical user interface can add, delete, or modify a project's data and control settings. However, for large projects where data currently reside in other electronic formats, such as CAD or GIS files, it may be more expeditious to extract data from these sources and save it to a formatted project file before running SWMM. The format of the project file is described in detail in Appendix D of this manual.

After a project file is saved to disk, a settings file will automatically be saved with it. This file has the same name as the project file except that its extension is .ini (e.g., if the project file were named project1.inp then its settings file would have the name project1.ini). It contains various settings used by SWMM's graphical user interface, such as map display options, legend colors and intervals, object default values, and calibration file information. Users should not edit this file. A SWMM project will still load and run even if the settings file is missing.

11.2 Report and Output Files

The report file is a plain text file created after every SWMM run that contains a status report on the results of a run. It can be viewed by selecting Report >> Status from the main menu. If the run was unsuccessful it will contain a list of error messages. For a successful run it will contain:

- the mass continuity errors for runoff quantity and quality as well as for flow and water quality routing,
- summary results tables for all drainage system nodes and links, and
- frequency distributions of time step size and iterations required when Dynamic Wave routing analyses are performed.
The output file is a binary file that contains the numerical results from a successful SWMM run. This file is used by SWMM’s user interface to interactively create time series plots and tables, profile plots, and statistical analyses of a simulation's results.

Whenever a successfully run project is closed (either by opening or creating a new project or by exiting SWMM), the user is asked if the current results should be saved. If the answer is yes, then the report and output files are saved with the same name as the project file, but with extensions of .rpt and .out, respectively. The next time that the project is opened, the results from these files will automatically be available for viewing.

If a project's data were modified before a successful run was made, then when the project is closed the user is asked if the updated project should be saved or not. If the answer is no, then results of the most recent run will not be saved either.

### 11.3 Rainfall Files

SWMM’s rain gage objects can utilize rainfall data stored in external rainfall files. The program currently recognizes the following formats for storing such data:

- **DSI-3240** and related formats which record hourly rainfall at U.S. National Weather Service (NWS) and Federal Aviation Agency stations, available online from the National Climatic Data Center (NCDC) at www.ncdc.noaa.gov/oa/ncdc.html.
- **DSI-3260** and related formats which record fifteen minute rainfall at NWS stations, also available online from NCDC.
- **HLY03** and **HLY21** formats for hourly rainfall at Canadian stations, available online from Environment Canada at www.climate.weatheroffice.ec.gc.ca.
- **FIF21** format for fifteen minute rainfall at Canadian stations, also available online from Environment Canada.
- A standard user-prepared format where each line of the file contains the station ID, year, month, day, hour, minute, and non-zero precipitation reading, all separated by one or more spaces.

An excerpt from a sample user-prepared Rainfall file is as follows:

```
STA01  2004  6  12   00  00  0.12
STA01  2004  6  12   01  00  0.04
STA01  2004  6  22   16  00  0.07
```

When a rain gage is designated as receiving its rainfall data from a file, the user must supply the name of the file, the name of the recording station referenced in the file, and optionally, the date within the file to begin reading rainfall records. For the standard user-prepared format, the rainfall type (e.g., intensity or volume), recording time interval, and depth units must also be supplied as rain gage properties. For the other file types these properties are defined by their respective file format and are automatically recognized by SWMM.
\textbf{11.4 Climate Files}

SWMM can use an external climate file that contains daily air temperature, evaporation, and wind speed data. The program currently recognizes the following formats:

- A DSI-3200 or DSI-3210 file available from the National Climatic Data Center at www.ncdc.noaa.gov/oa/ncdc.html.
- Canadian climate files available from Environment Canada at www.climate.weatheroffice.ec.gc.ca.
- A user-prepared climate file where each line contains a recording station name, the year, month, day, maximum temperature, minimum temperature, and optionally, evaporation rate, and wind speed. If no data are available for any of these items on a given date, then an asterisk should be entered as its value.

When a climate file has days with missing values, SWMM will use the value from the most recent previous day with a recorded value.

For a user-prepared climate file, the data must be in the same units as the project being analyzed. For US units, temperature is in degrees F, evaporation is in inches/day, and wind speed is in miles/hour. For metric units, temperature is in degrees C, evaporation is in mm/day, and wind speed is in km/hour.

\textbf{11.5 Calibration Files}

Calibration files contain measurements of variables at one or more locations that can be compared with simulated values in Time Series Plots. Separate files can be used for each of the following:

- Subcatchment Runoff
- Subcatchment Pollutant Washoff
- Node Depth
- Node Inflow
- Node Water Quality
- Link Flow

Calibration files are registered to a project by selecting \texttt{Project >> Calibration Data} from the main menu (see Section 5.5).

The format of the file is as follows:

1. The name of the first object with calibration data is entered on a single line.
2. Subsequent lines contain the following recorded measurements for the object:
   - measurement date (month/day/year, e.g., 6/21/2004) or number of whole days since the start of the simulation
   - measurement time (hours:minutes) on the measurement date or relative to the number of elapsed days
   - measurement value (for pollutants, a value is required for each pollutant).
3. Follow the same sequence for any additional objects.
An excerpt from an example calibration file is shown below. It contains flow values for two conduits: 1030 and 1602. Note that a semicolon can be used to begin a comment and also in this example, elapsed time rather than the actual measurement date was used.

;Flows for Selected Conduits
;Conduit  Days  Time  Flow
;---------------------------------
1030
  0  0:15  0
  0  0:30  0
  0  0:45  23.88
  0  1:00  94.58
  0  1:15  115.37
1602
  0  0:15  5.76
  0  0:30  38.51
  0  1:00  67.93
  0  1:15  68.01

11.6 Time Series Files

Time series files are external text files that contain data for SWMM's time series objects. Examples of time series data include rainfall, evaporation, inflows to nodes of the drainage system, and water stage at outfall boundary nodes. Normally these data are entered and edited through SWMM's Time Series Editor dialog. However there is an option to import data from an external file into the editor. Creating and editing this file can be done outside of SWMM, using text editors or spreadsheet programs.

The format of a time series file consists of two lines of descriptive text followed by the actual time series data, with one time series value per line. Typically, the first text line identifies the time series and the second line includes a detailed description of the time series. Time series values can either be in date / time / value format or in time / value format, where each entry is separated by one or more spaces or tab characters. For the date / time / value format, dates are entered as month/day/year (e.g., 7/21/2004) and times as 24-hour military time (e.g., 8:30 pm is 20:30). After the first date, additional dates need only be entered whenever a new day occurs. For the time / value format, time can either be decimal hours or military time since the start of a simulation (e.g., 2 days, 4 hours and 20 minutes can be entered as either 52.333 or 52:20). An example of a time series file is shown below:

When preparing rainfall time series files, it is only necessary to enter periods with non-zero rainfall amounts. SWMM interprets the rainfall value as a constant value lasting over the recording interval specified for the rain gage which utilizes the time series. For all other types of time series, SWMM uses interpolation to estimate values at times that fall in between the recorded values.
**11.7 Interface Files**

SWMM can use several different kinds of interface files that contain either externally imposed inputs (e.g., rainfall or infiltration/inflow hydrographs) or the results of previously run analyses (e.g., runoff or routing results). These files can help speed up simulations, simplify comparisons of different loading scenarios, and allow large study areas to be broken up into smaller areas that can be analyzed individually. The different types of interface files that are currently available include:

- rainfall interface file
- runoff interface file
- hotstart file
- RDII interface file
- routing interface files

Consult Section 8.1, *Setting Simulation Options*, for instructions on how to specify interface files for use as input and/or output in a simulation.

**Rainfall and Runoff Files**

The rainfall and runoff interface files are binary files created internally by SWMM that can be saved and reused from one analysis to the next.

The rainfall interface file collates a series of separate rain gage files into a single rainfall data file. Normally a temporary file of this type is created for every SWMM analysis that uses external rainfall data files and is then deleted after the analysis is completed. However, if the same rainfall data are being used with many different analyses, requesting SWMM to save the rainfall interface file after the first run and then reusing this file in subsequent runs can save computation time.

⚠️ The rainfall interface file should not be confused with a rainfall data file. The latter are external text files that provide rainfall time series data to rain gages. The former is a binary file created internally by SWMM that processes the rainfall data files used by a project.
The runoff interface file can be used to save the runoff results generated from a simulation run. If runoff is not affected in future runs, the user can request that SWMM use this interface file to supply runoff results without having to repeat the runoff calculations again.

**Hotstart Files**

Hotstart files are binary files created by SWMM that contain hydraulic and water quality variables for the drainage system at the end of a run. These data consist of the water depth and concentration of each pollutant at each node of the system as well as the flow rate and concentration of each pollutant in each link. The hotstart file saved after a run can be used to define the initial conditions for a subsequent run.

Hotstart files can be used to avoid the initial numerical instabilities that sometimes occur under Dynamic Wave routing. For this purpose they are typically generated by imposing a constant set of base flows (for a natural channel network) or set of dry weather sanitary flows (for a sewer network) over some startup period of time. The resulting hotstart file from this run is then used to initialize a subsequent run where the inflows of real interest are imposed.

It is also possible to both use and save a hotstart file in a single run, starting off the run with one file and saving the ending results to another. The resulting file can then serve as the initial conditions for a subsequent run if need be. This technique can be used to divide up extremely long continuous simulations into more manageable pieces.

**RDII Files**

The RDII interface file is a text file that contains a time series of rainfall-derived infiltration/inflow flows for a specified set of drainage system nodes. This file can be generated from a previous SWMM run when Unit Hydrographs and nodal RDII inflow data have been defined for the project, or it can be created outside of SWMM using some other source of RDII data (e.g., through measurements or output from a different computer program). The format of the file is the same as that of the routing interface file discussed below, where Flow is the only variable contained in the file.

**Routing Files**

A routing interface file stores a time series of flows and pollutant concentrations that are discharged from the outfall nodes of drainage system model. This file can serve as the source of inflow to another drainage system model that is connected at the outfalls of the first system. A **Combine** utility is available on the **File** menu that will combine pairs of routing interface files into a single interface file. This allows very large systems to be broken into smaller sub-systems that can be analyzed separately and linked together through the routing interface file. Figure 11-1 below illustrates this concept.

A single SWMM run can utilize an outflows routing file to save results generated at a system's outfalls, an inflows routing file to supply hydrograph and pollutograph inflows at selected nodes, or both.
Figure 11-1. Example of using the Combine utility to merge Routing files together.

RDII / Routing File Format

RDII interface files and routing interface files have the same text format:

- the first line contains the keyword "SWMM5" (without the quotes)
- a line of text that describes the file (can be blank)
- the time step used for all inflow records (integer seconds)
- the number of variables stored in the file, where the first variable must always be flow rate
- the name and units of each variable (one per line), where flow rate is the first variable listed and is always named FLOW
- the number of nodes with recorded inflow data
- the name of each node (one per line)
- a line of text that provides column headings for the data to follow (can be blank)
- for each node at each time step, a line with:
  1. the name of the node
  2. the date (year, month, and day separated by spaces)
  3. the time of day (hours, minutes, and seconds separated by spaces)
  4. the flow rate followed by the concentration of each quality constituent

Time periods with no values at any node can be skipped. An excerpt from an RDII / routing interface file is shown below.
SWMM5
Example File
300
1
FLOW CFS
2
N1
N2
Node Year Mon Day Hr Min Sec Flow
N1 2002 04 01 00 20 00 0.000000
N2 2002 04 01 00 20 00 0.002549
N1 2002 04 01 00 25 00 0.000000
N2 2002 04 01 00 25 00 0.002549
## APPENDIX A - USEFUL TABLES

### A.1 Units of Measurement

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>US CUSTOMARY</th>
<th>SI METRIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area (Subcatchment)</td>
<td>acres</td>
<td>Hectares</td>
</tr>
<tr>
<td>Area (Storage Unit)</td>
<td>square feet</td>
<td>square meters</td>
</tr>
<tr>
<td>Area (Ponding)</td>
<td>square feet</td>
<td>square meters</td>
</tr>
<tr>
<td>Capillary Suction</td>
<td>inches</td>
<td>millimeters</td>
</tr>
<tr>
<td>Concentration</td>
<td>mg/L</td>
<td>mg/L</td>
</tr>
<tr>
<td></td>
<td>ug/L</td>
<td>ug/L</td>
</tr>
<tr>
<td></td>
<td>Count/L</td>
<td>Count/L</td>
</tr>
<tr>
<td>Decay Constant (Infiltration)</td>
<td>1/hours</td>
<td>1/hours</td>
</tr>
<tr>
<td>Decay Constant (Pollutants)</td>
<td>1/days</td>
<td>1/days</td>
</tr>
<tr>
<td>Depression Storage</td>
<td>inches</td>
<td>millimeters</td>
</tr>
<tr>
<td>Depth</td>
<td>feet</td>
<td>meters</td>
</tr>
<tr>
<td>Diameter</td>
<td>feet</td>
<td>meters</td>
</tr>
<tr>
<td>Discharge Coefficient</td>
<td>dimensionless</td>
<td>dimensionless</td>
</tr>
<tr>
<td>Orifice</td>
<td>CFS/foot^n</td>
<td>CMS/meter^n</td>
</tr>
<tr>
<td>Weir</td>
<td>CFS/foot^n</td>
<td>CMS/meter^n</td>
</tr>
<tr>
<td>Elevation</td>
<td>feet</td>
<td>meters</td>
</tr>
<tr>
<td>Evaporation</td>
<td>inches/day</td>
<td>millimeters/day</td>
</tr>
<tr>
<td>Flow</td>
<td>CFS</td>
<td>CMS</td>
</tr>
<tr>
<td></td>
<td>GPM</td>
<td>LPS</td>
</tr>
<tr>
<td></td>
<td>MGD</td>
<td>MLD</td>
</tr>
<tr>
<td>Head</td>
<td>feet</td>
<td>meters</td>
</tr>
<tr>
<td>Hydraulic Conductivity</td>
<td>inches/hour</td>
<td>millimeters/hour</td>
</tr>
<tr>
<td>Infiltration Rate</td>
<td>inches/hour</td>
<td>millimeters/hour</td>
</tr>
<tr>
<td>Length</td>
<td>feet</td>
<td>meters</td>
</tr>
<tr>
<td>Manning's n</td>
<td>dimensionless</td>
<td>dimensionless</td>
</tr>
<tr>
<td>Pollutant Buildup</td>
<td>mass/length</td>
<td>mass/length</td>
</tr>
<tr>
<td></td>
<td>mass/acre</td>
<td>mass/hectare</td>
</tr>
<tr>
<td>Rainfall Intensity</td>
<td>inches/hour</td>
<td>millimeters/hour</td>
</tr>
<tr>
<td>Rainfall Volume</td>
<td>inches</td>
<td>millimeters</td>
</tr>
<tr>
<td>Slope (Subcatchments)</td>
<td>percent</td>
<td>percent</td>
</tr>
<tr>
<td>Slope (Cross Section)</td>
<td>rise/run</td>
<td>rise/run</td>
</tr>
<tr>
<td>Street Cleaning Interval</td>
<td>days</td>
<td>days</td>
</tr>
<tr>
<td>Volume</td>
<td>cubic feet</td>
<td>cubic meters</td>
</tr>
<tr>
<td>Width</td>
<td>feet</td>
<td>meters</td>
</tr>
</tbody>
</table>
## A.2 Soil Characteristics

<table>
<thead>
<tr>
<th>Soil Texture Class</th>
<th>K</th>
<th>Ψ</th>
<th>φ</th>
<th>FC</th>
<th>WP</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sand</td>
<td>4.74</td>
<td>1.93</td>
<td>0.437</td>
<td>0.062</td>
<td>0.024</td>
</tr>
<tr>
<td>Loamy Sand</td>
<td>1.18</td>
<td>2.40</td>
<td>0.437</td>
<td>0.105</td>
<td>0.047</td>
</tr>
<tr>
<td>Sandy Loam</td>
<td>0.43</td>
<td>4.33</td>
<td>0.453</td>
<td>0.190</td>
<td>0.085</td>
</tr>
<tr>
<td>Loam</td>
<td>0.13</td>
<td>3.50</td>
<td>0.463</td>
<td>0.232</td>
<td>0.116</td>
</tr>
<tr>
<td>Silt Loam</td>
<td>0.26</td>
<td>6.69</td>
<td>0.501</td>
<td>0.284</td>
<td>0.135</td>
</tr>
<tr>
<td>Sandy Clay Loam</td>
<td>0.06</td>
<td>8.66</td>
<td>0.398</td>
<td>0.244</td>
<td>0.136</td>
</tr>
<tr>
<td>Clay Loam</td>
<td>0.04</td>
<td>8.27</td>
<td>0.464</td>
<td>0.310</td>
<td>0.187</td>
</tr>
<tr>
<td>Silty Clay Loam</td>
<td>0.04</td>
<td>10.63</td>
<td>0.471</td>
<td>0.342</td>
<td>0.210</td>
</tr>
<tr>
<td>Sandy Clay</td>
<td>0.02</td>
<td>9.45</td>
<td>0.430</td>
<td>0.321</td>
<td>0.221</td>
</tr>
<tr>
<td>Silty Clay</td>
<td>0.02</td>
<td>11.42</td>
<td>0.479</td>
<td>0.371</td>
<td>0.251</td>
</tr>
<tr>
<td>Clay</td>
<td>0.01</td>
<td>12.60</td>
<td>0.475</td>
<td>0.378</td>
<td>0.265</td>
</tr>
</tbody>
</table>

K = saturated hydraulic conductivity, in/hr  
Ψ = suction head, in.  
φ = porosity, fraction  
FC = field capacity, fraction  
WP = wilting point, fraction

### A.3 NRCS Hydrologic Soil Group Definitions

<table>
<thead>
<tr>
<th>Group</th>
<th>Meaning</th>
<th>Saturated Hydraulic Conductivity (in/hr)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Low runoff potential. Soils having high infiltration rates even when thoroughly wetted and consisting chiefly of deep, well to excessively drained sands or gravels.</td>
<td>≥ 0.45</td>
</tr>
<tr>
<td>B</td>
<td>Soils having moderate infiltration rates when thoroughly wetted and consisting chiefly of moderately deep to deep, moderately well to well-drained soils with moderately fine to moderately coarse textures. E.g., shallow loess, sandy loam.</td>
<td>0.30 - 0.15</td>
</tr>
<tr>
<td>C</td>
<td>Soils having slow infiltration rates when thoroughly wetted and consisting chiefly of soils with a layer that impedes downward movement of water, or soils with moderately fine to fine textures. E.g., clay loams, shallow sandy loam.</td>
<td>0.15 - 0.05</td>
</tr>
<tr>
<td>D</td>
<td>High runoff potential. Soils having very slow infiltration rates when thoroughly wetted and consisting chiefly of clay soils with a high swelling potential, soils with a permanent high water table, soils with a clay-pan or clay layer at or near the surface, and shallow soils over nearly impervious material.</td>
<td>0.05 - 0.00</td>
</tr>
</tbody>
</table>
### A.4 SCS Curve Numbers

<table>
<thead>
<tr>
<th>Land Use Description</th>
<th>Hydrologic Soil Group</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>A</td>
</tr>
<tr>
<td>Cultivated land</td>
<td></td>
</tr>
<tr>
<td>Without conservation treatment</td>
<td>72</td>
</tr>
<tr>
<td>With conservation treatment</td>
<td>62</td>
</tr>
<tr>
<td>Pasture or range land</td>
<td></td>
</tr>
<tr>
<td>Poor condition</td>
<td>68</td>
</tr>
<tr>
<td>Good condition</td>
<td>39</td>
</tr>
<tr>
<td>Meadow</td>
<td></td>
</tr>
<tr>
<td>Good condition</td>
<td>30</td>
</tr>
<tr>
<td>Wood or forest land</td>
<td></td>
</tr>
<tr>
<td>Thin stand, poor cover, no mulch</td>
<td>45</td>
</tr>
<tr>
<td>Good cover</td>
<td>25</td>
</tr>
<tr>
<td>Open spaces, lawns, parks, golf courses, cemeteries, etc.</td>
<td></td>
</tr>
<tr>
<td>Good condition: grass cover on 75% or more of the area</td>
<td>39</td>
</tr>
<tr>
<td>Fair condition: grass cover on 50-75% of the area</td>
<td>49</td>
</tr>
<tr>
<td>Commercial and business areas (85% impervious)</td>
<td>89</td>
</tr>
<tr>
<td>Industrial districts (72% impervious)</td>
<td>81</td>
</tr>
<tr>
<td>Residential</td>
<td></td>
</tr>
<tr>
<td>Average lot size (% Impervious)</td>
<td></td>
</tr>
<tr>
<td>1/8 ac or less (65)</td>
<td>77</td>
</tr>
<tr>
<td>1/4 ac (38)</td>
<td>61</td>
</tr>
<tr>
<td>1/3 ac (30)</td>
<td>57</td>
</tr>
<tr>
<td>1/2 ac (25)</td>
<td>54</td>
</tr>
<tr>
<td>1 ac (20)</td>
<td>51</td>
</tr>
<tr>
<td>Paved parking lots, roofs, driveways, etc.</td>
<td>98</td>
</tr>
<tr>
<td>Streets and roads</td>
<td></td>
</tr>
<tr>
<td>Paved with curbs and storm sewers</td>
<td>98</td>
</tr>
<tr>
<td>Gravel</td>
<td>76</td>
</tr>
<tr>
<td>Dirt</td>
<td>72</td>
</tr>
</tbody>
</table>

