

**STORM WATER MANAGEMENT MODEL**

**USER'S MANUAL**

**Version 5.0**

By

Lewis A. Rossman  
Water Supply and Water Resources Division  
National Risk Management Research Laboratory  
Cincinnati, OH 45268

NATIONAL RISK MANAGEMENT RESEARCH LABORATORY  
OFFICE OF RESEARCH AND DEVELOPMENT  
U.S. ENVIRONMENTAL PROTECTION AGENCY  
CINCINNATI, OH 45268

## **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

Lewis Rossman  
Trent Schade  
Daniel Sullivan (retired)

### CDM

Robert Dickinson  
Carl Chan  
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.

## CONTENTS

<b>CHAPTER 1 - INTRODUCTION .....</b>	<b>1</b>
1.1 What is SWMM.....	1
1.2 Modeling Capabilities .....	1
1.3 Typical Applications of SWMM .....	2
1.4 Installing EPA SWMM .....	3
1.5 Steps in Using SWMM.....	3
1.6 About This Manual.....	4
<b>CHAPTER 2 - QUICK START TUTORIAL .....</b>	<b>7</b>
2.1 Example Study Area.....	7
2.2 Project Setup .....	7
2.3 Drawing Objects.....	10
2.4 Setting Object Properties.....	12
2.5 Running a Simulation.....	17
2.6 Simulating Water Quality.....	26
2.7 Running a Continuous Simulation .....	30
<b>CHAPTER 3 - SWMM'S CONCEPTUAL MODEL .....</b>	<b>33</b>
3.1 Introduction .....	33
3.2 Visual Objects .....	33
3.3 Non-Visual Objects .....	43
3.4 Computational Methods .....	53
<b>CHAPTER 4 - SWMM'S MAIN WINDOW .....</b>	<b>59</b>
4.1 Overview .....	59

4.2	Main Menu .....	60
4.3	Toolbars.....	63
4.4	Status Bar .....	65
4.5	Study Area Map.....	66
4.6	Data Browser .....	67
4.7	Map Browser .....	67
4.8	Property Editor .....	69
4.9	Setting Program Preferences .....	70
<b>CHAPTER 5 - WORKING WITH PROJECTS .....</b>		<b>73</b>
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	Measurement Units.....	76
5.6	Link Offset Conventions .....	77
5.7	Calibration Data .....	77
5.8	Viewing All Project Data .....	79
<b>CHAPTER 6 - WORKING WITH OBJECTS.....</b>		<b>81</b>
6.1	Types of Objects.....	81
6.2	Adding Objects.....	81
6.3	Selecting and Moving Objects .....	84
6.4	Editing Objects .....	84
6.5	Converting an Object.....	85
6.6	Copying and Pasting Objects .....	85
6.7	Shaping and Reversing Links.....	86
6.8	Shaping a Subcatchment .....	86

6.9	Deleting an Object.....	87
6.10	Editing or Deleting a Group of Objects.....	87
<b>CHAPTER 7 - WORKING WITH THE MAP .....</b>		<b>89</b>
7.1	Selecting a Map Theme.....	89
7.2	Setting the Map's Dimensions .....	89
7.3	Utilizing a Backdrop Image .....	90
7.4	Measuring Distances .....	93
7.5	Zooming the Map.....	94
7.6	Panning the Map.....	94
7.7	Viewing at Full Extent .....	94
7.8	Finding an Object.....	95
7.9	Submitting a Map Query .....	95
7.10	Using the Map Legends.....	96
7.11	Using the Overview Map .....	97
7.12	Setting Map Display Options .....	98
7.13	Exporting the Map.....	102
<b>CHAPTER 8 - RUNNING A SIMULATION.....</b>		<b>105</b>
8.1	Setting Simulation Options .....	105
8.2	Setting Reporting Options.....	111
8.3	Starting a Simulation .....	112
8.4	Troubleshooting Results.....	112
<b>CHAPTER 9 - VIEWING RESULTS .....</b>		<b>117</b>
9.1	Viewing a Status Report.....	117
9.2	Variables That Can Be Viewed .....	120
9.3	Viewing Results on the Map .....	120
9.4	Viewing Results with a Graph.....	121