**Notes:**

2. Good cover is protected from grazing and litter and brush cover soil.
3. Curve numbers are computed assuming that the runoff from the house and driveway is directed toward the street with a minimum of roof water directed to lawns where additional infiltration could occur.
4. The remaining pervious areas (lawn) are considered to be in good pasture condition for these curve numbers.
5. In some warmer climates of the country a curve number of 95 may be used.
A.5  Depression Storage

<table>
<thead>
<tr>
<th>Surface</th>
<th>Impervious surfaces</th>
<th>Lawn</th>
<th>Pasture</th>
<th>Forest litter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.05 - 0.10 inches</td>
<td>0.10</td>
<td>0.20</td>
<td>0.30 inches</td>
</tr>
</tbody>
</table>


A.6  Manning’s n – Overland Flow

<table>
<thead>
<tr>
<th>Surface</th>
<th>n</th>
</tr>
</thead>
<tbody>
<tr>
<td>Smooth asphalt</td>
<td>0.011</td>
</tr>
<tr>
<td>Smooth concrete</td>
<td>0.012</td>
</tr>
<tr>
<td>Ordinary concrete lining</td>
<td>0.013</td>
</tr>
<tr>
<td>Good wood</td>
<td>0.014</td>
</tr>
<tr>
<td>Brick with cement mortar</td>
<td>0.014</td>
</tr>
<tr>
<td>Vitrified clay</td>
<td>0.015</td>
</tr>
<tr>
<td>Cast iron</td>
<td>0.015</td>
</tr>
<tr>
<td>Corrugated metal pipes</td>
<td>0.024</td>
</tr>
<tr>
<td>Cement rubble surface</td>
<td>0.024</td>
</tr>
<tr>
<td>Fallow soils (no residue)</td>
<td>0.05</td>
</tr>
<tr>
<td>Cultivated soils</td>
<td></td>
</tr>
<tr>
<td>Residue cover &lt; 20%</td>
<td>0.06</td>
</tr>
<tr>
<td>Residue cover &gt; 20%</td>
<td>0.17</td>
</tr>
<tr>
<td>Range (natural)</td>
<td>0.13</td>
</tr>
<tr>
<td>Grass</td>
<td></td>
</tr>
<tr>
<td>Short, prairie</td>
<td>0.15</td>
</tr>
<tr>
<td>Dense</td>
<td>0.24</td>
</tr>
<tr>
<td>Bermuda grass</td>
<td>0.41</td>
</tr>
<tr>
<td>Woods</td>
<td></td>
</tr>
<tr>
<td>Light underbrush</td>
<td>0.40</td>
</tr>
<tr>
<td>Dense underbrush</td>
<td>0.80</td>
</tr>
</tbody>
</table>

Source: McCuen, R. et al. (1996), Hydrology, FHWA-SA-96-067, Federal Highway Administration, Washington, DC
A.7 Manning’s n – Closed Conduits

<table>
<thead>
<tr>
<th>Conduit Material</th>
<th>Manning n</th>
</tr>
</thead>
<tbody>
<tr>
<td>Asbestos-cement pipe</td>
<td>0.011 - 0.015</td>
</tr>
<tr>
<td>Brick</td>
<td>0.013 - 0.017</td>
</tr>
<tr>
<td>Cast iron pipe</td>
<td></td>
</tr>
<tr>
<td>- Cement-lined &amp; seal coated</td>
<td>0.011 - 0.015</td>
</tr>
<tr>
<td>Concrete (monolithic)</td>
<td></td>
</tr>
<tr>
<td>- Smooth forms</td>
<td>0.012 - 0.014</td>
</tr>
<tr>
<td>- Rough forms</td>
<td>0.015 - 0.017</td>
</tr>
<tr>
<td>Concrete pipe</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.011 - 0.015</td>
</tr>
<tr>
<td>Corrugated-metal pipe</td>
<td></td>
</tr>
<tr>
<td>(1/2-in. x 2-2/3-in. corrugations)</td>
<td></td>
</tr>
<tr>
<td>- Plain</td>
<td>0.022 - 0.026</td>
</tr>
<tr>
<td>- Paved invert</td>
<td>0.018 - 0.022</td>
</tr>
<tr>
<td>- Spun asphalt lined</td>
<td>0.011 - 0.015</td>
</tr>
<tr>
<td>Plastic pipe (smooth)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.011 - 0.015</td>
</tr>
<tr>
<td>Vitrified clay</td>
<td></td>
</tr>
<tr>
<td>- Pipes</td>
<td>0.011 - 0.015</td>
</tr>
<tr>
<td>- Liner plates</td>
<td>0.013 - 0.017</td>
</tr>
</tbody>
</table>

A.8 Manning’s n – Open Channels

<table>
<thead>
<tr>
<th>Channel Type</th>
<th>Manning n</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lined Channels</td>
<td></td>
</tr>
<tr>
<td>- Asphalt</td>
<td>0.013 - 0.017</td>
</tr>
<tr>
<td>- Brick</td>
<td>0.012 - 0.018</td>
</tr>
<tr>
<td>- Concrete</td>
<td>0.011 - 0.020</td>
</tr>
<tr>
<td>- Rubble or riprap</td>
<td>0.020 - 0.035</td>
</tr>
<tr>
<td>- Vegetal</td>
<td>0.030 - 0.40</td>
</tr>
<tr>
<td>Excavated or dredged</td>
<td></td>
</tr>
<tr>
<td>- Earth, straight and uniform</td>
<td>0.020 - 0.030</td>
</tr>
<tr>
<td>- Earth, winding, fairly uniform</td>
<td>0.025 - 0.040</td>
</tr>
<tr>
<td>- Rock</td>
<td>0.030 - 0.045</td>
</tr>
<tr>
<td>- Unmaintained</td>
<td>0.050 - 0.140</td>
</tr>
<tr>
<td>Natural channels (minor streams, top width at flood stage &lt; 100 ft)</td>
<td></td>
</tr>
<tr>
<td>- Fairly regular section</td>
<td>0.030 - 0.070</td>
</tr>
<tr>
<td>- Irregular section with pools</td>
<td>0.040 - 0.100</td>
</tr>
</tbody>
</table>


A.9 Water Quality Characteristics of Urban Runoff

<table>
<thead>
<tr>
<th>Constituent</th>
<th>Event Mean Concentrations</th>
</tr>
</thead>
<tbody>
<tr>
<td>TSS (mg/L)</td>
<td>180 - 548</td>
</tr>
<tr>
<td>BOD (mg/L)</td>
<td>12 - 19</td>
</tr>
<tr>
<td>COD (mg/L)</td>
<td>82 - 178</td>
</tr>
<tr>
<td>Total P (mg/L)</td>
<td>0.42 - 0.88</td>
</tr>
<tr>
<td>Soluble P (mg/L)</td>
<td>0.15 - 0.28</td>
</tr>
<tr>
<td>TKN (mg/L)</td>
<td>1.90 - 4.18</td>
</tr>
<tr>
<td>NO2/NO3-N (mg/L)</td>
<td>0.86 - 2.2</td>
</tr>
<tr>
<td>Total Cu (ug/L)</td>
<td>43 - 118</td>
</tr>
<tr>
<td>Total Pb (ug/L)</td>
<td>182 - 443</td>
</tr>
<tr>
<td>Total Zn (ug/L)</td>
<td>202 - 633</td>
</tr>
</tbody>
</table>

## APPENDIX B - VISUAL OBJECT PROPERTIES

### B.1 Rain Gage Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned rain gage name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the rain gage on the Study Area Map. If left blank then the rain gage will not appear on the map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the rain gage on the Study Area Map. If left blank then the rain gage will not appear on the map.</td>
</tr>
<tr>
<td>Description</td>
<td>Optional description of the rain gage.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the rain gage.</td>
</tr>
</tbody>
</table>
| Rain Format   | Format in which the rain data are supplied:  
|               | **INTENSITY**: each rainfall value is an average rate in inches/hour (or mm/hour) over the recording interval,  
|               | **VOLUME**: each rainfall value is the volume of rain that fell in the recording interval (in inches or millimeters),  
|               | **CUMULATIVE**: each rainfall value represents the cumulative rainfall that has occurred since the start of the last series of non-zero values (in inches or millimeters). |
| Rain Interval | Time interval between gage readings in either decimal hours or hours:minutes format. |
| Snow Catch Factor | Factor that corrects gage readings for snowfall. |
| Data Source   | Source of rainfall data; either **TIMESERIES** for user-supplied time series data or **FILE** for an external data file. |
| TIME SERIES   | Name of time series with rainfall data if Data Source selection was **TIMESERIES**; leave blank otherwise (double-click to edit the series). |
| - Series Name | Name of external file containing rainfall data. |
| - Station No. | Recording gage station number. |
| - Rain Units  | Depth units (**IN** or **MM**) for rainfall values in the file. |
### B.2 Subcatchment Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>User-assigned subcatchment name.</td>
</tr>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the subcatchment's centroid on the Study Area Map. If left blank then the subcatchment will not appear on the map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the subcatchment's centroid on the Study Area Map. If left blank then the subcatchment will not appear on the map.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the subcatchment.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the subcatchment.</td>
</tr>
<tr>
<td>Rain Gage</td>
<td>Name of the rain gage associated with the subcatchment.</td>
</tr>
<tr>
<td>Outlet</td>
<td>Name of the node or subcatchment which receives the subcatchment's runoff.</td>
</tr>
<tr>
<td>Area</td>
<td>Area of the subcatchment (acres or hectares).</td>
</tr>
<tr>
<td>Width</td>
<td>Characteristic width of the overland flow path for sheet flow runoff (feet or meters). An initial estimate of the characteristic width is given by the subcatchment area divided by the average maximum overland flow length. The maximum overland flow length is the length of the flow path from the inlet to the furthest drainage point of the subcatchment. Maximum lengths from several different possible flow paths should be averaged. These paths should reflect slow flow, such as over pervious surfaces, more than rapid flow over pavement, for example. Adjustments should be made to the width parameter to produce good fits to measured runoff hydrographs.</td>
</tr>
<tr>
<td>% Slope</td>
<td>Average percent slope of the subcatchment.</td>
</tr>
<tr>
<td>% Imperv</td>
<td>Percent of land area which is impervious.</td>
</tr>
<tr>
<td>N-Imperv</td>
<td>Manning's n for overland flow over the impervious portion of the subcatchment (see Section A.6 for typical values).</td>
</tr>
<tr>
<td>N-Perv</td>
<td>Manning's n for overland flow over the pervious portion of the subcatchment (see Section A.6 for typical values).</td>
</tr>
<tr>
<td>Dstore-Imperv</td>
<td>Depth of depression storage on the impervious portion of the subcatchment (inches or millimeters) (see Section A.5 for typical values).</td>
</tr>
<tr>
<td>Dstore-Perv</td>
<td>Depth of depression storage on the pervious portion of the subcatchment (inches or millimeters) (see Section A.5 for typical values).</td>
</tr>
<tr>
<td>% Zero-Imperv</td>
<td>Percent of the impervious area with no depression storage.</td>
</tr>
<tr>
<td>Subarea Routing</td>
<td>Choice of internal routing of runoff between pervious and impervious areas: IMPERV: runoff from pervious area flows to impervious area, PERV: runoff from impervious area flows to pervious area, OUTLET: runoff from both areas flows directly to outlet.</td>
</tr>
<tr>
<td>Percent Routed</td>
<td>Percent of runoff routed between subareas.</td>
</tr>
<tr>
<td>----------------</td>
<td>-----------------------------------------</td>
</tr>
<tr>
<td>Infiltration</td>
<td>Click the ellipsis button (or press Enter) to edit infiltration parameters for the subcatchment.</td>
</tr>
<tr>
<td>Groundwater</td>
<td>Click the ellipsis button (or press Enter) to edit groundwater flow parameters for the subcatchment.</td>
</tr>
<tr>
<td>Snow Pack</td>
<td>Name of snow pack parameter set (if any) assigned to the subcatchment.</td>
</tr>
<tr>
<td>Initial Buildup</td>
<td>Click the ellipsis button (or press Enter) to specify initial quantities of pollutant buildup over the subcatchment.</td>
</tr>
<tr>
<td>Land Uses</td>
<td>Click the ellipsis button (or press Enter) to assign land uses to the subcatchment.</td>
</tr>
<tr>
<td>Curb Length</td>
<td>Total length of curbs in the subcatchment (any length units). Used only when pollutant buildup is normalized to curb length.</td>
</tr>
</tbody>
</table>

### B.3 Junction Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned junction name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the junction on the Study Area Map. If left blank then the junction will not appear on the map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the junction on the Study Area Map. If left blank then the junction will not appear on the map.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the junction.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the junction.</td>
</tr>
<tr>
<td>Inflows</td>
<td>Click the ellipsis button (or press Enter) to assign time series, dry weather, or RDII inflows to the junction.</td>
</tr>
<tr>
<td>Treatment</td>
<td>Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the node.</td>
</tr>
<tr>
<td>Invert El.</td>
<td>Invert elevation of the junction (feet or meters).</td>
</tr>
<tr>
<td>Max. Depth</td>
<td>Maximum depth of junction (i.e., from ground surface to invert) (feet or meters).</td>
</tr>
<tr>
<td>Initial Depth</td>
<td>Depth of water at the junction at the start of the simulation (feet or meters).</td>
</tr>
<tr>
<td>Surcharge Depth</td>
<td>Additional depth of water beyond the maximum depth that is allowed before the junction floods (feet or meters). This parameter can be used to simulate bolted manhole covers.</td>
</tr>
<tr>
<td>Ponded Area</td>
<td>Area occupied by ponded water atop the junction after flooding occurs (sq. feet or sq. meters). If the Allow Ponding Simulation Option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the conveyance system when capacity exists.</td>
</tr>
</tbody>
</table>
### B.4 Outfall Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>User-assigned outfall name.</td>
</tr>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the outfall on the Study Area Map. If left blank then the outfall will not appear on the map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the outfall on the Study Area Map. If left blank then the outfall will not appear on the map.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the outfall.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the outfall.</td>
</tr>
<tr>
<td>Inflows</td>
<td>Click the ellipsis button (or press Enter) to assign time series, dry weather, or RDII inflows to the outfall.</td>
</tr>
<tr>
<td>Treatment</td>
<td>Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the node.</td>
</tr>
<tr>
<td>Invert El.</td>
<td>Invert elevation of the outfall (feet or meters).</td>
</tr>
</tbody>
</table>
| Tide Gate       | YES - tide gate present which prevents backflow  
|                 | NO - no tide gate present                                                  |
| Type            | Type of outfall boundary condition:                                         |
|                 | **FREE**: outfall stage determined by minimum of critical flow depth and normal flow depth in the connecting conduit |
|                 | **NORMAL**: outfall stage based on normal flow depth in connecting conduit |
|                 | **FIXED**: outfall stage set to a fixed value                                |
|                 | **TIDAL**: outfall stage given by a table of tide elevation versus time of day |
|                 | **TIMESERIES**: outfall stage supplied from a time series of elevations.     |
| Fixed Stage     | Water elevation for a **FIXED** type of outfall (feet or meters).           |
| Tidal Curve Name| Name of the Tidal Curve relating water elevation to hour of the day for a **TIDAL** outfall (double-click to edit the curve). |
| Time Series Name| Name of time series containing time history of outfall elevations for a **TIMESERIES** outfall (double-click to edit the series). |
### B.5 Flow Divider Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned divider name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the divider on the Study Area Map. If left blank then the divider will not appear on the map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the divider on the Study Area Map. If left blank then the divider will not appear on the map.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the divider.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the divider.</td>
</tr>
<tr>
<td>Inflows</td>
<td>Click the ellipsis button (or press Enter) to assign time series, dry weather, or RDII inflows to the divider.</td>
</tr>
<tr>
<td>Treatment</td>
<td>Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the node.</td>
</tr>
<tr>
<td>Invert El.</td>
<td>Invert elevation of the divider (feet or meters).</td>
</tr>
<tr>
<td>Max. Depth</td>
<td>Maximum depth of divider (i.e., from ground surface to invert) (feet or meters).</td>
</tr>
<tr>
<td>Initial Depth</td>
<td>Depth of water at the divider at the start of the simulation (feet or meters).</td>
</tr>
<tr>
<td>Surcharge Depth</td>
<td>Additional depth of water beyond the maximum depth that is allowed before the junction floods (feet or meters). This parameter can be used to simulate bolted manhole covers.</td>
</tr>
<tr>
<td>Ponded Area</td>
<td>Area occupied by ponded water atop the junction after flooding occurs (sq. feet or sq. meters). If the Allow Ponding Simulation Option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the conveyance system when capacity exists.</td>
</tr>
<tr>
<td>Diverted Link</td>
<td>Name of link which receives the diverted flow.</td>
</tr>
<tr>
<td>Type</td>
<td>Type of flow divider. Choices are: CUTOFF (all flow above a fixed value is diverted), OVERFLOW (all flow above the non-diverted conduit’s full depth flow is diverted), TABULAR (a diversion curve specifies diverted flow as a function of the total flow entering the divider), WEIR (diverted flow is linearly proportional to the total flow in excess of a given amount).</td>
</tr>
</tbody>
</table>

#### CUTOFF Divider

- **Cutoff Flow**
  Cutoff flow value used for a **CUTOFF** divider (flow units).

#### TABULAR Divider

- **Curve Name**
  Name of Diversion Curve used with a **TABULAR** divider (double-click to edit the curve).

#### WEIR Divider
- Min. Flow  Minimum flow at which diversion begins for a **WEIR** divider (flow units).
- Max. Depth  Vertical height of **WEIR** opening (feet or meters)
- Coefficient  Product of **WEIR**'s discharge coefficient and its length. Weir coefficients are typically in the range of 2.65 to 3.10 per foot, for flows in CFS.

### B.6 Storage Unit Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned storage unit name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the storage unit on the Study Area Map. If left blank then the storage unit will not appear on the map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the storage unit on the Study Area Map. If left blank then the storage unit will not appear on the map.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the storage unit.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the storage unit.</td>
</tr>
<tr>
<td>Inflows</td>
<td>Click the ellipsis button (or press Enter) to assign time series, dry weather, or RDII inflows to the storage unit.</td>
</tr>
<tr>
<td>Treatment</td>
<td>Click the ellipsis button (or press Enter) to edit a set of treatment functions for pollutants entering the storage unit.</td>
</tr>
<tr>
<td>Invert El.</td>
<td>Elevation of the bottom of the storage unit (feet or meters).</td>
</tr>
<tr>
<td>Max. Depth</td>
<td>Maximum depth of the storage unit (feet or meters).</td>
</tr>
<tr>
<td>Initial Depth</td>
<td>Initial depth of water in the storage unit at the start of the simulation (feet or meters).</td>
</tr>
<tr>
<td>Ponded Area</td>
<td>Surface area occupied by ponded water atop the storage unit once the water depth exceeds the maximum depth (sq. feet or sq. meters). If the <strong>Allow Ponding</strong> analysis option is turned on, a non-zero value of this parameter will allow ponded water to be stored and subsequently returned to the drainage system when capacity exists.</td>
</tr>
<tr>
<td>Evap. Factor</td>
<td>The fraction of the potential evaporation from the storage unit’s water surface that is actually realized.</td>
</tr>
<tr>
<td>Shape Curve</td>
<td>Method of describing the geometric shape of the storage unit. <strong>FUNCTIONAL</strong> shape uses the function [ \text{Area} = A \times (\text{Depth})^B + C ] to describe how surface area varies with depth. <strong>TABULAR</strong> shape uses a tabulated area versus depth curve. In either case, depth is measured in feet (or meters) and surface area in sq. feet (or sq. meters).</td>
</tr>
</tbody>
</table>
FUNCTIONAL

- Coeff.  A-value in the functional relationship between surface area and storage depth.
- Exponent  B-value in the functional relationship between surface area and storage depth.
- Constant  C-value in the functional relationship between surface area and storage depth.

TABULAR

- Curve Name  Name of the Storage Curve containing the relationship between surface area and storage depth (double-click to edit the curve).

B.7  Conduit Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned conduit name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Node</td>
<td>Name of node on the inlet end of the conduit (which is normally the end at higher elevation).</td>
</tr>
<tr>
<td>Outlet Node</td>
<td>Name of node on the outlet end of the conduit (which is normally the end at lower elevation).</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the conduit.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the conduit.</td>
</tr>
<tr>
<td>Shape</td>
<td>Click the ellipsis button (or press Enter) to edit the geometric properties of the conduit’s cross section.</td>
</tr>
<tr>
<td>Length</td>
<td>Conduit length (feet or meters).</td>
</tr>
<tr>
<td>Roughness</td>
<td>Manning's roughness coefficient (see Section A.7 for closed conduit values or Section A.8 for open channel values).</td>
</tr>
<tr>
<td>Inlet Offset</td>
<td>Height of the conduit invert above the node invert at the upstream end of the conduit (feet or meters).</td>
</tr>
<tr>
<td>Outlet Offset</td>
<td>Height of the conduit invert above the node invert at the downstream end of the conduit (feet or meters).</td>
</tr>
<tr>
<td>Initial Flow</td>
<td>Initial flow in the conduit at the start of the simulation (flow units).</td>
</tr>
<tr>
<td>Entry Loss Coeff.</td>
<td>Head loss coefficient associated with energy losses at the entrance of the conduit.</td>
</tr>
<tr>
<td>Exit Loss Coeff.</td>
<td>Coefficient associated with energy losses at the exit of the conduit.</td>
</tr>
<tr>
<td>Avg. Loss Coeff.</td>
<td>Coefficient associated with energy losses along the length of the conduit.</td>
</tr>
<tr>
<td>Flap Gate</td>
<td><strong>YES</strong> if a flap gate exists which prevents backflow through the conduit, or <strong>NO</strong> if no flap gate exists.</td>
</tr>
</tbody>
</table>
### B.8 Pump Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned pump name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Node</td>
<td>Name of node on the inlet side of the pump.</td>
</tr>
<tr>
<td>Outlet Node</td>
<td>Name of node on the outlet side of the pump.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the pump.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the pump.</td>
</tr>
<tr>
<td>Pump Curve</td>
<td>Name of the Pump Curve which contains the pump's operating data (double-click to edit the curve).</td>
</tr>
<tr>
<td>Initial Status</td>
<td>Status of the pump (<strong>ON</strong> or <strong>OFF</strong>) at the start of the simulation.</td>
</tr>
</tbody>
</table>

### B.9 Orifice Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned orifice name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Node</td>
<td>Name of node on the inlet side of the orifice.</td>
</tr>
<tr>
<td>Outlet Node</td>
<td>Name of node on the outlet side of the orifice.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the orifice.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the orifice.</td>
</tr>
<tr>
<td>Type</td>
<td>Type of orifice (<strong>SIDE</strong> or <strong>BOTTOM</strong>).</td>
</tr>
<tr>
<td>Shape</td>
<td>Orifice shape (<strong>CIRCULAR</strong> or <strong>RECT_CLOSED</strong>).</td>
</tr>
<tr>
<td>Height</td>
<td>Height of orifice when fully open (feet or meters). Corresponds to the diameter of a circular orifice or the height of a rectangular orifice.</td>
</tr>
<tr>
<td>Width</td>
<td>Width of rectangular orifice when fully opened (feet or meters)</td>
</tr>
<tr>
<td>Crest Height</td>
<td>Height of bottom of orifice above invert of inlet node (feet or meters).</td>
</tr>
<tr>
<td>Discharge Coeff.</td>
<td>Discharge coefficient (unitless). A typical value is 0.65.</td>
</tr>
<tr>
<td>Flap Gate</td>
<td><strong>YES</strong> if a flap gate exists which prevents backflow through the orifice, or <strong>NO</strong> if no flap gate exists.</td>
</tr>
</tbody>
</table>
# B.10 Weir Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned weir name.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Node</td>
<td>Name of node on inlet side of weir.</td>
</tr>
<tr>
<td>Outlet Node</td>
<td>Name of node on outlet side of weir.</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the weir.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the weir.</td>
</tr>
<tr>
<td>Type</td>
<td>Weir type: <strong>TRANSVERSE</strong>, <strong>SIDEFLOW</strong>, <strong>V-NOTCH</strong>, or <strong>TRAPEZOIDAL</strong></td>
</tr>
<tr>
<td>Height</td>
<td>Vertical height of weir opening (feet or meters)</td>
</tr>
<tr>
<td>Length</td>
<td>Horizontal length of weir opening (feet or meters)</td>
</tr>
<tr>
<td>Side Slope</td>
<td>Slope (width-to-height) of side walls for a <strong>V-NOTCH</strong> or <strong>TRAPEZOIDAL</strong> weir.</td>
</tr>
<tr>
<td>Crest Height</td>
<td>Height of bottom of weir opening from invert of inlet node (feet or meters).</td>
</tr>
<tr>
<td>Discharge Coeff.</td>
<td>Discharge coefficient for flow through the central portion of the weir (for flow in CFS when using US units or CMS when using SI units). Typical values are: 3.33 US (1.84 SI) for sharp crested transverse weirs, 2.5 - 3.3 US (1.38 - 1.83 SI) for broad crested rectangular weirs, 2.4 - 2.8 US (1.35 - 1.55 SI) for V-notch (triangular) weirs</td>
</tr>
<tr>
<td>Flap Gate</td>
<td><strong>YES</strong> if the weir has a flap gate that prevents backflow, <strong>NO</strong> if it does not.</td>
</tr>
<tr>
<td>End Coeff.</td>
<td>Discharge coefficient for flow through the triangular ends of a <strong>TRAPEZOIDAL</strong> weir. See the recommended values for V-notch weirs listed above.</td>
</tr>
<tr>
<td>End Contractions</td>
<td>Number of end contractions for a <strong>TRANSVERSE</strong> or <strong>TRAPEZOIDAL</strong> weir whose length is shorter than the channel it is placed in. Either 0, 1, or 2 depending on if no ends, one end, or both ends are beveled in from the side walls.</td>
</tr>
</tbody>
</table>
B.11 Outlet Properties

<table>
<thead>
<tr>
<th>Name</th>
<th>User-assigned outlet name</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Node</td>
<td>Name of node on inflow side of outlet</td>
</tr>
<tr>
<td>Outlet Node</td>
<td>Name of node on discharge side of outlet</td>
</tr>
<tr>
<td>Description</td>
<td>Click the ellipsis button (or press Enter) to edit an optional description of the outlet.</td>
</tr>
<tr>
<td>Tag</td>
<td>Optional label used to categorize or classify the outlet</td>
</tr>
<tr>
<td>Height</td>
<td>Height of outlet above inlet node invert (ft or m)</td>
</tr>
<tr>
<td>Flap Gate</td>
<td><strong>YES</strong> if a flap gate exists which prevents backflow through the outlet, or <strong>NO</strong> if no flap gate exists.</td>
</tr>
<tr>
<td>Rating Curve</td>
<td>Method of defining flow ($Q$) as a function of head ($h$) across the outlet. A <strong>FUNCTIONAL</strong> curve uses a power function ($Q = Ah^B$) to describe this relation while a <strong>TABULAR</strong> curve uses a tabulated curve of flow versus head values.</td>
</tr>
</tbody>
</table>

**FUNCTIONAL**

- Coefficient | Coefficient (A) for the functional relationship between head and flow rate. |
- Exponent     | Exponent (B) used for the functional relationship between head and flow rate. |

**TABULAR**

- Curve Name | Name of Rating Curve containing the relationship between head and flow rate (double-click to edit the curve). |

B.12 Map Label Properties

<table>
<thead>
<tr>
<th>Text</th>
<th>Text of label.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-Coordinate</td>
<td>Horizontal location of the upper-left corner of the label on the Study Area Map.</td>
</tr>
<tr>
<td>Y-Coordinate</td>
<td>Vertical location of the upper-left corner of the label on the Study Area Map.</td>
</tr>
<tr>
<td>Anchor Node</td>
<td>Name of node (or subcatchment) that anchors the label's position when the map is zoomed in (i.e., the pixel distance between the node and the label remains constant). Leave blank if anchoring is not used.</td>
</tr>
<tr>
<td>Font</td>
<td>Click the ellipsis button (or press Enter) to modify the font used to draw the label.</td>
</tr>
</tbody>
</table>
C.1 Aquifer Editor

The Aquifer Editor is invoked whenever a new aquifer object is created or an existing aquifer object is selected for editing. It contains the following data fields:

- **Name**
  User-assigned aquifer name.

- **Porosity**
  Volume of voids / total soil volume (volumetric fraction).

- **Wilting Point**
  Soil moisture content at which plants cannot survive (volumetric fraction).

- **Field Capacity**
  Soil moisture content after all free water has drained off (volumetric fraction).

- **Conductivity**
  Soil's saturated hydraulic conductivity (in/hr or mm/hr).

- **Conductivity Slope**
  Average slope of conductivity versus soil moisture content curve (in/hr or mm/hr).

- **Tension Slope**
  Average slope of soil tension versus soil moisture content curve (inches or mm).

- **Upper Evaporation Fraction**
  Fraction of total evaporation available for evapotranspiration in the upper unsaturated zone.

- **Lower Evaporation Depth**
  Maximum depth into the lower saturated zone over which evapotranspiration can occur (ft or m).

- **Lower Groundwater Loss Rate**
  Rate of percolation from saturated zone to deep groundwater (in/hr or mm/hr).
Bottom Elevation
Elevation of the bottom of the aquifer (ft or m).

Water Table Elevation
Elevation of the water table in the aquifer at the start of the simulation (ft or m).

Unsaturated Zone Moisture
Moisture content of the unsaturated upper zone of the aquifer at the start of the simulation (volumetric fraction) (cannot exceed soil porosity).

C.2 Climatology Editor

The Climatology Editor is used to enter values for various climate-related variables required by certain SWMM simulations. The dialog is divided into five tabbed pages, where each page provides a separate editor for a specific category of climate data.

Temperature Page

The Temperature page of the Climatology Editor dialog is used to specify the source of temperature data used for snowmelt computations. There are three choices available:

No Data: Select this choice if snowmelt is not being simulated.
**Time Series:** Select this choice if the variation in temperature over the simulation period will be described by one of the project's time series. Also enter (or select) the name of the time series. Click the button to make the Time Series Editor appear for the selected time series.

**External Climate File:** Select this choice if min/max daily temperatures will be read from an external climate file. Also enter the name of the file (or click the button to search for the file).

### Evaporation Page

![Evaporation Page](image)

The Evaporation page of the Climatology Editor dialog is used to supply evaporation rates, in inches/day (or mm/day), for a study area. There are four choices for specifying these rates:

**Constant:** Use this choice if evaporation remains constant over time. Enter the value in the edit box provided.

**Time Series:** Select this choice if evaporation rates will be specified in a time series. Enter or select the name of the time series.
in the dropdown combo box provided. Click the button to bring up the Time Series editor for the selected series. Note that for each date specified in the time series, the evaporation rate remains constant at the value supplied for that date until the next date in the series is reached (i.e., interpolation is not used on the series).

**From Climate File:** This choice indicates that evaporation rates will be read from the same climate file that was specified for temperature. Enter values for monthly pan coefficients in the data grid provided.

**Monthly Averages:** Use this choice to supply an average rate for each month of the year. Enter the value for each month in the data grid provided. Note that rates remain constant within each month.

**Wind Speed Page**

The Wind Speed page of the Climatology Editor dialog is used to provide average monthly wind speeds. These are used when computing snowmelt rates under rainfall conditions. Melt rates increase with increasing wind speed. Units of wind speed are miles/hour for US units and km/hour for metric units. There are two choices for specifying wind speeds:
From Climate File: Wind speeds will be read from the same climate file that was specified for temperature.

Monthly Averages: Wind speed is specified as an average value that remains constant in each month of the year. Enter a value for each month in the data grid provided. The default values are all zero.

Snowmelt Page

The Snowmelt page of the Climatology Editor dialog is used to supply values for the following parameters related to snow melt calculations:

Dividing Temperature Between Snow and Rain
Enter the temperature below which precipitation falls as snow instead of rain. Use degrees F for US units or degrees C for metric units.

ATI (Antecedent Temperature Index) Weight
This parameter reflects the degree to which heat transfer within a snow pack during non-melt periods is affected by prior air temperatures. Smaller values reflect a thicker surface layer of snow which results in reduced rates of heat transfer. Values must be between 0 and 1, and the default is 0.5.
**Negative Melt Ratio**
This is the ratio of the heat transfer coefficient of a snow pack during non-melt conditions to the coefficient during melt conditions. It must be a number between 0 and 1. The default value is 0.6.

**Elevation Above MSL**
Enter the average elevation above mean sea level for the study area, in feet or meters. This value is used to provide a more accurate estimate of atmospheric pressure. The default is 0.0, which results in a pressure of 29.9 inches Hg. The effect of wind on snow melt rates during rainfall periods is greater at higher pressures, which occur at lower elevations.

**Latitude**
Enter the latitude of the study area in degrees North. This number is used when computing the hours of sunrise and sunset, which in turn are used to extend min/max daily temperatures into continuous values. The default is 50 degrees North.

**Longitude Correction**
This is a correction, in minutes of time, between true solar time and the standard clock time. It depends on a location's longitude (θ) and the standard meridian of its time zone (SM) through the expression 4(θ-SM). This correction is used to adjust the hours of sunrise and sunset when extending daily min/max temperatures into continuous values. The default value is 0.

**Areal Depletion Page**
The Areal Depletion page of the Climatology Editor Dialog is used to specify points on the Areal Depletion Curves for both impervious and pervious surfaces within a project's study area. These curves define the relation between the area that remains snow covered and snow pack depth. Each curve is defined by 10 equal increments of relative depth ratio between 0 and 0.9. (Relative depth ratio is the ratio of an area's current snow depth to the depth at which there is 100% areal coverage).

Enter values in the data grid provided for the fraction of each area that remains snow covered at each specified relative depth ratio. Valid numbers must be between 0 and 1, and be increasing with increasing depth ratio.

Clicking the **Natural Area** button fills the grid with values that are typical of natural areas. Clicking the **No Depletion** button will fill the grid with all 1’s, indicating that no areal depletion occurs. This is the default for new projects.
C.3 Control Rules Editor

RULE PUMP1A
IF NODE SU1 DEPTH >= 4
THEN PUMP PUMP1 status = ON
PRIORITY 1

RULE PUMP1B
IF NODE SU1 DEPTH < 1
THEN PUMP PUMP1 status = OFF
PRIORITY 1
The Control Rules Editor is invoked whenever a new control rule is created or an existing rule is selected for editing. The editor contains a memo field where the entire collection of control rules is displayed and can be edited.

Control Rule Format

Each control rule is a series of statements of the form:

\[
\begin{align*}
\text{RULE} & \quad \text{ruleID} \\
\text{IF} & \quad \text{condition}_1 \\
\text{AND} & \quad \text{condition}_2 \\
\text{OR} & \quad \text{condition}_3 \\
\text{AND} & \quad \text{condition}_4 \\
\text{Etc.} & \\
\text{THEN} & \quad \text{action}_1 \\
\text{AND} & \quad \text{action}_2 \\
\text{Etc.} & \\
\text{ELSE} & \quad \text{action}_3 \\
\text{AND} & \quad \text{action}_4 \\
\text{Etc.} & \\
\text{PRIORITY} & \quad \text{value}
\end{align*}
\]

where keywords are shown in boldface and ruleID is an ID label assigned to the rule, condition\_n is a Condition Clause, action\_n is an Action Clause, and value is a priority value (e.g., a number from 1 to 5). The formats used for Condition and Action clauses are discussed below.

Only the RULE, IF and THEN portions of a rule are required; the ELSE and PRIORITY portions are optional.

Blank lines between clauses are permitted and any text to the right of a semicolon is considered a comment.

When mixing AND and OR clauses, the OR operator has higher precedence than AND, i.e., IF A or B and C is equivalent to IF (A or B) and C.

If the interpretation was meant to be IF A or (B and C) then this can be expressed using two rules as in

IF A THEN ...
IF B and C THEN ...

The PRIORITY value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always
has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

**Condition Clauses**

A Condition Clause of a control rule has the following format:

```
object id attribute relation value
```

where
- **object** = a category of object
- **id** = the object's ID label
- **attribute** = an attribute or property of the object
- **relation** = a relational operator (=, <>, <, <=, >, >=)
- **value** = an attribute value

Some examples of condition clauses are:

```
NODE N23  DEPTH  >  10
PUMP P45  STATUS =  OFF
SIMULATION CLOCKTIME = 22:45:00
```

The objects and attributes that can appear in a condition clause are as follows:

<table>
<thead>
<tr>
<th>Object</th>
<th>Attributes</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>NODE</td>
<td>DEPTH</td>
<td>numerical value</td>
</tr>
<tr>
<td></td>
<td>HEAD</td>
<td>numerical value</td>
</tr>
<tr>
<td></td>
<td>INFLOW</td>
<td>numerical value</td>
</tr>
<tr>
<td>LINK</td>
<td>FLOW</td>
<td>numerical value</td>
</tr>
<tr>
<td></td>
<td>DEPTH</td>
<td>numerical value</td>
</tr>
<tr>
<td>PUMP</td>
<td>STATUS</td>
<td>ON or OFF</td>
</tr>
<tr>
<td></td>
<td>FLOW</td>
<td>numerical value</td>
</tr>
<tr>
<td>ORIFICE</td>
<td>SETTING</td>
<td>fraction open</td>
</tr>
<tr>
<td>WEIR</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SIMULATION</td>
<td>TIME</td>
<td>elapsed time in decimal hours or hr:min:sec</td>
</tr>
<tr>
<td></td>
<td>DATE</td>
<td>month/day/year</td>
</tr>
<tr>
<td></td>
<td>CLOCKTIME</td>
<td>time of day in hr:min:sec</td>
</tr>
</tbody>
</table>
**Action Clauses**

An Action Clause of a control rule can have one of the following formats:

- **PUMP** \( id \) **STATUS** = **ON/OFF**
- **ORIFICE** \( id \) **SETTING** = **value**
- **WEIR** \( id \) **SETTING** = **value**

where **SETTING** refers to the fractional amount that an orifice is fully open or to the fractional amount of the original height between the crest and the top of a weir that remains (i.e., weir control is accomplished by moving the crest height up and down).

Some examples of action clauses are:

- \( \text{PUMP P67 STATUS = OFF} \)
- \( \text{ORIFICE O212 SETTING = 0.5} \)

**C.4 Cross-Section Editor**

The Cross-Section Editor dialog is used to specify the shape and dimensions of a conduit's cross-section.

When a shape is selected from the dropdown combo box an appropriate set of edit fields appears for describing the dimensions of that shape. Length dimensions are in units of feet for US units and meters for SI units. Slope values represent ratios of horizontal to vertical distance. The Barrels field specifies how many identical parallel conduits exist between its end nodes.

If an Irregular shaped section is chosen, a drop-down edit box will appear where you can enter or select the name of a Transect object that describes the cross-section's geometry. Clicking the Edit button next to the edit box will bring up the Transect Editor which allows you to edit the transect's data.
C.5 Curve Editor

The Curve Editor dialog is invoked whenever a new curve object is created or an existing curve object is selected for editing. The editor adapts itself to the category of curve being edited (Storage, Tidal, Diversion, Pump, or Rating). To use the Curve Editor:

- Enter values for the following data entry fields:
  - **Name**: Name of the curve.
  - **Type** (Pump Curves Only): Choice of pump curve type as described in Section 3.2 - Pumps.
  - **Description**: Optional comment or description of what the curve represents. Click the button to launch a multi-line comment editor if more than one line is needed.
  - **Data Grid**: The curve's X,Y data.

- Click the View button to see a graphical plot of the curve drawn in a separate window.

- If additional rows are needed in the Data Grid, simply press the Enter key when in the last row.

- Right-clicking over the Data Grid will make a popup Edit menu appear. It contains commands to cut, copy, insert, and paste selected cells in the grid as well as options to insert or delete a row.
You can also click the **Load** button to load in a curve that was previously saved to file or click the **Save** button to save the current curve's data to a file.