9.5	Customizing a Graph's Appearance .....	126
9.6	Viewing Results with a Table .....	130
9.7	Viewing a Statistics Report .....	132
<b>CHAPTER 10 - PRINTING AND COPYING .....</b>		<b>137</b>
10.1	Selecting a Printer .....	137
10.2	Setting the Page Format .....	138
10.3	Print Preview .....	139
10.4	Printing the Current View .....	139
10.5	Copying to the Clipboard or to a File .....	139
<b>CHAPTER 11 - FILES USED BY SWMM .....</b>		<b>141</b>
11.1	Project Files .....	141
11.2	Report and Output Files .....	141
11.3	Rainfall Files .....	142
11.4	Climate Files .....	142
11.5	Calibration Files .....	143
11.6	Time Series Files .....	144
11.7	Interface Files .....	145
<b>CHAPTER 12 - USING ADD-IN TOOLS .....</b>		<b>149</b>
12.1	What Are Add-In Tools.....	149
12.2	Configuring Add-In Tools.....	150
<b>APPENDIX A - USEFUL TABLES .....</b>		<b>155</b>
A.1	Units of Measurement .....	155
A.2	Soil Characteristics.....	156
A.3	NRCS Hydrologic Soil Group Definitions.....	157
A.4	SCS Curve Numbers <sup>1</sup> .....	158



A.5	Depression Storage.....	159
A.6	Manning’s n – Overland Flow.....	159
A.7	Manning’s n – Closed Conduits.....	160
A.8	Manning’s n – Open Channels.....	161
A.9	Water Quality Characteristics of Urban Runoff.....	161
A.10	Culvert Code Numbers.....	162
A.11	Culvert Entrance Loss Coefficients.....	164

**APPENDIX B - VISUAL OBJECT PROPERTIES ..... 165**

B.1	Rain Gage Properties.....	165
B.2	Subcatchment Properties.....	166
B.3	Junction Properties.....	167
B.4	Outfall Properties.....	168
B.5	Flow Divider Properties.....	169
B.6	Storage Unit Properties.....	170
B.7	Conduit Properties.....	171
B.8	Pump Properties.....	172
B.9	Orifice Properties.....	172
B.10	Weir Properties.....	173
B.11	Outlet Properties.....	174
B.12	Map Label Properties.....	175

**APPENDIX C - SPECIALIZED PROPERTY EDITORS ..... 177**

C.1	Aquifer Editor.....	177
C.2	Climatology Editor.....	178
C.3	Control Rules Editor.....	184
C.4	Cross-Section Editor.....	188
C.5	Curve Editor.....	189

C.6	Groundwater Flow Editor.....	190
C.7	Infiltration Editor.....	192
C.8	Inflows Editor.....	195
C.9	Initial Buildup Editor.....	199
C.10	Land Use Editor.....	199
C.11	Land Use Assignment Editor .....	203
C.12	Pollutant Editor.....	204
C.13.	Snow Pack Editor .....	205
C.14	Time Pattern Editor .....	208
C.15	Time Series Editor.....	209
C.16	Title/Notes Editor .....	210
C.17	Transect Editor .....	211
C.18	Treatment Editor.....	212
C.19	Unit Hydrograph Editor.....	213
	<b>APPENDIX D - COMMAND LINE SWMM.....</b>	<b>217</b>
	<b>APPENDIX E - ERROR AND WARNING MESSAGES .....</b>	<b>259</b>

# CHAPTER 1 - INTRODUCTION

---

## 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<sup>1</sup> and has undergone several major upgrades since then<sup>2</sup>. 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

---

<sup>1</sup> Metcalf & Eddy, Inc., University of Florida, Water Resources Engineers, Inc. "Storm Water Management Model, Volume I – Final Report", 11024DOC07/71, Water Quality Office, Environmental Protection Agency, Washington, DC, July 1971.