### C.6 Groundwater Flow Editor

The Groundwater Flow Editor dialog is invoked when the Groundwater property of a subcatchment is being edited. It is used to link a subcatchment to both an aquifer and to a node of the drainage system that exchanges groundwater with the aquifer. It also specifies coefficients that determine the rate of groundwater flow between the aquifer and the node. These coefficients (A1, A2, B1, B2, and A3) appear in the following equation that computes groundwater flow as a function of groundwater and surface water heads:

\[
Q_{gw} = A1(H_{gw} - E)^{B1} - A2(H_{sw} - E)^{B2} + A3H_{gw}H_{sw}
\]

where:

- \(Q_{gw}\) = groundwater flow (cfs per acre or cms per hectare)
- \(H_{gw}\) = elevation of groundwater table (ft or m)
- \(H_{sw}\) = elevation of surface water at receiving node (ft or m)
- \(E\) = elevation of node invert (ft or m).

The properties listed in the editor are as follows:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aquifer Name</td>
<td></td>
</tr>
<tr>
<td>Receiving Node</td>
<td></td>
</tr>
<tr>
<td>Surface Elevation</td>
<td>0</td>
</tr>
<tr>
<td>Groundwater Flow Coef.</td>
<td>0</td>
</tr>
<tr>
<td>Groundwater Flow Expon.</td>
<td>0</td>
</tr>
<tr>
<td>Surf. Water Flow Coef.</td>
<td>0</td>
</tr>
<tr>
<td>Surf. Water Flow Expon.</td>
<td>0</td>
</tr>
<tr>
<td>Surface-GW Interaction Coef.</td>
<td>0</td>
</tr>
<tr>
<td>Fixed Surf. Water Depth</td>
<td>0</td>
</tr>
</tbody>
</table>

Name of aquifer object that lies below subcatchment (leave blank for no groundwater).

The properties listed in the editor are as follows:
Aquifer Name
Name of aquifer object that supplies groundwater. Leave this field blank if you want the subcatchment not to generate any groundwater flow.

Receiving Node
Name of node that receives groundwater from the aquifer.

Surface Elevation
Elevation of ground surface for the subcatchment that lies above the aquifer in feet or meters.

Groundwater Flow Coefficient
Value of A1 in the groundwater flow formula.

Groundwater Flow Exponent
Value of B1 in the groundwater flow formula.

Surface Water Flow Coefficient
Value of A2 in the groundwater flow formula.

Surface Water Flow Exponent
Value of B2 in the groundwater flow formula.

Surface-GW Interaction Coefficient
Value of A3 in the groundwater flow formula.

Fixed Surface Water Depth
Fixed depth of surface water at the receiving node (feet or meters) (set to zero if surface water depth will vary as computed by flow routing).

The values of the flow coefficients must be in units that are consistent with the groundwater flow units of cfs/acre for US units or cms/ha for metric units.

⚠️ If groundwater flow is simply proportional to the difference in groundwater and surface water heads, then set the Groundwater and Surface Water Flow Exponents (B1 and B2) to 1.0, set the Groundwater Flow Coefficient (A1) to the proportionality factor, set the Surface Water Flow Coefficient (A2) to the same value as A1, and set the Interaction Coefficient (A3) to zero.

C.7 Infiltration Editor

The Infiltration Editor dialog is used to specify values for the parameters that describe the rate at which rainfall infiltrates into the upper soil zone in a subcatchment's pervious area. It is invoked when editing the Infiltration property of a subcatchment. The infiltration parameters depend on which infiltration model was selected for the project: Horton, Green-Ampt, or Curve Number. The choice of infiltration model can be made either by editing the project's Simulation Options (see Section 8.1) or by changing the project's Default Properties (see Section 5.4).
The following data fields appear in the Infiltration Editor for Horton infiltration:

**Max. Infil. Rate**
Maximum infiltration rate on the Horton curve (in/hr or mm/hr). Representative values are as follows:

1. **DRY soils (with little or no vegetation):**
   - Sandy soils: 5 in/hr
   - Loam soils: 3 in/hr
   - Clay soils: 1 in/hr

2. **DRY soils (with dense vegetation):**
   - Multiply values in A. by 2

3. **MOIST soils:**
   - Soils which have drained but not dried out (i.e., field capacity):
     - Divide values from A and B by 3.
   - Soils close to saturation:
     - Choose value close to min. infiltration rate.
   - Soils which have partially dried out:
     - Divide values from A and B by 1.5 - 2.5.

**Min. Infil. Rate**
Minimum infiltration rate on the Horton curve (in/hr or mm/hr). Equivalent to the soil’s saturated hydraulic conductivity. See the Soil Characteristics Table in Section A.2 for typical values.
**Decay Const.**
Infiltration rate decay constant for the Horton curve (1/hours). Typical values range between 2 and 7.

**Drying Time**
Time in days for a fully saturated soil to dry completely. Typical values range from 2 to 14 days.

**Max. Infil. Vol.**
Maximum infiltration volume possible (inches or mm, 0 if not applicable). Can be estimated as the difference between a soil's porosity and its wilting point times the depth of the infiltration zone.

**Green-Ampt Infiltration Parameters**
The following data fields appear in the Infiltration Editor for Green-Ampt infiltration:

**Suction Head**
Average value of soil capillary suction along the wetting front (inches or mm).

**Conductivity**
Soil saturated hydraulic conductivity (in/hr or mm/hr).

**Initial Deficit**
Difference between soil porosity and initial moisture content (a fraction). For a completely drained soil, it is the difference between the soil's porosity and its field capacity.

Typical values for all of these parameters can be found in the Soil Characteristics Table in Section A.2.

**Curve Number Infiltration Parameters**
The following data fields appear in the Infiltration Editor for Curve Number infiltration:

**Curve Number**
This is the SCS curve number which is tabulated in the publication *SCS Urban Hydrology for Small Watersheds*, 2nd Ed., (TR-55), June 1986. Consult the Curve Number Table (Section A.4) for a listing of values by soil group, and the accompanying Soil Group Table (Section A.3) for the definitions of the various groups.

**Conductivity**
The soil's saturated hydraulic conductivity (in/hr or mm/hr). Typical ranges are shown in both the Soil Group Table (Section A.3) and in the Soil Characteristics Table (Section A.2).

**Drying Time**
The number of days it takes a fully saturated soil to dry. Typical values range between 2 and 14 days.
C.8 Inflows Editor

The Inflows Editor dialog is used to assign Direct, Dry Weather, and RDII inflow into a node of the drainage system. It is invoked whenever the Inflows property of a Node object is selected in the Property Editor. The dialog consists of three tabbed pages that provide a special editor for each type of inflow.

Direct Inflows Page

The Direct page on the Inflows Editor dialog is used to specify time series of direct external inflow and pollutant inflows entering a node of the drainage system. The dialog consists of the following input fields:

Constituent
Selects the constituent (FLOW or one of the project's specified pollutants) whose direct inflow will be described.

Time Series
Specifies the name of the time series that contains inflow data for the selected constituent. If left blank then no direct inflow will occur for the selected constituent at the node in question. You can click the button to bring up the Time Series Editor dialog for the selected time series.

Inflow Type
For pollutants, selects the type of inflow data contained in the time series as being either concentrations or mass flows.
Conversion Factor
Numerical conversion factor used to convert a pollutant mass flow rate in the time series data into concentration times flow units used by the project. For example, if the time series data were in pounds per day and the pollutant concentration defined in the project was mg/L while the flow units for the project were CFS, then the conversion factor value would be 454,000 mg/lb / 86400 sec/day = 5.25.

More than one constituent can be edited while the dialog is active by simply selecting another choice for the Constituent property. However, if the Cancel button is clicked then any changes made to all constituents will be ignored.

If a pollutant is assigned a direct inflow in terms of concentration, then one must also assign a direct inflow to flow, otherwise no pollutant inflow will occur. If pollutant inflow is defined in terms of mass, then a flow inflow time series is not required.

Dry Weather Inflows Page

The Dry Weather Inflows page of the Inflows Editor dialog is used to specify a continuous source of dry weather flow entering a node of the drainage system. The dialog consists of the following input fields:

Constituent
Selects the constituent (FLOW or one of the project's specified pollutants) whose dry weather inflow will be specified.
Average Value
Specifies the average (or baseline) value of the dry weather inflow of the constituent in the relevant units (flow units for flow, concentration units for pollutants). Leave blank if there is no dry weather flow for the selected constituent.

Time Patterns
Specifies the names of the time patterns to be used to allow the dry weather flow to vary in a periodic fashion by month of the year, by day of the week, and by time of day (for both weekdays and weekends). One can either type in a name or select a previously defined pattern from the dropdown list of each combo box. Up to four different types of patterns can be assigned. You can click the 🚀 button next to each Time Pattern field to edit the respective pattern.

More than one constituent can be edited while the dialog is active by simply selecting another choice for the Constituent property. However, if the Cancel button is clicked then any changes made to all constituents will be ignored.

RDII Inflow Page

The RDII page of the Inflows Editor dialog is used to specify RDII (rainfall-derived infiltration/inflow) for the node in question. The editor contains the following two input fields:

Unit Hydrograph Group
Enter (or select from the dropdown list) the name of the Unit Hydrograph group that applies to the node in question. The unit hydrographs in the group are used in

The RDII page of the Inflows Editor dialog is used to specify RDII (rainfall-derived infiltration/inflow) for the node in question. The editor contains the following two input fields:

Unit Hydrograph Group
Enter (or select from the dropdown list) the name of the Unit Hydrograph group that applies to the node in question. The unit hydrographs in the group are used in
combination with the group's assigned rain gage to develop a time series of RDII inflows per unit area over the period of the simulation. Leave this field blank to indicate that the node receives no RDII inflow. Clicking the button will launch the Unit Hydrograph Editor for the UH group specified.

**Sewershed Area**
Enter the area (in acres or hectares) of the sewershed that contributes RDII to the node in question. Note this area will typically be only a small, localized portion of the subcatchment area that contributes surface runoff to the node.

### C.9 Initial Buildup Editor

The Initial Buildup Editor is invoked from the Property Editor when editing the Initial Buildup property of a subcatchment. It specifies the amount of pollutant buildup existing over the subcatchment at the start of the simulation. The editor consists of a data entry grid with two columns. The first column lists the name of each pollutant in the project and the second column contains edit boxes for entering the initial buildup values. If no buildup value is supplied for a pollutant, it is assumed to be 0. The units for buildup are either pounds per acre when US units are in use or kilograms per hectare when SI metric units are in use.

<table>
<thead>
<tr>
<th>Pollutant</th>
<th>Initial Buildup (lbs/ac)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TSS</td>
<td>10</td>
</tr>
<tr>
<td>Lead</td>
<td></td>
</tr>
</tbody>
</table>

If a non-zero value is specified for the initial buildup of a pollutant, it will override any initial buildup computed from the Antecedent Dry Days parameter specified on the Dates page of the Simulation Options dialog.
C.10 Land Use Editor

The Land Use Editor dialog is used to define a category of land use for the study area and to define its pollutant buildup and washoff characteristics. The dialog contains three tabbed pages of land use properties:

- General Page (provides land use name and street sweeping parameters)
- Buildup Page (defines rate at which pollutant buildup occurs)
- Washoff Page (defines rate at which pollutant washoff occurs)

General Page

![Land Use Editor dialog]

The General page of the Land Use Editor dialog describes the following properties of a particular land use category:

- **Land Use Name**
  The name assigned to the land use.

- **Description**
  An optional comment or description of the land use (click the ellipsis button or press Enter to edit).

- **Street Sweeping Interval**
  Days between street sweeping within the land use.
**Street Sweeping Availability**  
Fraction of the buildup of all pollutants that is available for removal by sweeping.

**Last Swept**  
Number of days since last swept at the start of the simulation.

If street sweeping does not apply to the land use, then the last three properties can be left blank.

**Buildup Page**

The Buildup page of the Land Use Editor dialog describes the properties associated with pollutant buildup over the land during dry weather periods. These consist of:

**Pollutant**  
Select the pollutant whose buildup properties are being edited.

**Function**  
The type of buildup function to use for the pollutant. The choices are **NONE** for no buildup, **POW** for power function buildup, **EXP** for exponential function buildup and **SAT** for saturation function buildup. See the discussion of Pollutant Buildup in Section 3.3.9 for explanations of these different functions. Select **NONE** if no buildup occurs.

**Max. Buildup**  
The maximum buildup that can occur, expressed as lbs (or kg) of the pollutant per unit of the normalizer variable (see below). This is the same as the C1 coefficient used in the buildup formulas discussed in Section 3.3.9.
Rate Constant
The time constant that governs the rate of pollutant buildup. This is the C2 coefficient in the Power and Exponential buildup formulas discussed in Section 3.3.9. For Power buildup its units are mass / days raised to a power, while for Exponential buildup its units are 1/days.

Power/Sat. Constant
The exponent C3 used in the Power buildup formula, or the half-saturation constant C2 used in the Saturation buildup formula discussed in Section 3.3.9. For the latter case, its units are days.

Normalizer
The variable to which buildup is normalized on a per unit basis. The choices are either land area (in acres or hectares) or curb length. Any units of measure can be used for curb length, as long as they remain the same for all subcatchments in the project.

When there are multiple pollutants, the user must select each pollutant separately from the Pollutant dropdown list and specify its pertinent buildup properties.

Washoff Page

The Washoff page of the Land Use Editor dialog describes the properties associated with pollutant washoff over the land use during wet weather events. These consist of:

Pollutant
The name of the pollutant whose washoff properties are being edited.

The Washoff function: \( \text{EXP} = \text{exponential}, \text{RC} = \text{rating curve}, \text{EMC} = \text{event mean concentration.} \)
**Function**
The choice of washoff function to use for the pollutant. The choices are:
- **NONE** no washoff
- **EXP** exponential washoff
- **RC** rating curve washoff
- **EMC** event-mean concentration washoff.
The formula for each of these functions is discussed in Section 3.3.9 under the Pollutant Washoff topic.

**Coefficient**
This is the value of C1 in the exponential and rating curve formulas, or the event-mean concentration.

**Exponent**
The exponent used in the exponential and rating curve washoff formulas.

**Cleaning Efficiency**
The street cleaning removal efficiency (percent) for the pollutant. It represents the fraction of the amount that is available for removal on the land use as a whole (set on the General page of the editor) which is actually removed.

**BMP Efficiency**
Removal efficiency (percent) associated with any Best Management Practice that might have been implemented. The washoff load computed at each time step is simply reduced by this amount.

As with the Buildup page, each pollutant must be selected in turn from the Pollutant dropdown list and have its pertinent washoff properties defined.

### C.11 Land Use Assignment Editor

The Land Use Assignment editor is invoked from the Property Editor when editing the Land Uses property of a subcatchment. Its purpose is to assign land uses to the subcatchment for water quality simulations. The percent of land area in the subcatchment covered by each land use is entered next to its respective land use category. If the land use is not present its field can be left blank. The percentages entered do not necessarily have to add up to 100.
C.12 Pollutant Editor

The Pollutant Editor is invoked when a new pollutant object is created or an existing pollutant is selected for editing. It contains the following fields:
Name
The name assigned to the pollutant.

Units
The concentration units (\text{mg/L, ug/L}, or \#/L (counts/L)) in which the pollutant concentration is expressed.

Rain Concentration
Concentration of the pollutant in rain water (concentration units).

GW Concentration
Concentration of the pollutant in ground water (concentration units).

I&I Concentration
Concentration of the pollutant in any Infiltration/Inflow (concentration units)

Decay Coefficient
First-order decay coefficient of the pollutant (1/days).

Snow Only
\text{YES} if pollutant buildup occurs only when snowfall occurs, \text{NO} otherwise (default is \text{NO}).

Co-Pollutant
Name of another pollutant whose runoff concentration contributes to the runoff concentration of the current pollutant.

Co-Fraction
Fraction of the co-pollutant's runoff concentration that contributes to the runoff concentration of the current pollutant.

An example of a co-pollutant relationship would be where the runoff concentration of a particular heavy metal is some fixed fraction of the runoff concentration of suspended solids. In this case suspended solids would be declared as the co-pollutant for the heavy metal.

C13. Snow Pack Editor

The Snow Pack Editor is invoked when a new snow pack object is created or an existing snow pack is selected for editing. The editor contains a data entry field for the snow pack’s name and two tabbed pages, one for snow pack parameters and one for snow removal parameters.

Snow Pack Parameters Page

The Parameters page of the Snow Pack Editor dialog provides snow melt parameters and initial conditions for snow that accumulates over three different types of areas: the impervious area that is plowable (i.e., subject to snow removal), the remaining
impervious area, and the entire pervious area. The page contains a data entry grid which has a column for each type of area and a row for each of the following parameters:

**Snow Pack Editor**

<table>
<thead>
<tr>
<th>Subcatchment Surface Type</th>
<th>Plowable</th>
<th>Impervious</th>
<th>Pervious</th>
</tr>
</thead>
<tbody>
<tr>
<td>Min. Melt Coeff. (in/hr/deg F)</td>
<td>0.001</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>Max. Melt Coeff. (in/hr/deg F)</td>
<td>0.001</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>Base Temperature (deg F)</td>
<td>32.0</td>
<td>32.0</td>
<td>32.0</td>
</tr>
<tr>
<td>Fraction Free Water Capacity</td>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
</tr>
<tr>
<td>Initial Snow Depth (in)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>Initial Free Water (in)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>Depth at 100% Cover (in)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Fraction of Impervious Area That is Plowable: 0.5

**Min. Melt Coefficient**
The degree-day snow melt coefficient that occurs on December 21. Units are either in/hr-deg F or mm/hr-deg C.

**Max. Melt Coefficient**
The degree-day snow melt coefficient that occurs on June 21. Units are either in/hr-deg F or mm/hr-deg C. For a short term simulation of less than a week or so it is acceptable to use a single value for both the minimum and maximum melt coefficients.

The minimum and maximum snow melt coefficients are used to estimate a melt coefficient that varies by day of the year. The latter is used in the following degree-day equation to compute the melt rate for any particular day:

\[
\text{Melt Rate} = (\text{Melt Coefficient}) \times (\text{Air Temperature} - \text{Base Temperature}).
\]

**Base Temperature**
Temperature at which snow begins to melt (degrees F or C).
Fraction Free Water Capacity
The volume of a snow pack's pore space which must fill with melted snow before liquid runoff from the pack begins, expressed as a fraction of snow pack depth.

Initial Snow Depth
Depth of snow at the start of the simulation (water equivalent depth in inches or millimeters).

Initial Free Water
Depth of melted water held within the pack at the start of the simulation (inches or mm). This number should be at or below the product of the initial snow depth and the fraction free water capacity.

Depth at 100% Cover
The depth of snow beyond which the entire area remains completely covered and is not subject to any areal depletion effect (inches or mm).

Fraction of Impervious Area That is Plowable
The fraction of impervious area that is plowable and therefore is not subject to areal depletion.

Snow Removal Parameters Page

The Snow Removal page of the Snow Pack Editor describes how snow removal occurs within the Plowable area of a snow pack. The following parameters govern this process:
Depth at which snow removal begins (in or mm)
No removal occurs at depths below this and the fractions specified below are applied to the snow depths in excess of this number.

Fraction transferred out of the watershed
The fraction of excess snow depth that is removed from the system (and does not become runoff).

Fraction transferred to the impervious area
The fraction of excess snow depth that is added to snow accumulation on the pack's impervious area.

Fraction transferred to the pervious area
The fraction of excess snow depth that is added to snow accumulation on the pack's pervious area.

Fraction converted to immediate melt
The fraction of excess snow depth that becomes liquid water which runs onto any subcatchment associated with the snow pack.
Fraction moved to another subcatchment
The fraction of excess snow depth which is added to the snow accumulation on some other subcatchment. The name of the subcatchment must also be provided.

C.14 Time Pattern Editor

The Time Pattern Editor is invoked when a new time pattern object is created or an existing time pattern is selected for editing. The editor contains that following data entry fields:

Name
Enter the name assigned to the time pattern.

Type
Select the type of time pattern being specified.

Description
You can provide an optional comment or description for the time pattern. If more than one line is needed, click the button to launch a multi-line comment editor.

Multipliers
Enter a value for each multiplier. The number and meaning of the multipliers changes with the type of time pattern selected:

- MONTHLY One multiplier for each month of the year.
- DAILY One multiplier for each day of the week.
- HOURLY One multiplier for each hour from 12 midnight to 11 PM.
- WEEKEND Same as for HOURLY except applied to weekend days.
In order to maintain an average dry weather flow or pollutant concentration at its specified value (as entered on the Inflows Editor), the multipliers for a pattern should sum to 1.0.

C.15 Time Series Editor

The Time Series Editor is invoked whenever a new time series object is created or an existing time series is selected for editing. To use the Time Series Editor:

- Enter values for the following data entry fields:
  
  **Name**
  Name of the time series.

  **Description**
  Optional comment or description of what the time series represents. Click the button to launch a multi-line comment editor if more than one line is needed.

  **Date Column**
  Optional date (in month/day/year format) of the time series values (only needed at points in time where a new date occurs).

  **Time Column**
  If dates are used, enter the military time of day for each time series value (as hours:minutes or decimal hours). If dates are not used, enter time as hours since the start of the simulation.

  **Value Column**
  The time series’ numerical values.
- Click the **View** button to see a graphical plot of the time series data drawn in a separate window.

- If more rows in the data entry grid are needed as the series extends out in time, simply press the **Enter** key when in the last row to append a new row to the grid.

- Right-clicking over the Data Grid will make a popup Edit menu appear. It contains commands to cut, copy, insert, and paste selected cells in the grid as well as options to insert or delete a row.

![Time Series Editor](image)

Note that there are two methods for describing the occurrence time of time series data:

a. as calendar date/time of day (which requires that at least one date, at the start of the series, be entered in the Date column)

b. as elapsed hours since the start of the simulation (where the Date column remains empty).

You can also click the **Load** button to load in a time series that was previously saved to file or click the **Save** button to save the current time series' data to a file.
C.16 Title/Notes Editor

The Title/Notes editor is invoked when a project’s Title/Notes data category is selected for editing. As shown below, the editor contains a multi-line edit field where a description of a project can be entered. It also contains a check box used to indicate whether or not the first line of notes should be used as a header for printing.

![Project Title/Notes](image)

C.17 Transect Editor

The Transect Editor is invoked when a new transect object is created or an existing transect is selected for editing. It contains the following data entry fields:

Name
The name assigned to the transect.

Description
An optional comment or description of the transect.

Station/Elevation Data Grid
Values of distance from the left side of the channel along with the corresponding elevation of the channel bottom as one moves across the channel from left to right, looking in the downstream direction. Up to 1500 data values can be entered.

Roughness
Values of Manning's roughness for the left overbank, right overbank, and main channel portion of the transect. The overbank roughness values can be zero if no overbank exists.

Bank Stations
The distance values appearing in the Station/Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.

Modifiers
The Stations modifier is a factor by which the distance between each station will be multiplied when the transect data is processed by SWMM. Use a value of 0 if no such
factor is needed. The Elevations modifier is a constant value that will be added to each elevation value.

Right-clicking over the Data Grid will make a popup Edit menu appear. It contains commands to cut, copy, insert, and paste selected cells in the grid as well as options to insert or delete a row.

Clicking the View button will bring up a window that illustrates the shape of the transect cross section.

C.18 Treatment Editor

The Treatment Editor is invoked whenever the Treatment property of a node is selected from the Property Editor. It displays a list of the project's pollutants with an edit box next to each as shown below.
Enter a valid treatment expression in the box next to each pollutant which receives treatment. Refer to the Treatment topic in Section 3.3 to learn what constitutes a valid treatment expression.

### C.19 Unit Hydrograph Editor

The Unit Hydrograph Editor is invoked whenever a new unit hydrograph object is created or an existing one is selected for editing. It is used to specify the shape parameters and rain gage for a group of triangular unit hydrographs. These hydrographs are used to compute rainfall-derived infiltration/inflow (RDII) flow at selected nodes of the drainage system. A UH group can contain up to 12 sets of unit hydrographs (one for each month of the year), and each set can consist of up to 3 individual hydrographs (for short-term, intermediate-term, and long-term responses, respectively). The editor, shown below, contains the following data entry fields:

#### Name of UH Group
Enter the name assigned to the UH Group.

#### Rain Gage
Type in (or select from the dropdown list) the name of the rain gage that supplies rainfall data to the unit hydrographs in the group.

#### Month
Select a month from the list box for which hydrograph parameters will be defined. Select **ALL MONTHS** to specify a default set of hydrographs that apply to all months of the year. Then select specific months that need to have special hydrographs defined.
R-T-K Parameters Grid
Use this data grid to provide the R-T-K shape parameters for each set of unit hydrographs in selected months of the year. The first row is used to specify parameters for a short-term response hydrograph (i.e., small value of T), the second for a medium-term response hydrograph, and the third for a long-term response hydrograph (largest value of T). It is not required that all three hydrographs be defined. The shape parameters for each UH consist of:

R: the fraction of rainfall volume that enters the sewer system
T: the time from the onset of rainfall to the peak of the UH in hours
K: the ratio of time to recession of the UH to the time to peak

If a grid cell is left empty its corresponding parameter value is assumed to be 0.

Right-clicking over the data grid will make a popup Edit menu appear. It contains commands to cut, copy, and paste text to or from selected cells in the grid.
A P P E N D I X  D -  C O M M A N D  L I N E  S W M M

D.1 General Instructions

EPA SWMM can also be run as a console application from the command line within a DOS window. In this case the study area data are placed into a text file and results are written to a text file. The command line for running SWMM in this fashion is:

```
swmm5 inpfile rptfile outfile
```

where `inpfile` is the name of the input file, `rptfile` is the name of the output report file, and `outfile` is the name of an optional binary output file that stores results in a special binary format. If the latter file is not needed then just the input and report file names should be supplied. As written, the above command assumes that you are working in the directory in which EPA SWMM was installed or that this directory has been added to the PATH variable in your user profile (or the autoexec.bat file in older versions of Windows). Otherwise full pathnames for the executable swmm5.exe and the files on the command line must be used.

D.2 Input File Format

The input file for command line SWMM has the same format as the project file used by the Windows version of the program. Figure D-1 illustrates an example SWMM5 input file. It is organized in sections, where each section begins with a keyword enclosed in brackets. The various keywords are listed below.

```
[TITLE] project title
[OPTIONS] analysis options
[REPORT] output reporting instructions
[FILES] interface file options
[RAINAGES] rain gage information
[HYDROGRAPHS] unit hydrograph data used to construct RDII inflows
[EVAPORATION] evaporation data
[Temperature] air temperature and snow melt data
[SUBCATCHMENTS] basic subcatchment information
[SUBAREAS] subcatchment impervious/pervious sub-area data
[INfiltration] subcatchment infiltration parameters
[AQUIFERS] groundwater aquifer parameters
[GROUNDWATER] subcatchment groundwater parameters
[SNOWPACKS] subcatchment snow pack parameters
[JUNCTIONS] junction node information
[OUTFALLS] outfall node information
```
The sections can appear in any arbitrary order in the input file, and not all sections must
be present. Each section can contain one or more lines of data. Blank lines may appear
anywhere in the file. A semicolon (;) can be used to indicate that what follows on the line
is a comment, not data. Data items can appear in any column of a line. Observe how in
Figure D-1 these features were used to create a tabular appearance for the data, complete
with column headings.

Section keywords can appear in mixed lower and upper case, and only the first four
characters (plus the open bracket) are used to distinguish one keyword from another (e.g.,
[DIVIDERS] and [Divi] are equivalent). An option is available in the [OPTIONS]
section to choose flow units from among cubic feet per second (CFS), gallons per minute
(GPM), million gallons per day (MGD), cubic meters per second (CMS), liters per
second, (LPS), or million liters per day (MLD). If cubic feet or gallons are chosen for
flow units, then US units are used for all other quantities. If cubic meters or liters are
chosen, then metric units apply to all other quantities. The default flow units are CFS.

A detailed description of the data in each section of the input file will now be given.
When listing the format of a line of data, mandatory keywords are shown in boldface
while optional items appear in parentheses. A list of keywords separated by a slash
(YES/NO) means that only one of the words should appear in the data line.
Figure D-1. Example SWMM project file (continued on next page).
Figure D-1. Example SWMM project file (continued from previous page).
Section: [TITLE]

Purpose: Attaches a descriptive title to the to the problem being analyzed.

Format: Any number of lines may be entered. The first line will be used as a page header in the output report.

Section: [OPTIONS]

Purpose: Provides values for various analysis options.

Format:

<table>
<thead>
<tr>
<th>Option</th>
<th>Format</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLOW_UNITS</td>
<td>CFS / GPM / MGD / CMS / LPS / MLD</td>
</tr>
<tr>
<td>INFILTRATION</td>
<td>HORTON / GREEN_AMPT / CURVE_NUMBER</td>
</tr>
<tr>
<td>FLOW_ROUTING</td>
<td>STEADY / KINWAVE / DYNWAVE</td>
</tr>
<tr>
<td>ALLOW_PONDING</td>
<td>YES / NO</td>
</tr>
<tr>
<td>START_DATE</td>
<td>month/day/year</td>
</tr>
<tr>
<td>START_TIME</td>
<td>hours:minutes</td>
</tr>
<tr>
<td>END_DATE</td>
<td>month/day/year</td>
</tr>
<tr>
<td>END_TIME</td>
<td>hours:minutes</td>
</tr>
<tr>
<td>REPORT_START_DATE</td>
<td>month/day/year</td>
</tr>
<tr>
<td>REPORT_START_TIME</td>
<td>hours:minutes</td>
</tr>
<tr>
<td>SWEEP_START</td>
<td>month/day</td>
</tr>
<tr>
<td>SWEEP_END</td>
<td>month/day</td>
</tr>
<tr>
<td>DRY_DAYS</td>
<td>days</td>
</tr>
<tr>
<td>WET_STEP</td>
<td>hours:minutes:seconds</td>
</tr>
<tr>
<td>DRY_STEP</td>
<td>hours:minutes:seconds</td>
</tr>
<tr>
<td>ROUTING_STEP</td>
<td>hours:minutes:seconds</td>
</tr>
<tr>
<td>REPORT_STEP</td>
<td>hours:minutes:seconds</td>
</tr>
<tr>
<td>INERTIAL_DAMPING</td>
<td>NONE / PARTIAL / FULL</td>
</tr>
<tr>
<td>VARIABLE_STEP</td>
<td>value</td>
</tr>
<tr>
<td>LENGTHENING_STEP</td>
<td>seconds</td>
</tr>
<tr>
<td>MIN_SURFAREA</td>
<td>value</td>
</tr>
<tr>
<td>COMPATIBILITY</td>
<td>5 / 4 / 3</td>
</tr>
<tr>
<td>TEMPLDIR</td>
<td>directory</td>
</tr>
</tbody>
</table>

Remarks: FLOW_UNITS makes a choice of flow units. Selecting a US flow unit means that all other quantities will be expressed in US units, while choosing a metric flow unit will force all quantities to be expressed in metric units. The default is CFS.

INFILTRATION selects a model for computing infiltration of rainfall into the upper soil zone of subcatchments. The default model is HORTON.

FLOW_ROUTING determines which method is used to route flows through the drainage system. STEADY refers to sequential steady state routing (i.e. hydrograph translation), KINWAVE to kinematic wave routing, and DYNWAVE to dynamic wave routing. The default routing method is KINWAVE.
ALLOW_PONDING determines whether excess water is allowed to collect atop nodes and be re-introduced into the system as conditions permit. The default is NO ponding. In order for ponding to actually occur at a particular node, a non-zero value for its Ponded Area attribute must be used.

START_DATE is the date when the simulation begins. If not supplied, a date of 1/1/2002 is used.

START_TIME is the time of day on the starting date when the simulation begins. The default is 12 midnight (0:00:00).

END_DATE is the date when the simulation is to end. The default is the start date.

END_TIME is the time of day on the ending date when the simulation will end. The default is 24:00:00.

REPORT_START_DATE is the date when reporting of results is to begin. The default is the simulation start date.

REPORT_START_TIME is the time of day on the report starting date when reporting is to begin. The default is the simulation start time of day.

SWEEP_START is the day of the year (month/day) when street sweeping operations begin. The default is 1/1.

SWEEP_END is the day of the year (month/day) when street sweeping operations end. The default is 12/31.

DRY_DAYS is the number of days with no rainfall prior to the start of the simulation. The default is 0.

WET_STEP is the time step length used to compute runoff from subcatchments during periods of rainfall or when ponded water still remains on the surface. The default is 0:05:00.

DRY_STEP is the time step length used for runoff computations (consisting essentially of pollutant buildup) during periods when there is no rainfall and no ponded water. The default is 1:00:00.

ROUTING_STEP is the time step length used for routing flows and water quality constituents through the conveyance system. The default is 0:05:00, which should be reduced if using dynamic wave routing.

REPORT_STEP is the time interval for reporting of computed results. The default is 0:15:00.

INERTIAL_DAMPING indicates how the inertial terms in the Saint Venant momentum equation will be handled under dynamic wave flow routing. Choosing NONE maintains these terms at their full value under all conditions. Selecting
PARTIAL will reduce the terms as flow comes closer to being critical (and ignores them when flow is supercritical). Choosing FULL will drop the terms altogether.

VARIABLE_STEP is a safety factor applied to a variable time step computed for each time period under dynamic wave flow routing. The variable time step is computed so as to satisfy the Courant stability criterion for each conduit, to prevent an excessive change in water depth at each node, and yet not exceed the ROUTING_STEP value. If the safety factor is 0 (the default), then no variable time step is used.

LENGTHENING_STEP is a time step, in seconds, used to lengthen conduits under dynamic wave routing, so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer conduits will require lengthening. A value of 0 (the default) means that no conduits will be lengthened.

MIN_SURFAREA is a minimum surface area used at nodes when computing changes in water depth under dynamic wave routing. If 0 is entered, then the default value of 12.566 ft² (i.e., the area of a 4-ft diameter manhole) is used.

COMPATIBILITY indicates which version of SWMM's dynamic wave solution method should be used.

5 - uses Picard iterations (a method of successive approximations) to integrate the nodal continuity equation at each time step. The Preissmann slot method is used to handle surcharging.

4 - uses the SWMM 4 method of modified Euler integration and a special iterative procedure to handle surcharging.

3 – also uses the SWMM 4 method but with the weights used in SWMM 3 to compute an average conduit flow area and hydraulic radius based on values computed at either end.

The default solution method is 5.

TEMPDIR provides the name of a file directory (or folder) where SWMM writes its temporary files. If the directory name contains spaces then it should be placed within double quotes. If no directory is specified, then the temporary files are written to the root directory of the current drive.

---

**Section:** [REPORT]

**Purpose:** Describes the contents of the report file that is produced.

**Formats:**

- **INPUT** YES / NO
- **CONTINUITY** YES / NO
- **FLOWSTATS** YES / NO
- **CONTROLS** YES / NO
- **SUBCATCHMENTS** ALL / NONE / <list of subcatchment names>
- **NODES** ALL / NONE / <list of node names>
- **LINKS** ALL / NONE / <list of link names>
Remarks: **INPUT** specifies whether or not a summary of the input data should be provided in the output report. The default is **NO**.

**CONTINUITY** specifies whether continuity checks should be reported or not. The default is **YES**.

**FLOWSTATS** specifies whether summary flow statistics should be reported or not. The default is **YES**.

**CONTROLS** specifies whether all control actions taken during a simulation should be listed or not. The default is **NO**.

**SUBCATCHMENTS** gives a list of subcatchments whose results are to be reported. The default is **NONE**.

**NODES** gives a list of nodes whose results are to be reported. The default is **NONE**.

**LINKS** gives a list of links whose results are to be reported. The default is **NONE**.

The **SUBCATCHMENTS**, **NODES**, and **LINKS** lines can be repeated multiple times.

---

**Section:** [FILES]

**Purpose:** Identifies optional interface files used or saved by a run.

**Formats:**
- **USE / SAVE RAINFALL**  
  Fname
- **USE / SAVE RUNOFF**  
  Fname
- **USE / SAVE HOTSTART**  
  Fname
- **USE / SAVE RDII**  
  Fname
- **USE INFLOWS**  
  Fname
- **SAVE OUTFLOWS**  
  Fname

**Remarks:**  
Fname name of interface file.

Refer to Section 11.7 for a description of interface files. Rainfall, Runoff, and RDII files can either be used or saved in a run, but not both. A run can both use and save a Hotstart file (with different names).

---

**Section:** [RAINGAGES]

**Purpose:** Identifies each rain gage that provides rainfall data for the study area.

**Formats:**
- **Name Form Intvl SCF TIMESERIES Tseries**
- **Name Form Intvl SCF FILE Fname Sta Units**
Remarks: Name  name assigned to rain gage.
          Form  form of recorded rainfall, either INTENSITY, VOLUME or CUMULATIVE.
          Intvl time interval between gage readings in decimal hours or hours:minutes format (e.g., 0:15 for 15-minute readings).
          SCF  snow catch deficiency correction factor (use 1.0 for no adjustment).
          Tseries  name of time series in [TIMESERIES] section with rainfall data.
          Fname  name of external file with rainfall data. Rainfall files are discussed in Section 11.3.
          Sta  name of recording station used in the rain file.
          Units  rain depth units used in the rain file, either IN (inches) or MM (millimeters).

Section: [EVAPORATION]

Purpose: Specifies how daily evaporation rates vary with time for the study area.

Formats: CONSTANT  evap
          MONTHLY  evap1 evap2 ... evap12
          TIMESERIES  Tseries
          FILE  (pan1 pan2 ... pan12)

Remarks: evap  constant evaporation rate (in/day or mm/day).
          evap1  evaporation rate in January (in/day or mm/day).
          ...
          evap12  evaporation rate in December (in/day or mm/day).
          Tseries  name of time series in [TIMESERIES] section with evaporation data.
          pan1  pan coefficient for January.
          ...
          pan12  pan coefficient for December.

Use only one of the above formats. If no [EVAPORATION] section appears, then evaporation is assumed to be 0.

FILE indicates that evaporation data will be read from the same external climate file used for air temperatures (see below).

Section: [TEMPERATURE]

Purpose: Specifies daily air temperatures, monthly wind speed, and various snow melt parameters for the study area. Required only when snow melt is being modeled or when evaporation rates are read from an external climate file.

Formats: TIMESERIES  Tseries
          FILE  Fname (Start)
WINDSPEED MONTHLY s1 s2 ... s11 s12
WINDSPEED FILE

SNOWMELT Stemp ATIwt RNM Elev Lat DTLong
ADC IMPERVIOUS f.0 f.1 ... f.8 f.9
ADC PERVIOUS f.0 f.1 ... f.8 f.9

Remarks:

Tseries  name of time series in [TIMESERIES] section with temperature data.
Fname   name of external Climate file with temperature data.
Start   date to begin reading from the file in month/day/year format (default is the beginning of the file).
s1      average wind speed in January (mph or km/hr).
...
s12     average wind speed in December (mph or km/hr).
Stemp   air temperature at which precipitation falls as snow (deg F or C).
ATIwt   antecedent temperature index weight (default is 0.5).
RNM     negative melt ratio (default is 0.6).
Elev    average elevation of study area above mean sea level (ft or m) (default is 0).
Lat     latitude of the study area in degrees North (default is 50).
DTLong  correction, in minutes of time, between true solar time and the standard clock time (default is 0).
f.0     fraction of area covered by snow when ratio of snow depth to depth at 100% cover is 0.
...
f.9     fraction of area covered by snow when ratio of snow depth to depth at 100% cover is 0.9.

Use the TIMESERIES line to read air temperature from a time series or the FILE line to read it from an external Climate file. Climate files are discussed in Section 11.4. If neither format is used, then air temperature remains constant at 70 degrees F.

Wind speed can be specified either by monthly average values or by the same Climate file used for air temperature. If neither option appears, then wind speed is assumed to be 0.

Separate Areal Depletion Curves (ADC) can be defined for impervious and pervious sub-areas. The ADC parameters will default to 1.0 (meaning no depletion) if no data are supplied for a particular type of sub-area.

Section: [SUBCATCHMENTS]

Purpose: Identifies each subcatchment within the study area. Subcatchments are land area units which generate runoff from rainfall.

Format: Name Rgage OutID Area %Imperv Width Slope Clength (Spack)
Remarks:

Name name assigned to subcatchment.

Rgage name of rain gage in [RAINGAGES] section assigned to subcatchment.

OutID name of node or subcatchment that receives runoff from subcatchment.

Area area of subcatchment (acres or hectares).

%Imperv percent imperviousness of subcatchment.

Width characteristic width of subcatchment (ft or meters).

Slope subcatchment slope (percent).

Clength total curb length (any length units).

Spack name of snow pack object (from [SNOWPACKS] section) that characterizes snow accumulation and melting over the subcatchment.

---

Section: [SUBAREAS]

Purpose: Supplies information about pervious and impervious areas for each subcatchment. Each subcatchment can consist of a pervious sub-area, an impervious sub-area with depression storage, and an impervious sub-area without depression storage.

Format: Subcat Nimp Nperv Simp Sperv %Zero (RouteTo %Rted)

Remarks:

Subcat subcatchment name.

Nimp Manning's n for overland flow over the impervious sub-area.

Nperv Manning's n for overland flow over the pervious sub-area.

Simp depression storage for impervious sub-area (inches or mm).