<sup>2</sup> Huber, W. C. and Dickinson, R.E., "Storm Water Management Model, Version 4: User's Manual", EPA/600/3-88/001a, Environmental Research Laboratory, U.S. Environmental Protection Agency, Athens, GA, October 1992.

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/Vista 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, and 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`.)

Under Windows 2000, XP, and Vista a user's personal settings for running SWMM are stored in a folder named EPASWMM under the user's Application Data directory. If you need to save these settings to a different location, you can install a shortcut to SWMM 5 on the 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 personal settings will be stored. An example might be:

*`"c:\Program Files\EPA SWMM 5.0\epaswmm5.exe" /s "My Folders\SWMM5"`*.

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 5.4).
2. Draw a network representation of the physical components of the study area (see Section 6.2).
3. Edit the properties of the objects that make up the system (see Section 6.4).
4. Select a set of analysis options (see Section 8.1).
5. Run a simulation (see Section 8.2).

6. View the results of the simulation (see Chapter 9).

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 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.

(This page intentionally left blank.)



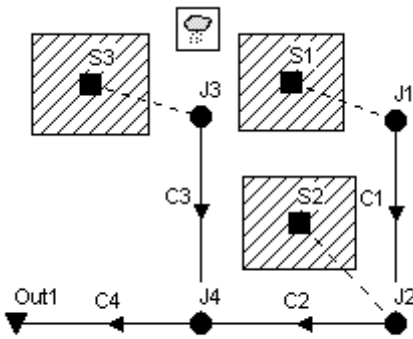
## CHAPTER 2 - QUICK START TUTORIAL

---

*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<sup>3</sup> *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.**

### 2.2 Project Setup

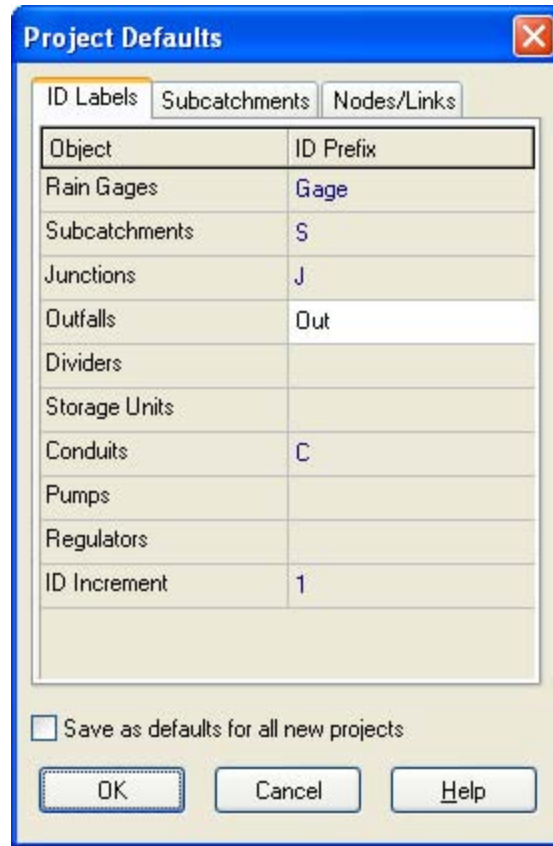
Our first task is to create a new SWMM project and make sure that certain default options are selected. Using these defaults will simplify the data entry tasks later on.

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.

---

<sup>3</sup> 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.

- 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.



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

- On the Subcatchments page of the dialog set the following default values:
 