Sperv depression storage for pervious sub-area (inches or mm).

%Zero percent of impervious area with no depression storage.

RouteTo Use IMPERVIOUS if pervious area runoff runs onto impervious area, PERVIOUS if impervious runoff runs onto pervious area, or OUTLET if both areas drain to the subcatchment's outlet (default = OUTLET).

%Rted Percent of runoff routed from one type of area to another (default = 100).

---

Section: [INFILTRATION]

Purpose: Supplies infiltration parameters for each subcatchment. Rainfall lost to infiltration only occurs over the pervious sub-area of a subcatchment.

Formats: Subcat MaxRate MinRate Decay DryTime MaxInf

Subcat Suction Conduct InitDef

Subcat CurveNo Conduct DryTime

Remarks: Subcat subcatchment name.

For Horton Infiltration:

MaxRate Maximum infiltration rate on Horton curve (in/hr or mm/hr).

MinRate Minimum infiltration rate on Horton curve (in/hr or mm/hr).

Decay Decay rate constant of Horton curve (1/hr).
DryTime  Time it takes for fully saturated soil to dry (days).
MaxInf   Maximum infiltration volume possible (0 if not applicable) (in or mm).

**For Green-Ampt Infiltration:**
Suction  Soil capillary suction (in or mm).
Conduct  Soil saturated hydraulic conductivity (in/hr or mm/hr).
InitDef  Initial soil moisture deficit (volume of voids / total volume).

**For Curve-Number Infiltration:**
CurveNo  SCS Curve Number.
Conduct  Soil saturated hydraulic conductivity (in/hr or mm/hr).
DryTime  Time it takes for fully saturated soil to dry (days).

---

**Section: [AQUIFERS]**

**Purpose:** Supplies parameters for each unconfined groundwater aquifer in the study area. Aquifers consist of two zones – a lower saturated zone and an upper unsaturated zone.

**Formats:** Name Por WP FC K Ks Ps UEF LED GWR BE WTE UMC

**Remarks:**
- **Name**  name assigned to aquifer.
- **Por**   soil porosity (volumetric fraction).
- **WP**    soil wilting point (volumetric fraction).
- **FC**    soil field capacity (volumetric fraction).
- **K**     saturated hydraulic conductivity (in/hr or mm/hr).
- **Ks**    slope of hydraulic conductivity versus moisture content curve (in/hr or mm/hr).
- **Ps**    slope of soil tension versus moisture content curve (inches or mm).
- **UEF**   fraction of total evaporation available for evapotranspiration in the upper unsaturated zone.
- **LED**   maximum depth into the lower saturated zone over which evapotranspiration can occur (ft or m).
- **GWR**   rate of percolation from saturated zone to deep groundwater when water table is at ground surface (in/hr or mm/hr).
- **BE**    elevation of the bottom of the aquifer (ft or m).
- **WTE**   water table elevation at start of simulation (ft or m).
- **UMC**  unsaturated zone moisture content at start of simulation (volumetric fraction).

---

**Section: [GROUNDWATER]**

**Purpose:** Supplies parameters that determine the rate of groundwater flow between the aquifer underneath a subcatchment and a node of the conveyance system.

**Formats:** Subcat Aquifer Node SurfEl A1 B1 A2 B2 A3 TW
Remarks:  
Subcat  subcatchment name.  
Aquifer name of groundwater aquifer underneath the subcatchment.  
Node name of node in conveyance system exchanging groundwater with aquifer.  
SurfEl surface elevation of subcatchment (ft or m).  
A1 groundwater flow coefficient (see below).  
B1 groundwater flow exponent (see below).  
A2 surface water flow coefficient (see below).  
B2 surface water flow exponent (see below).  
A3 surface water – groundwater interaction coefficient (see below).  
TW fixed depth of surface water at receiving node (ft or m) (set to zero if surface water depth will vary as computed by flow routing).

The flow coefficients are used in the following equation that determines a groundwater flow rate based on groundwater and surface water elevations:

\[ Q_{gw} = A_1(H_{gw} - E)^{B_1} - A_2(H_{sw} - E)^{B_2} + A_3H_{gw}H_{sw} \]

where:

- \( Q_{gw} \) = groundwater flow (cfs per acre or cms per hectare),
- \( H_{gw} \) = computed elevation of groundwater table (ft or m),
- \( H_{sw} \) = computed elevation of surface water at receiving node (ft or m) if \( TW \) is 0 or \( TW + E \) otherwise,
- \( E \) = fixed elevation of node invert as specified in input (ft or m).

Section:  [SNOWPACKS]

Purpose:  Specifies parameters that govern how snowfall accumulates and melts on the plowable, impervious and pervious surfaces of subcatchments.

Formats:  
Name  PLOWABLE  Cmin  Cmax  Tbase  FWF  SD0  FW0  SNN0  
Name  IMPERVIOUS  Cmin  Cmax  Tbase  FWF  SD0  FW0  SD100  
Name  PERVIOUS  Cmin  Cmax  Tbase  FWF  SD0  FW0  SD100  
Name  REMOVAL  Dplow  Fout  Fimp  Fperv  Fimelt  (Fsub Scatch)  

Remarks:  
Name  name assigned to snowpack parameter set .  
Cmin  minimum assigned to snowpack parameter set .

- \( C_{min} \) = minimum melt coefficient (in/hr-deg F or mm/hr-deg C).
- \( C_{max} \) = maximum melt coefficient (in/hr-deg F or mm/hr-deg C).
- \( T_{base} \) = snow melt base temperature (deg F or deg C).
- \( FWF \) = ratio of free water holding capacity to snow depth (fraction).
- \( SD0 \) = initial snow depth (in or mm water equivalent).
- \( FW0 \) = initial free water in pack (in or mm).
- \( SNN0 \) = fraction of impervious area that can be plowed.
- \( SD100 \) = snow depth above which there is 100% cover (in or mm water equivalent).
**SDplow**  depth of snow on plowable areas at which redistribution through plowing occurs (in or mm).

**Fout**  fraction of excess snow on plowable area transferred out of watershed.

**Fimp**  fraction of excess snow on plowable area transferred to impervious area by plowing.

**Fperv**  fraction of excess snow on plowable area transferred to pervious area by plowing.

**Fimelt**  fraction of excess snow on plowable area converted into immediate melt.

**Fsub**  fraction of excess snow on plowable area transferred to pervious area in another subcatchment.

**Scatch**  name of subcatchment receiving the Fsubcatch fraction of transferred snow.

Use one set of **PLOWABLE, IMPERVIOUS, and PERVIOUS** lines for each snow pack parameter set created. Snow pack parameter sets are associated with specific subcatchments in the [SUBCATCHMENTS] section. Multiple subcatchments can share the same set of snow pack parameters.

The **PLOWABLE** line contains parameters for the impervious area of a subcatchment that is subject to snow removal by plowing but not to areal depletion. This area is the fraction **SNN0** of the total impervious area. The **IMPERVIOUS** line contains parameter values for the remaining impervious area and the **PERVIOUS** line does the same for the entire pervious area. Both of the latter two areas are subject to areal depletion.

The **REMOVAL** line describes how snow removed from the plowable area is transferred onto other areas. The various transfer fractions should either sum to 1.0 or be all 0.0 to indicate that no plowing is done (or the line can simply be omitted).

---

**Section:** [JUNCTIONS]

**Purpose:** Identifies each junction node of the drainage system. Junctions are points in space where channels and pipes connect together. For sewer systems they can be either connection fittings or manholes.

**Format:** Name Elev (Ymax Y0 Ysur Apond)

**Remarks:**

- **Name**  name assigned to junction node.
- **Elev**  elevation of junction invert (ft or m).
- **Ymax**  depth from ground to invert elevation (ft or m) (default is 0).
- **Y0**  water depth at start of simulation (ft or m) (default is 0).
- **Ysur**  maximum additional head above ground elevation that manhole junction can sustain under surcharge conditions (ft or m) (default is 0).
- **Apond**  area subjected to surface ponding once water depth exceeds \( Y_{max} + Y_{sur} \) (ft2 or m2) (default is 0).
Section: \[OUTFALLS]\]

**Purpose:** Identifies each outfall node (i.e., final downstream boundary) of the drainage system and the corresponding water stage elevation. Only one link can be incident on an outfall node.

**Formats:**
- Name Elev \textbf{FREE} Gate
- Name Elev \textbf{NORMAL} Gate
- Name Elev \textbf{FIXED} Stage Gate
- Name Elev \textbf{TIDAL} Tcurve Gate
- Name Elev \textbf{TIMESERIES} Tseries Gate

**Remarks:**
- Name \text{name assigned to outfall node.}
- Elev \text{invert elevation (ft or m).}
- Stage \text{elevation of fixed stage outfall (ft or m).}
- Tcurve \text{name of curve in [CURVES] section containing tidal height (i.e., outfall stage) v. hour of day over a complete tidal cycle.}
- Tseries \text{name of time series in [TIMESERIES] section that describes how outfall stage varies with time.}
- Gate \text{YES or NO depending on whether a flap gate is present that prevents reverse flow.}

Section: \[DIVIDERS]\]

**Purpose:** Identifies each flow divider node of the drainage system. Flow dividers are junctions with exactly two outflow conduits where the total outflow is divided between the two in a prescribed manner.

**Formats:**
- Name Elev DivLink \textbf{OVERFLOW} (Ymax Y0 Ysur Apond)
- Name Elev DivLink \textbf{CUTOFF} Qmin (Ymax Y0 Ysur Apond)
- Name Elev DivLink \textbf{TABULAR} Dcurve (Ymax Y0 Ysur Apond)
- Name Elev DivLink \textbf{WEIR} Qmin Ht Cd (Ymax Y0 Ysur Apond)

**Remarks:**
- Name \text{name assigned to divider node.}
- Elev \text{invert elevation (ft or m).}
- DivLink \text{name of link to which flow is diverted.}
- Qmin \text{flow at which diversion begins for either a CUTOFF or WEIR divider (flow units).}
- Dcurve \text{name of curve for TABULAR divider that relates diverted flow to total flow.}
- Ht \text{height of WEIR divider (ft or m).}
- Cd \text{discharge coefficient for WEIR divider.}
- Ymax \text{depth from ground to invert elevation (ft or m) (default is 0).}
- Y0 \text{water depth at start of simulation (ft or m) (default is 0).}
- Ysur \text{maximum additional head above ground elevation that node can sustain under surcharge conditions (ft or m) (default is 0).}
- Apond \text{area subjected to surface ponding once water depth exceeds } Ymax + Ysur \text{ (ft2 or m2) (default is 0).}
Section: [STORAGE]

Purpose: Identifies each storage node of the drainage system. Storage nodes can have any shape as specified by a surface area versus water depth relation.

Format: Name Elev Ymax Y0 TABULAR Acurve (Apond Fevap)
Name Elev Ymax Y0 FUNCTIONAL A1 A2 A0 (Apond Fevap)

Remarks:

Name name assigned to storage node.
Elev invert elevation (ft or m).
Ymax maximum water depth possible (ft or m).
Y0 water depth at start of simulation (ft or m).
Acurve name of curve in [CURVES] section with surface area (ft² or m²) as a function of depth (ft or m) for TABULAR geometry.
A1 coefficient of FUNCTIONAL relation between surface area and depth.
A2 exponent of FUNCTIONAL relation between surface area and depth.
A0 constant of FUNCTIONAL relation between surface area and depth.
Apond surface area subjected to ponding once water depth exceeds Ymax (ft² or m²) (default is 0).
Fevap fraction of potential evaporation from surface realized (default is 0).

A1, A2, and A0 are used in the following expression that relates surface area (ft² or m²) to water depth (ft or m) for a storage unit with FUNCTIONAL geometry:

\[ \text{Area} = A0 + A1 \times \text{Depth}^{A2} \]

Section: [CONDUITS]

Purpose: Identifies each conduit link of the drainage system. Conduits are pipes or channels that convey water from one node to another.

Format: Name Node1 Node2 Length N Z1 Z2 (Q0)

Remarks:

Name name assigned to conduit link.
Node1 name of upstream node.
Node2 name of downstream node.
Length conduit length (ft or m).
N value of n (i.e., roughness parameter) in Manning’s equation.
Z1 offset height of upstream end of conduit invert above the invert elevation of its upstream node (ft or m).
Z2 offset height of downstream end of conduit invert above the invert elevation of its downstream node (ft or m).
Q0 flow in conduit at start of simulation (flow units) (default is 0).
The figure below illustrates the meaning of the \( Z1 \) and \( Z2 \) parameters.

---

**Section:** [PUMPS]

**Purpose:** Identifies each pump link of the drainage system.

**Format:** 

\[ \text{Name} \quad \text{Node1} \quad \text{Node2} \quad \text{Pcurve} \quad (\text{Status}) \]

**Remarks:**

- **Name** name assigned to pump link.
- **Node1** name of node on inlet side of pump.
- **Node2** name of node on outlet side of pump.
- **Pcurve** name of pump curve listed in the [CURVES] section of the input.
- **Status** status at start of simulation (either ON or OFF; default is ON).

See Section 3.2 for a description of the different types of pumps available.

---

**Section:** [ORIFICES]

**Purpose:** Identifies each orifice link of the drainage system. An orifice link serves to limit the flow exiting a node and is often used to model flow diversions.

**Format:** 

\[ \text{Name} \quad \text{Node1} \quad \text{Node2} \quad \text{Type} \quad \text{Height} \quad \text{Cd} \quad (\text{Flap}) \]

**Remarks:**

- **Name** name assigned to orifice link.
- **Node1** name of node on inlet end of orifice.
- **Node2** name of node on outlet end of orifice.
- **Type** orientation of orifice: either SIDE or BOTTOM.
- **Height** height of a Side orifice’s bottom from invert of inlet node (ft or m) (overridden to 0 for a Bottom orifice).
- **Cd** discharge coefficient (unitless).
- **Flap** YES if flap gate present to prevent reverse flow, NO if not (default is NO).

The geometry of an orifice’s opening must be described in the [XSECTIONS] section. The only allowable shapes are CIRCULAR and RECT_CLOSED (closed rectangular).
Section:  [WEIRS]

Purpose:  Identifies each weir link of the drainage system. Weirs are used to model flow diversions.

Format:  Name  Node1  Node2  Type  Height  Cd  (Flap  EC  Cd2)

Remarks:  Name   name assigned to weir link.
           Node1  name of node on inlet side of weir.
           Node2  name of node on outlet side of weir.
           Type   TRANSVERSE, SIDEFLOW, V–NOTCH, or TRAPEZOIDAL.
           Height height of weir crest above invert of inlet node (ft or m).
           Cd     weir discharge coefficient (for CFS if using US flow units or CMS if using metric flow units).
           Flap   YES if flap gate present to prevent reverse flow, NO if not (default is NO).
           EC     number of end contractions for TRANSVERSE or TRAPEZOIDAL weir (default is 0).
           Cd2    discharge coefficient for triangular ends of a TRAPEZOIDAL weir (for CFS if using US flow units or CMS if using metric flow units) (default is value of Cd).

The geometry of a weir’s opening is described in the [XSECTIONS] section. The following shapes must be used with each type of weir:

<table>
<thead>
<tr>
<th>Weir Type</th>
<th>Cross-Section Shape</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transverse</td>
<td>RECT_OPEN</td>
</tr>
<tr>
<td>Sideflow</td>
<td>RECT_OPEN</td>
</tr>
<tr>
<td>V-Notch</td>
<td>TRIANGULAR</td>
</tr>
<tr>
<td>Trapezoidal</td>
<td>TRAPEZOIDAL</td>
</tr>
</tbody>
</table>
Section: [OUTLETS]

Purpose: Identifies each outlet flow control device of the drainage system. These devices are used to model outflows from storage units or flow diversions that have a user-defined relation between flow rate and water depth.

Format: Name  Node1  Node2  Height  TABULAR  Qcurve (Flap)  
Name  Node1  Node2  Height  FUNCTIONAL  C1  C2  (Flap)

Remarks:
Name name assigned to outlet link.
Node1 name of node on inlet end of link.
Node2 name of node on outflow end of link.
Height minimum water depth at inlet node for outflow to occur (ft or m).
Qcurve name of rating curve listed in [CURVES] section that describes outflow rate (flow units) as a function of head (ft or m) across the outlet for a TABULAR outlet.
C1, C2 coefficient and exponent, respectively, of power function that relates outflow (Q) to head across the link (H) for a FUNCTIONAL outlet (i.e., \( Q = C1(H)^{C2} \)).
Flap YES if flap gate present to prevent reverse flow, NO if not (default is NO).

Section: [XSECTIONS]

Purpose: Provides cross-section geometric data for conduit and regulator links of the drainage system.

Formats: Link  Shape  Geom1  Geom2  Geom3  Geom4  ( Barrels )
Link  IRREGULAR  Tsect

Remarks:
Link name of conduit, orifice, or weir.
Shape cross-section shape (see Table D-1 below for available shapes).
Geom1 maximum depth (ft or m).
Geom2 width parameter (ft or m).
Geom3, Geom4 auxiliary parameters (e.g., side slopes). (See Table D-1 for details).
Barrels number of barrels (i.e., number of parallel pipes of equal size, slope, and roughness) associated with a conduit (default is 1).
Tsect name of entry in [TRANSECTS] section which describes the cross-section geometry of an irregular channel.
Table D-1. Geometric parameters of conduit cross sections.

<table>
<thead>
<tr>
<th>Shape</th>
<th>Geom1</th>
<th>Geom2</th>
<th>Geom3</th>
<th>Geom4</th>
</tr>
</thead>
<tbody>
<tr>
<td>CIRCULAR</td>
<td>Diameter</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FILLED_CIRCULAR(^1)</td>
<td>Diameter</td>
<td>Sediment</td>
<td>Depth</td>
<td></td>
</tr>
<tr>
<td>RECT_CLOSED</td>
<td>Height</td>
<td>Top Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RECT_OPEN</td>
<td>Height</td>
<td>Top Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRAPEZOIDAL</td>
<td>Height</td>
<td>Top Width</td>
<td>Left Slope</td>
<td>Right Slope</td>
</tr>
<tr>
<td>TRIANGULAR</td>
<td>Height</td>
<td>Top Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>HORIZ_ELLIPSE</td>
<td>Height</td>
<td>Max. Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VERT_ELLIPSE</td>
<td>Height</td>
<td>Max. Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ARCH (standard)</td>
<td>Size Code(^3)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ARCH (non-standard)</td>
<td>Height</td>
<td>Max. Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PARABOLIC</td>
<td>Height</td>
<td>Top Width</td>
<td></td>
<td></td>
</tr>
<tr>
<td>POWER_FUNCTION</td>
<td>Height</td>
<td>Top Width</td>
<td>Exponent</td>
<td></td>
</tr>
<tr>
<td>RECT_TRIANGULAR</td>
<td>Height</td>
<td>Top Width</td>
<td>Triangle</td>
<td>Height</td>
</tr>
<tr>
<td>RECT_CIRCULAR</td>
<td>Height</td>
<td>Top Width</td>
<td>Bottom</td>
<td>Radius</td>
</tr>
<tr>
<td>MODBASKETHANDLE</td>
<td>Height</td>
<td>Bottom</td>
<td>Width</td>
<td></td>
</tr>
<tr>
<td>EGG</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HORSESHOE</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>GOTHIC</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CATENARY</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SEMIELLIPTICAL</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BASKETHANDLE</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SEMICIRCULAR</td>
<td>Height</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

\(^1\) A circular conduit partially filled with sediment to a specified depth.
\(^2\) Set to zero to use a standard shaped elliptical pipe as cataloged in the publications mentioned in the footnote below.
\(^3\) As listed in either the "Concrete Pipe Design Manual" published by the American Concrete Pipe Association or "Modern Sewer Design" published by the American Iron and Steel Institute.

Section: [LOSSES]

Purpose: Specifies minor head loss coefficients and flap gates for conduits.

Formats: Conduit  Kentry  Kexit  Kavg (Flap)

Remarks: Conduit name of conduit.
Kentry entrance minor head loss coefficient.
Kexit exit minor head loss coefficient.
Kavg average minor head loss coefficient across length of conduit.
Flap    YES if conduit has a flap valve that prevents back flow, NO otherwise. (Default is NO).

Minor losses are only computed for the Dynamic Wave flow routing option (see [OPTIONS] section). They are computed as $Kv^2/2g$ where $K =$ minor loss coefficient, $v =$ velocity, and $g =$ acceleration of gravity. Entrance losses are based on the velocity at the entrance of the conduit, exit losses on the exit velocity, and average losses on the average velocity.

Only enter data for conduits that actually have minor losses or flap valves.

Section: [TRANSECTS]

Purpose: Describes the cross-section geometry of natural channels or conduits with irregular shapes following the HEC-2 data format.

Formats: 

NC   Nleft  Nright  Nchanl
X1   Name  Nsta  Xleft  Xright  0  0  0  Wfactor  Eoffset
GR   Elev  Station  ...  Elev  Station

Remarks: 

Nleft   Manning’s n of right overbank portion of channel (use 0 if no change from previous NC line).
Nright  Manning’s n of right overbank portion of channel (use 0 if no change from previous NC line).
Nchanl  Manning’s n of main channel portion of channel (use 0 if no change from previous NC line).
Name    name assigned to transect.
Nsta    number of stations across cross-section at which elevation data is supplied.
Xleft   station position which ends the left overbank portion of the channel (ft or m).
Xright  station position which begins the right overbank portion of the channel (ft or m).
Wfactor factor by which distances between stations should be multiplied to increase (or decrease) the width of the channel (enter 0 if not applicable).
Eoffset amount added (or subtracted) from the elevation of each station (ft or m).
Elev    elevation of the channel bottom at a cross-section station relative to some fixed reference (ft or m).
Station distance of a cross-section station from some fixed reference (ft or m).

Transect geometry is described as shown below, assuming that one is looking in a downstream direction:
The first line in this section must always be a **NC** line. After that, the **NC** line is only needed when a transect has different N values than the previous one.

The Manning’s n values on the **NC** line will supercede any roughness value entered for the conduit which uses the irregular cross-section.

There should be one **X1** line for each transect. Any number of **GR** lines may follow, and each **GR** line can have any number of Elevation-Station data pairs. (In HEC-2 the **GR** line is limited to 5 stations.)

The station that defines the left overbank boundary on the **X1** line must correspond to one of the station entries on the **GR** lines that follow. The same holds true for the right overbank boundary. If there is no match, a warning will be issued and the program will assume that no overbank area exists.

---

**Section:** [CONTROLS]

**Purpose:** Determines how pumps and regulators will be adjusted based on simulation time or conditions at specific nodes and links.

**Formats:** Each control rule is a series of statements of the form:

```
RULE ruleID
IF condition_1
AND condition_2
OR condition_3
AND condition_4
Etc.
THEN action_1
AND action_2
Etc.
ELSE action_3
AND action_4
Etc.
PRIOR  value
```
Remarks: 

- **RuleID**: an ID label assigned to the rule.
- **condition_n**: a condition clause.
- **action_n**: an action clause.
- **value**: a priority value (e.g., a number from 1 to 5).

A condition clause of a Control Rule has the following format:

```
Object  Name  Attribute  Relation  Value
```

where **Object** is a category of object, **Name** is the object’s assigned ID name, **Attribute** is the name of an attribute or property of the object, **Relation** is a relational operator (=, <>, <, <=, >, >=), and **Value** is an attribute value.

Some examples of condition clauses are:
- NODE  N23  DEPTH  >  10
- PUMP  P45  STATUS = OFF
- SIMULATION TIME = 12:45:00

The objects and attributes that can appear in a condition clause are as follows:

<table>
<thead>
<tr>
<th>Object</th>
<th>Attributes</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>NODE</td>
<td>DEPTH, HEAD, INFLOW</td>
<td>numerical value numerical value numerical value</td>
</tr>
<tr>
<td>LINK</td>
<td>FLOW, DEPTH</td>
<td>numerical value numerical value</td>
</tr>
<tr>
<td>PUMP</td>
<td>STATUS, FLOW</td>
<td>ON or OFF numerical value</td>
</tr>
<tr>
<td>ORIFICE</td>
<td>SETTING</td>
<td>fraction open</td>
</tr>
<tr>
<td>WEIR</td>
<td>SETTING</td>
<td>fraction open</td>
</tr>
<tr>
<td>SIMULATION</td>
<td>TIME</td>
<td>elapsed time in decimal hours or hr:min:sec</td>
</tr>
<tr>
<td>SIMULATION</td>
<td>DATE, CLOCKTIME</td>
<td>month-day-year time of day in hr:min:sec</td>
</tr>
</tbody>
</table>

An action clause of a Control Rule can have one of the following formats:

- **PUMP** **Name** **STATUS** = **ON/OFF**
- **ORIFICE** **Name** **SETTING** = **Value**
- **WEIR** **Name** **SETTING** = **Value**

where **Name** is the name of the link being controlled and **SETTING** refers to the fractional amount that an orifice is fully open or to the fractional amount of the original height between the crest and the top of a weir that remains (i.e., weir control is accomplished by moving the crest height up and down).
Some examples of action clauses are:

PUMP P67 STATUS = OFF
ORIFICE O212 SETTING = 0.5

Only the **RULE, IF and THEN** portions of a rule are required; the other portions are optional.

When mixing **AND** and OR clauses, the OR operator has higher precedence than AND, i.e.,

```
IF A or B and C
```

is equivalent to

```
IF (A or B) and C.
```

If the interpretation was meant to be

```
IF A or (B and C)
```

then this can be expressed using two rules as in

```
IF A THEN ...        
IF B and C THEN ...  
```

The **PRIORITY** value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

**Examples:**

; Simple time-based pump control
RULE R1
IF SIMULATION TIME > 8
THEN PUMP 12 STATUS = ON
ELSE PUMP 12 STATUS = OFF

; Multi-condition orifice gate control
RULE R2A
IF NODE 23 DEPTH > 12
AND LINK 165 FLOW > 100
THEN ORIFICE R55 SETTING = 0.5

RULE R2B
IF NODE 23 DEPTH > 12
AND LINK 165 FLOW > 200
THEN ORIFICE R55 SETTING = 1.0

RULE R2C
IF NODE 23 DEPTH <= 12
OR LINK 165 FLOW <= 100
THEN ORIFICE R55 SETTING = 0

; Pump station operation
RULE R3A
IF NODE N1 DEPTH > 5
THEN PUMP N1A STATUS = ON
RULE R3B
IF NODE N1 DEPTH > 7
THEN PUMP N1B STATUS = ON

RULE R3C
IF NODE N1 DEPTH <= 3
THEN PUMP N1A STATUS = OFF
AND PUMP N1B STATUS = OFF

Section: [POLLUTANTS]

Purpose: Identifies the pollutants being analyzed.

Format: Name Units Crain Cgw Cii Kdecay (Sflag CoPoll CoFract)

Remarks: Name name assigned to pollutant.
Units concentration units (MG/L for milligrams per liter, UG/L for micrograms per liter, or #/L for direct count per liter).
Crain concentration of pollutant in rainfall (concentration units).
Cgw concentration of pollutant in groundwater (concentration units).
Cii concentration of pollutant in infl ow/infiltration (concentration units).
Kdecay first-order decay coefficient (1/days).
Sflag YES if pollutant buildup occurs only when snowfall occurs, NO otherwise (default is NO).
CoPoll name of co-pollutant (default is no co-pollutant).
CoFract fraction of co-pollutant concentration (default is 0).

FLOW is a reserved word and cannot be used to name a pollutant.

If pollutant buildup is not restricted to times of snowfall and there is no co-pollutant, then the last three parameters can be omitted.

When pollutant X has a co-pollutant Y, it means that fraction CoFract of pollutant Y’s runoff concentration is added to pollutant X’s runoff concentration when wash off from a subcatchment is computed.

Section: [LANDUSES]

Purpose: Identifies the various categories of land uses within the drainage area. Each subcatchment area can be assigned a different mix of land uses. Each land use can be subjected to a different street sweeping schedule.

Format: Name (SweepInterval Availability LastSweep)

Remarks: Name land use name.
SweepInterval days between street sweeping.
Availability  
Fraction of pollutant buildup available for removal by street sweeping.

LastSweep  
Days since last sweeping at start of the simulation.

---

**Section: [COVERAGES]**

**Purpose:** Specifies the percentage of a subcatchment’s area that is covered by each category of land use.

**Format:** Subcat  Landuse  Percent  Landuse  Percent  ...

**Remarks:** Subcat  subcatchment name.
Landuse  land use name.
Percent  percent of subcatchment area.

More than one pair of land use - percentage values can be entered per line. If more than one line is needed, then the subcatchment name must still be entered first on the succeeding lines.

If a land use does not pertain to a subcatchment, then it does not have to be entered.

If no land uses are associated with a subcatchment then no contaminants will appear in the runoff from the subcatchment.

---

**Section: [BUILDUP]**

**Purpose:** Specifies the rate at which pollutants build up over different land uses between rain events.

**Format:** Landuse  Pollutant  FuncType  C1  C2  C3  PerUnit

**Remarks:** Landuse  land use name.
Pollutant  pollutant name.
FuncType  buildup function type: (POW / EXP / SAT).
C1,C2,C3  buildup function parameters (see Table D-2).
PerUnit  AREA if buildup is per unit area, CURBLENGTH if per length of curb.

Buildup is measured in pounds (kilograms) per unit of area (or curb length) for pollutants whose concentration units are either mg/L or ug/L. If the concentration units are counts/L, then the buildup is expressed as counts per unit of area (or curb length).
Table D-2. Available pollutant buildup functions (t is antecedent dry days).

<table>
<thead>
<tr>
<th>Name</th>
<th>Function</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>POW</td>
<td>Power</td>
<td>Min (C1, C2*t^{C3})</td>
</tr>
<tr>
<td>EXP</td>
<td>Exponential</td>
<td>C1*(1 – exp(-C2*t))</td>
</tr>
<tr>
<td>SAT</td>
<td>Saturation</td>
<td>C1*t / (C3 + t)</td>
</tr>
</tbody>
</table>

Section: [WASHOFF]

Purpose: Specifies the rate at which pollutants are washed off from different land uses during rain events.

Format: Landuse Pollutant FuncType C1 C2 SweepEffic BMPEffic

Remarks: Landuse land use name.
Pollutant pollutant name.
FuncType washoff function type: EXP / RC / EMC.
C1, C2 washoff function coefficients (see Table D-3).
SweepEffic street sweeping removal efficiency (percent).
BMPEffic BMP removal efficiency (percent).

Table D-3. Pollutant wash off functions.

<table>
<thead>
<tr>
<th>Name</th>
<th>Function</th>
<th>Equation</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXP</td>
<td>Exponential</td>
<td>C1 (runoff)^C2 (buildup)</td>
<td>Mass/hour</td>
</tr>
<tr>
<td>RC</td>
<td>Rating Curve</td>
<td>C1 (runoff)^C2</td>
<td>Mass/sec</td>
</tr>
<tr>
<td>EMC</td>
<td>Event Mean Concentration</td>
<td>C1</td>
<td>Mass/Liter</td>
</tr>
</tbody>
</table>

Each washoff function expresses its results in different units.

For the Exponential function the runoff variable is expressed in catchment depth per unit of time (inches per hour or millimeters per hour), while for the Rating Curve function it is in whatever flow units were specified in the [OPTIONS] section of the input file (e.g., CFS, CMS, etc.). The buildup parameter in the Exponential function is the current buildup over the subcatchment’s land use area in mass units. The units of C1 in the Exponential function are (in/hr)^C2 per hour (or (mm/hr)^C2 per hour).

For the Rating Curve function, the units of C1 depend on the flow units employed. For the EMC (event mean concentration) function, C1 is always in concentration units.
**Section:** [TREATMENT]

**Purpose:** Specifies the degree of treatment received by pollutants at specific nodes of the drainage system.

**Format:** Node  Pollut  Result = Func

**Remarks:**
- **Node**  Name of node where treatment occurs.
- **Pollut**  Name of pollutant receiving treatment.
- **Result**  Result computed by treatment function. Choices are:
  - C – function computes effluent concentration
  - R – function computes fractional removal.
- **Func**  mathematical function expressing treatment result in terms of pollutant concentrations, pollutant removals, and other standard variables (see below).

Treatment functions can be any well-formed mathematical expression involving:
- inlet pollutant concentrations (use the pollutant name to represent a concentration)
- removal of other pollutants (use R_ prepended to the pollutant name to represent removal)
- process variables which include:
  - FLOW for flow rate into node (user’s flow units)
  - DEPTH for water depth above node invert (ft or m)
  - AREA for node surface area (ft² or m²)
  - DT for routing time step (seconds)
  - HRT for hydraulic residence time (hours)

**Examples:**

- ; 1-st order decay of BOD
  Node23  BOD   C = BOD * exp(-0.05*HRT)

- ; lead removal is 20% of TSS removal
  Node23  Lead  R = 0.2 * R_TSS

---

**Section:** [DWF]

**Purpose:** Specifies dry weather flow and its quality entering the drainage system at specific nodes.

**Format:** Node  Item  Value  (Pat1  Pat2  Pat3  Pat4)

**Remarks:**
- **Node**  name of node where dry weather flow enters.
- **Item**  keyword FLOW for flow or pollutant name for quality constituent.
- **Value**  average baseline value for corresponding Item (flow or concentration units).
*Pat1*, *Pat2*, etc. names of up to four time patterns appearing in the [*PATTERNS*] section.

The actual dry weather input will equal the product of the baseline value and any adjustment factors supplied by the specified patterns. (If not supplied, an adjustment factor defaults to 1.0.)

---

**Section:** [*PATTERNS*]

**Purpose:** Specifies time pattern of dry weather flow or quality in the form of adjustment factors applied as multipliers to baseline values.

**Format:**
- Name  *MONTHLY*  Factor1  Factor2  ...  Factor12
- Name  *DAILY*  Factor1  Factor2  ...  Factor7
- Name  *HOURLY*  Factor1  Factor2  ...  Factor24
- Name  *WEEKEND*  Factor1  Factor2  ...  Factor24

**Remarks:** Name name used to identify the pattern.

Factor1, Factor2, etc. multiplier values.

The *MONTHLY* format is used to set monthly pattern factors for dry weather flow constituents.

The *DAILY* format is used to set dry weather pattern factors for each day of the week, where Sunday is day 1.

The *HOURLY* format is used to set dry weather factors for each hour of the of the day starting from midnight. If these factors are different for weekend days than for weekday days then the *WEEKEND* format can be used to specify hourly adjustment factors just for weekends.

More than one line can be used to enter a pattern’s factors by repeating the pattern’s name (but not the pattern type) at the beginning of each additional line.

The pattern factors are applied as multipliers to any baseline dry weather flows or quality concentrations supplied in the [*DWF*] section.

**Examples:**

```
; Day of week adjustment factors
D1  DAILY  0.5  1.0  1.0  1.0  1.0  1.0  0.5
D2  DAILY  0.8  0.9  1.0  1.1  1.0  0.9  0.8
```
; Hourly adjustment factors
H1 HOURLY  0.5 0.6 0.7 0.8 0.8 0.9
H1         1.1 1.2 1.3 1.5 1.1 1.0
H1         0.9 0.8 0.7 0.6 0.5 0.5
H1         0.5 0.5 0.5 0.5 0.5 0.5

Section: [INFLows]

Purpose: Specifies external hydrographs and pollutographs that enter the drainage system at specific nodes.

Formats: Node FLOW FlowSeries
Node Pollut PollSeries Format (ConvFactor)

Remarks: Node name of node where external inflow enters.
FlowSeries name of time series in [TIMESERIES] section describing how external inflows vary with time.
Pollut name of pollutant.
PollSeries name of time series describing how external pollutant loading varies with time.
Format CONCEN if pollutant inflow is described as a concentration, MASS if it is described as a mass flow rate.
ConvFactor the factor that converts the inflow’s mass flow rate units into the product of the project’s concentration units times flow units, where concentration units are those specified for the pollutant in the [POLLUTANTS] section and flow units are those specified in the [OPTIONS] section (see example below).

If an external inflow of a pollutant concentration is specified for a node, then there must also be an external inflow of FLOW provided for the same node.

Examples: NODE2 FLOW N2FLOW
NODE33 TSS N33TSS CONCEN

;Mass inflow of BOD in time series N65BOD given in lbs/hr
;(4.5 converts lbs/hr to cfs-mg/L)
NODE65 BOD N65BOD MASS 4.5

Section: [LOADINGS]

Purpose: Specifies the pollutant buildup that exists on each subcatchment at the start of a simulation.

Format: Subcat Pollut InitBuildup Pollut InitBuildup ...
Remarks: Subcat name of a subcatchment.
Pollut name of a pollutant.
InitBuildup initial buildup of pollutant (lbs/acre or kg/hectare).

More than one pair of pollutant - buildup values can be entered per line. If more than one line is needed, then the subcatchment name must still be entered first on the succeeding lines.

If an initial buildup is not specified for a pollutant, then its initial buildup is computed by applying the `DRY_DAYS` option (specified in the `OPTIONS` section) to the pollutant’s buildup function for each land use in the subcatchment.

Section: [RDII]

Purpose: Specifies the parameters that describe rainfall-derived infiltration/inflow entering the drainage system at specific nodes.

Format: Node UHgroup SewerArea

Remarks: Node name of a node.
UHgroup name of an RDII unit hydrograph group specified in the [HYDROGRAPHS] section.
SewerArea area of the sewershed which contributes RDII to the node (acres or hectares).

Section: [HYDROGRAPHS]

Purpose: Specifies the shapes of the triangular unit hydrographs that determine the amount of rainfall-derived infiltration/inflow (RDII) entering the drainage system.

Formats: Name Raingage
Name Month R1 T1 K1 R2 T2 K2 R3 T3 K3

Remarks: Name name assigned to a unit hydrograph (UH) group.
Raingage name of rain gage used by UH group.
Month month of the year (e.g., JAN, FEB, etc. or ALL for all months).
R1,R2,R3 response ratios for the short-term, intermediate-term, and long-term UH responses, respectively.
T1,T2,T3 time to peak (hours) for the short-term, intermediate-term, and long-term UH responses, respectively.
K1,K2,K3 recession limb ratios for short-term, intermediate-term, and long-term UH responses, respectively.
For each group of unit hydrographs, use one line to specify its rain gage followed by one or more lines containing UH shape parameters for months with RDII flow. Months not listed are assumed to have no RDII.

The response ratios (R) are the fraction of a unit of rainfall depth that becomes RDII. The sum of the ratios for the three UH’s do not have to equal 1.0.

The recession limb ratios (K) are the ratio of the duration of the UH recession limb to the time to peak (T) making the UH time base $T*(1+K)$ hours. The area under each UH is 1 inch (or mm).

**Examples:**

All UH sets in this group have the same shapes except those in July

<table>
<thead>
<tr>
<th>UH</th>
<th>Rain Gage</th>
<th>Month</th>
<th>Response Ratio</th>
<th>Recession Limb Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>UH101</td>
<td>ALL</td>
<td>ALL</td>
<td>0.033</td>
<td>0.5</td>
</tr>
<tr>
<td>UH101</td>
<td>JUL</td>
<td>JUL</td>
<td>0.011</td>
<td>0.0</td>
</tr>
<tr>
<td>UH101</td>
<td></td>
<td>ALL</td>
<td>0.033</td>
<td>0.0</td>
</tr>
<tr>
<td>UH101</td>
<td></td>
<td>JUL</td>
<td>0.033</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**Section:** [CURVES]

**Purpose:** Describes a relationship between two variables in tabular format.

**Format:**

```
Name  Type  X-value  Y-value  ...
```

**Remarks:**

- **Name** name assigned to table
- **Type** STORAGE / DIVERSION / TIDAL / PUMP1 / PUMP2 / PUMP3 / PUMP4 / RATING
- **X-value** an x (independent variable) value
- **Y-value** the y (dependent variable) value corresponding to x

Multiple pairs of x-y values can appear on a line. If more than one line is needed, repeat the curve's name (but not the type) on subsequent lines. The x-values must be entered in increasing order.

Choices for curve type have the following meanings (flows are expressed in the user’s choice of flow units set in the [OPTIONS] section):

- **STORAGE** (surface area in ft$^2$ (m$^2$) v. depth in ft (m) for a storage unit node)
- **DIVERSION** (diverted outflow v. total inflow for a flow divider node)
- **TIDAL** (water surface elevation in ft (m) v. hour of the day for an outfall node)
- **PUMP1** (pump outflow v. increment of inlet node volume in ft$^3$ (m$^3$))
- **PUMP2** (pump outflow v. increment of inlet node depth in ft (m))
- **PUMP3** (pump outflow v. head difference between outlet and inlet nodes in ft (m))
- **PUMP4** (pump outflow v. continuous depth in ft (m))
- **RATING** (outlet flow v. head in ft (m))

See Section 3.2 for illustrations of the different types of pump curves.

**Examples:**

Storage curve (x = depth, y = surface area)

| Name | Type  | X-value | Y-value |...
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>AC1</td>
<td>STORAGE</td>
<td>0</td>
<td>1000</td>
</tr>
</tbody>
</table>
Section: [TIMESERIES]

Purpose: Describes how a quantity varies over time.

Formats: Name ( Date ) Hour Value ...
          Name Time Value ...

Remarks: Name name assigned to time series.
         Date date in Month/Day/Year format (e.g., June 15, 2001 would be 6/15/2001).
         Hour 24-hour military time (e.g., 8:40 pm would be 20:40) relative to the last
date specified (or to midnight of the starting date of the simulation if no
previous date was specified).
         Time hours since the start of the simulation, expressed as a decimal number or
         as Hours:Minutes.
         Value value corresponding to given date and time.

Multiple date-time-value or time-value entries can appear on a line. If more than one
line is needed, the table's name must be repeated as the first entry on subsequent
lines.

Note that there are two methods for describing the occurrence time of time series
data:
- as calendar date/time of day (which requires that at least one date, at the start of
  the series, be entered)
- as elapsed hours since the start of the simulation.

For the first method, dates need only be entered at points in time when a new day
occurs.

Examples: ;Rainfall time series with dates specified
          TS1 6-15-2001 7:00 0.1 8:00 0.2 9:00 0.05 10:00 0
          TS1 6-21-2001 4:00 0.2 5:00 0 14:00 0.1 15:00 0

          ;Inflow hydrograph - time relative to start of simulation
          ;(hours can be expressed as decimal hours or hr:min)
          HY1 0 0 1.25 100 2:30 150 3.0 120 4.5 0
          HY1 32:10 0 34.0 57 35.33 85 48.67 24 50 0

D3. Map Data Section

SWMM’s graphical user interface (GUI) can display a schematic map of the drainage
area being analyzed. This map displays subcatchments as polygons, nodes as circles,
links as polylines, and rain gages as bitmap symbols. In addition it can display text labels
and a backdrop image, such as a street map. The GUI has tools for drawing, editing,
moving, and displaying these map elements. The map’s coordinate data are stored in the format described below. Normally these data are simply appended to the SWMM input file by the GUI so users do not have to concern themselves with it. However it is sometimes more convenient to import map data from some other source, such as a CAD or GIS file, rather than drawing a map from scratch using the GUI. In this case the data can be added to the SWMM project file using any text editor or spreadsheet program. SWMM does not provide any automated facility for converting coordinate data from other file formats into the SWMM map data format.

SWMM's map data are organized into the following seven sections:

- **[MAP]** X,Y coordinates of the map’s bounding rectangle
- **[POLYGONS]** X,Y coordinates for each vertex of subcatchment polygons
- **[COORDINATES]** X,Y coordinates for nodes
- **[VERTICES]** X,Y coordinates for each interior vertex of polyline links
- **[LABELS]** X,Y coordinates and text of labels
- **[SYMBOLS]** X,Y coordinates for rain gages
- **[BACKDROP]** X,Y coordinates of the bounding rectangle and file name of the backdrop image.

Figure D-2 displays a sample map and Figure D-3 the data that describes it. Note that only one link, 3, has interior vertices which give it a curved shape. Also observe that this map’s coordinate system has no units, so that the positions of its objects may not necessarily coincide to their real-world locations.

A detailed description of each map data section will now be given. Remember that map data are only used as a visualization aid for SWMM’s GUI and they play no role in any of the runoff or routing computations. Map data are not needed for running the command line version of SWMM.

![Figure D-2. Example study area map.](image-url)
### Section: [MAP]

**Purpose:** Provides dimensions and distance units for the map.

**Formats:**

<table>
<thead>
<tr>
<th>DIMENSIONS</th>
<th>X1</th>
<th>Y1</th>
<th>X2</th>
<th>Y2</th>
</tr>
</thead>
</table>

| UNITS | FEET / METERS / DEGREES / NONE |

**Remarks:**

- **X1**: lower-left X coordinate of full map extent
- **Y1**: lower-left Y coordinate of full map extent
- **X2**: upper-right X coordinate of full map extent
- **Y2**: upper-right Y coordinate of full map extent

---

**Figure D-3.** Data for map shown in Figure D-2.

---

**[COORDINATES]**

<table>
<thead>
<tr>
<th>;; Node</th>
<th>X-Coord</th>
<th>Y-Coord</th>
</tr>
</thead>
<tbody>
<tr>
<td>N1</td>
<td>4006.62</td>
<td>5463.58</td>
</tr>
<tr>
<td>N2</td>
<td>6953.64</td>
<td>4768.21</td>
</tr>
<tr>
<td>N3</td>
<td>4635.76</td>
<td>3443.71</td>
</tr>
<tr>
<td>N4</td>
<td>8509.93</td>
<td>827.81</td>
</tr>
</tbody>
</table>

**[VERTICES]**

<table>
<thead>
<tr>
<th>;; Link</th>
<th>X-Coord</th>
<th>Y-Coord</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>5430.46</td>
<td>2019.87</td>
</tr>
<tr>
<td>3</td>
<td>7251.66</td>
<td>927.15</td>
</tr>
</tbody>
</table>

**[SYMBOLS]**

<table>
<thead>
<tr>
<th>;; Gage</th>
<th>X-Coord</th>
<th>Y-Coord</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1</td>
<td>5298.01</td>
<td>9139.07</td>
</tr>
</tbody>
</table>

**[Polygons]**

<table>
<thead>
<tr>
<th>;; Subcatchment</th>
<th>X-Coord</th>
<th>Y-Coord</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>3708.61</td>
<td>8543.05</td>
</tr>
<tr>
<td>S1</td>
<td>4834.44</td>
<td>7019.87</td>
</tr>
<tr>
<td>S1</td>
<td>3675.50</td>
<td>4834.44</td>
</tr>
</tbody>
</table>

< additional vertices not listed >

<table>
<thead>
<tr>
<th>;; Link</th>
<th>X-Coord</th>
<th>Y-Coord</th>
</tr>
</thead>
<tbody>
<tr>
<td>S2</td>
<td>6523.18</td>
<td>8079.47</td>
</tr>
<tr>
<td>S2</td>
<td>8112.58</td>
<td>8841.06</td>
</tr>
</tbody>
</table>

**[LABELS]**

<table>
<thead>
<tr>
<th>;; X-Coord</th>
<th>Y-Coord</th>
<th>Label</th>
</tr>
</thead>
<tbody>
<tr>
<td>5033.11</td>
<td>8807.95</td>
<td>&quot;G1&quot;</td>
</tr>
<tr>
<td>1655.63</td>
<td>7450.33</td>
<td>&quot;S1&quot;</td>
</tr>
<tr>
<td>7715.23</td>
<td>7549.67</td>
<td>&quot;S2&quot;</td>
</tr>
</tbody>
</table>
Section: [COORDINATES]

Purpose: Assigns X,Y coordinates to drainage system nodes.

Format: Node  Xcoord  Ycoord

Remarks:  
Node  name of node.  
Xcoord horizontal coordinate relative to origin in lower left of map.  
Ycoord vertical coordinate relative to origin in lower left of map.

Section: [VERTICES]

Purpose: Assigns X,Y coordinates to interior vertex points of curved drainage system links.

Format: Link  Xcoord  Ycoord

Remarks:  
Link  name of link.  
Xcoord horizontal coordinate of vertex relative to origin in lower left of map.  
Ycoord vertical coordinate of vertex relative to origin in lower left of map.

Include a separate line for each interior vertex of the link, ordered from the inlet node to the outlet node.

Straight-line links have no interior vertices and therefore are not listed in this section.

Section: [POLYGONS]

Purpose: Assigns X,Y coordinates to vertex points of polygons that define a subcatchment boundary.

Format: Subcat  Xcoord  Ycoord

Remarks:  
Subcat name of subcatchment.  
Xcoord horizontal coordinate of vertex relative to origin in lower left of map.  
Ycoord vertical coordinate of vertex relative to origin in lower left of map.

Include a separate line for each vertex of the subcatchment polygon, ordered in a consistent clockwise or counter-clockwise sequence.

Section: [SYMBOLS]

Purpose: Assigns X,Y coordinates to rain gage symbols.

Format: Gage  Xcoord  Ycoord

Remarks:  
Gage name of rain gage.
**Section:** [LABELS]

**Purpose:** Assigns X,Y coordinates to user-defined map labels.

**Format:** 
```
Xcoord  Ycoord  Label (Anchor  Font  Size  Bold  Italic)
```

**Remarks:** 
- **Xcoord** horizontal coordinate relative to origin in lower left of map.
- **Ycoord** vertical coordinate relative to origin in lower left of map.
- **Label** text of label surrounded by double quotes.
- **Anchor** name of node or subcatchment that anchors the label on zoom-ins (use an empty pair of double quotes if there is no anchor).
- **Font** name of label’s font (surrounded by double quotes if the font name includes spaces).
- **Size** font size in points.
- **Bold** YES for bold font, NO otherwise.
- **Italic** YES for italic font, NO otherwise.

Use of the anchor node feature will prevent the label from moving outside the viewing area when the map is zoomed in on.

If no font information is provided then a default font is used to draw the label.

**Section:** [BACKDROP]

**Purpose:** Specifies file name and coordinates of map’s backdrop image.

**Formats:**
```
FILE  Fname
DIMENSIONS  X1  Y1  X2  Y2
```

**Remarks:**
- **Fname** name of file containing backdrop image.
- **X1** lower-left X coordinate of backdrop image.
- **Y1** lower-left Y coordinate of backdrop image.
- **X2** upper-right X coordinate of backdrop image.
- **Y2** upper-right Y coordinate of backdrop image.
APPENDIX E - ERROR MESSAGES

ERROR 101: memory allocation error.
There is not enough physical memory in the computer to analyze the study area.

ERROR 103: cannot solve KW equations for Link xxx.
The internal solver for Kinematic Wave routing failed to converge for the specified link at some stage of the simulation.

ERROR 105: cannot open ODE solver.
The system could not open its Ordinary Differential Equation solver.

ERROR 107: cannot compute a valid time step.
A valid time step for runoff or flow routing calculations (i.e., a number greater than 0) could not be computed at some stage of the simulation.

ERROR 108: ambiguous outlet ID name for Subcatchment xxx.
The name of the element identified as the outlet of a subcatchment belongs to both a node and a subcatchment in the project's database.

ERROR 109: invalid parameter values for Aquifer xxx.
The properties entered for an aquifer object were either invalid numbers or were inconsistent with one another (e.g., the soil field capacity was higher than the porosity).

ERROR 111: invalid length for Conduit xxx.
Conduits cannot have zero or negative lengths.

ERROR 113: invalid roughness for Conduit xxx.
Conduits cannot have zero or negative roughness values.

ERROR 114: invalid number of barrels for Conduit xxx.
Conduits must consist of one or more barrels.

ERROR 115: adverse slope for Conduit xxx.
Under Steady or Kinematic Wave routing, all conduits must have positive slopes. This can usually be corrected by reversing the inlet and outlet nodes of the conduit (i.e., right click on the conduit and select Reverse from the popup menu that appears). Adverse slopes are permitted under Dynamic Wave routing.

ERROR 117: no cross section defined for Link xxx.
A cross section geometry was never defined for the specified link.

ERROR 119: invalid cross section for Link xxx.
Either an invalid shape or invalid set of dimensions was specified for a link's cross section.

ERROR 121: missing or invalid pump curve assigned to Pump xxx.
Either no pump curve or an invalid type of curve was specified for a pump.
ERROR 131:  the following links form cyclic loops in the drainage system.
The Steady and Kinematic Wave flow routing methods cannot be applied to
systems where a cyclic loop exists (i.e., a directed path along a set of links that
begins and ends at the same node). The same is true for Dynamic Wave routing
where water quality analysis is performed. Most often the cyclic nature of the
loop can be eliminated by reversing the direction of one of its links (i.e.,
switching the inlet and outlet nodes of the link). The names of the links that form
the loop will be listed following this message.

ERROR 133:  Node xxx has more than one outlet link.
Under Steady and Kinematic Wave flow routing, a junction node can have only a
single outlet link.

ERROR 134:  Node xxx has more than one DUMMY outlet link.
Only a single conduit with a DUMMY cross-section can be directed out of a
node.

ERROR 135:  Divider xxx does not have two outlet links.
Flow divider nodes must have two outlet links connected to them.

ERROR 136:  Divider xxx has invalid diversion link.
The link specified as being the one carrying the diverted flow from a flow divider
node was defined with a different inlet node.

ERROR 137:  Weir Divider xxx has invalid parameters.
The parameters of a Weir-type divider node either are non-positive numbers or
are inconsistent (i.e., the value of the discharge coefficient times the weir height
raised to the 3/2 power must be greater than the minimum flow parameter).

ERROR 138:  Node xxx has initial depth greater than maximum depth.
Self-explanatory.

ERROR 139:  Regulator xxx is the outlet of a non-storage node.
Under Steady or Kinematic Wave flow routing, orifices, weirs, and outlet links
can only be used as outflow links from storage nodes.

ERROR 141:  Outfall xxx has more than 1 inlet link or an outlet link.
An outfall node is only permitted to have one link attached to it.

ERROR 143:  Regulator xxx has invalid cross-section shape.
An orifice must have either a CIRCULAR or RECT_CLOSED shape, while a
weir must have either a RECT_OPEN, TRAPEZOIDAL, or TRIANGULAR
shape.

ERROR 145:  Drainage system has no acceptable outlet nodes.
Under Dynamic Wave flow routing, there must be at least one node designated as
an outfall.

ERROR 151:  a Unit Hydrograph in set xxx has invalid time base.
The time base of a Unit hydrograph must be greater than 0.
ERROR 153:  **a Unit Hydrograph in set xxx has invalid response ratios.**  
The response ratios for a set of Unit Hydrographs (the short-, medium-, and long-term response hydrographs) must be between 0 and 1.0 and cannot add up to a value greater than 1.0

ERROR 155:  **invalid sewer area for RDII at Node xxx.**  
The sewer area contributing RDII inflow to a node cannot be a negative number.

ERROR 161:  **cyclic dependency in treatment functions at Node xxx.**  
An example would be where the removal of pollutant 1 is defined as a function of the removal of pollutant 2 while the removal of pollutant 2 is defined as a function of the removal of pollutant 1.

ERROR 171:  **Curve xxx has its data out of sequence.**  
The X-values of a curve object must be entered in increasing order.

ERROR 173:  **Time Series xxx has its data out of sequence.**  
The time (or date/time) values of a time series must be entered in sequential order.

ERROR 191:  **simulation start date comes after ending date.**  
Self-explanatory.

ERROR 193:  **report start date comes after ending date.**  
Self-explanatory.

ERROR 195:  **reporting time step is less than routing time step.**  
Self-explanatory.

ERROR 200:  **one or more errors in input file.**  
This message appears when one or more input file parsing errors (the 200-series errors) occur.

ERROR 201:  **too many characters in input line.**  
A line in the input file cannot exceed 1024 characters.

ERROR 203:  **too few items at line n of input file.**  
Not enough data items were supplied on a line of the input file.

ERROR 205:  **invalid keyword at line n of input file.**  
An unrecognized keyword was encountered when parsing a line of the input file.

ERROR 207:  **duplicate ID name at line n of input file.**  
An ID name used for an object was already assigned to an object of the same category.

ERROR 209:  **undefined object xxx at line n of input file.**  
A reference was made to an object that was never defined. An example would be if node 123 were designated as the outlet point of a subcatchment, yet no such node was ever defined in the study area.
ERROR 211: invalid number xxx at line n of input file.  
Either a string value was encountered where a numerical value was expected or an invalid number (e.g., a negative value) was supplied.

ERROR 213: invalid date/time xxx at line n of input file.  
An invalid format for a date or time was encountered. Dates must be entered as month/day/year and times as either decimal hours or as hour:minute:second.

ERROR 217: control rule clause out of sequence at line n of input file.  
Errors of this nature can occur when the format for writing control rules is not followed correctly (see Section C.3).

ERROR 219: data provided for unidentified transect at line n of input file.  
A GR line with Station-Elevation data was encountered in the [TRANSECTS] section of the input file after an NC line but before any X1 line that contains the transect’s ID name.

ERROR 221: transect station out of sequence at line n of input file.  
The station distances specified for the transect of an irregular cross section must be in increasing numerical order starting from the left bank.

ERROR 223: Transect xxx has too few stations.  
A transect for an irregular cross section must have at least 2 stations defined for it.

ERROR 225: Transect xxx has too many stations.  
A transect cannot have more than 1500 stations defined for it.

ERROR 227: Transect xxx has no Manning's N.  
No Manning’s N was specified for a transect (i.e., there was no NC line in the [TRANSECTS] section of the input file.

ERROR 229: Transect xxx has invalid overbank locations.  
The distance values specified for either the left or right overbank locations of a transect do not match any of the distances listed for the transect's stations.

ERROR 231: Transect xxx has no depth.  
All of the stations for a transect were assigned the same elevation.

ERROR 233: invalid treatment function expression at line n of input file.  
A treatment function supplied for a pollutant at a specific node is either not a correctly formed mathematical expression or refers to unknown pollutants, process variables, or math functions.

ERROR 301: files share same names.  
The input, report, and binary output files specified on the command line cannot have the same names.

ERROR 303: cannot open input file.
The input file either does not exist or cannot be opened (e.g., it might be in use by another program).

**ERROR 305**: cannot open report file.  
The report file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

**ERROR 307**: cannot open binary results file.  
The binary output file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

**ERROR 309**: error writing to binary results file.  
There was an error in trying to write results to the binary output file (e.g., the disk might be full or the file size exceed the limit imposed by the operating system).

**ERROR 311**: error reading from binary results file.  
The command line version of SWMM could not read results saved to the binary output file when writing results to the report file.

**ERROR 313**: cannot open scratch rainfall interface file.  
SWMM could not open the temporary file it uses to collate data together from external rainfall files.

**ERROR 315**: cannot open rainfall interface file xxx.  
SWMM could not open the specified rainfall interface file, possibly because it does not exist or because the user does not have write privileges to its directory.

**ERROR 317**: cannot open rainfall data file xxx.  
An external rainfall data file could not be opened, most likely because it does not exist.

**ERROR 319**: invalid format for rainfall interface file.  
SWMM was trying to read data from a designated rainfall interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

**ERROR 321**: no data in rainfall interface file for gage xxx.  
This message occurs when a project wants to use a previously saved rainfall interface file, but cannot find any data for one of its rain gages in the interface file.

**ERROR 323**: cannot open runoff interface file xxx.  
A runoff interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

**ERROR 325**: incompatible data found in runoff interface file.  
SWMM was trying to read data from a designated runoff interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).
ERROR 327: attempting to read beyond end of runoff interface file.
This error can occur when a previously saved runoff interface file is being used in a simulation with a longer duration than the one that created the interface file.

ERROR 329: error in reading from runoff interface file.
A format error was encountered while trying to read data from a previously saved runoff interface file.

ERROR 330: hotstart interface files have same names.
In cases where a run uses one hotstart interface file to start a simulation and another to save results at the end of the simulation, the two files cannot both have the same name.

ERROR 331: cannot open hotstart interface file xxx.
A hotstart interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 333: incompatible data found in hotstart interface file.
SWMM was trying to read data from a designated hotstart interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

ERROR 335: error in reading from hotstart interface file.
A format error was encountered while trying to read data from a previously saved hotstart interface file.

ERROR 336: no climate file specified for evaporation and/or wind speed.
This error occurs when the user specifies that evaporation or wind speed data will be read from an external climate file, but no name is supplied for the file.

ERROR 337: cannot open climate file xxx.
An external climate data file could not be opened, most likely because it does not exist.

ERROR 338: error in reading from climate file xxx.
SWMM was trying to read data from an external climate file with the wrong format.

ERROR 339: attempt to read beyond end of climate file xxx.
The specified external climate does not include data for the period of time being simulated.

ERROR 341: cannot open scratch RDII interface file.
SWMM could not open the temporary file it uses to store RDII flow data.

ERROR 343: cannot open RDII interface file xxx.
An RDII interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

ERROR 345: invalid format for RDII interface file.
SWMM was trying to read data from a designated RDII interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

**ERROR 351:** cannot open routing interface file *xxx.*
A routing interface file could not be opened, possibly because it does not exist or because the user does not have write privileges to its directory.

**ERROR 353:** invalid format for routing interface file *xxx.*
SWMM was trying to read data from a designated routing interface file with the wrong format (i.e., it may have been created for some other project or actually be some other type of file).

**ERROR 355:** mis-matched names in routing interface file *xxx.*
The names of pollutants found in a designated routing interface file do not match the names used in the current project.

**ERROR 357:** inflows and outflows interface files have same name.
In cases where a run uses one routing interface file to provide inflows for a set of locations and another to save outflow results, the two files cannot both have the same name.