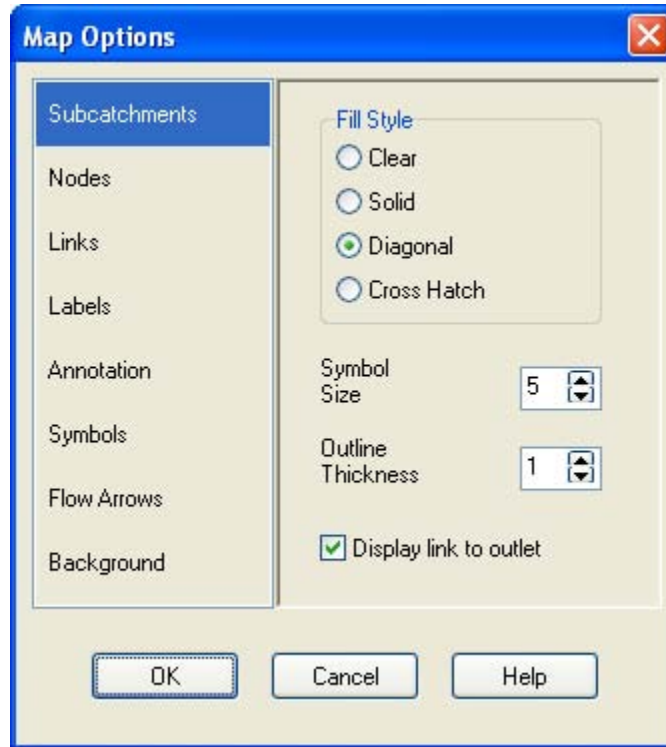
Area	4
Width	400
% Slope	0.5
% Imperv.	50
N-Imperv.	0.01
N-Perv.	0.10
Dstore-Imperv.	0.05
Dstore-Perv	0.05
%Zero-Imperv.	25
Infil. Model	<click to edit>
- Method	Green-Ampt
- Suction Head	3.5
- Conductivity	0.5
- Initial Deficit	0.26

5. On the Nodes/Links page set the following default values:
 

Node Invert	0
Node Max. Depth	4
Node Poned Area	0
Conduit Length	400
Conduit Geometry	<click to edit>
- Barrels	1
- Shape	Circular
- Max. Depth	1.0
Conduit Roughness	0.01
Flow Units	CFS
Link Offsets	DEPTH
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 Subcatchments, 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.



**Figure 2-3. Map Options 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.

### 2.3 Drawing Objects

We are now ready to begin adding components to the Study Area Map<sup>4</sup>. We will start with the subcatchments.

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.



---

<sup>4</sup> 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.

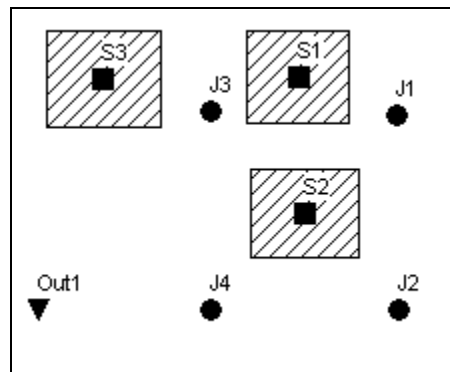
2. Move the mouse to the map location where one of the corners of subcatchment *S1* lies and left-click the mouse.
3. Do the same for the next three corners and then right-click the mouse (or hit the **Enter** key) to close up the rectangle that represents subcatchment *S1*. You can press the **Esc** 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*<sup>5</sup>.

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.**

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*.


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.

<sup>5</sup> 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.


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.



**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 *Subcatchment* as the type 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.



**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<sup>6</sup>.

<u>Node</u>	<u>Invert</u>
<i>J1</i>	96
<i>J2</i>	90
<i>J3</i>	93
<i>J4</i>	88
<i>Out1</i>	85

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, select conduit *C4* into the Property Editor and set the Max. Depth value to 1.5.

---

<sup>6</sup> 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.




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:

<i>Rain Format</i>	<i>INTENSITY</i>
<i>Rain Interval</i>	<i>1:00</i>
<i>Data Source</i>	<i>TIMESERIES</i>
<i>Series Name</i>	<i>TS1</i>

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 2-7)<sup>7</sup>.
3. Enter *TS1* in the Time Series Name field.
4. Enter the values shown in Figure 2-7 into the *Time* and *Value* columns of the data entry grid (leave the *Date* column blank<sup>8</sup>).
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.

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 2-8), 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.)
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.

---

<sup>7</sup> 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.

<sup>8</sup> 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.

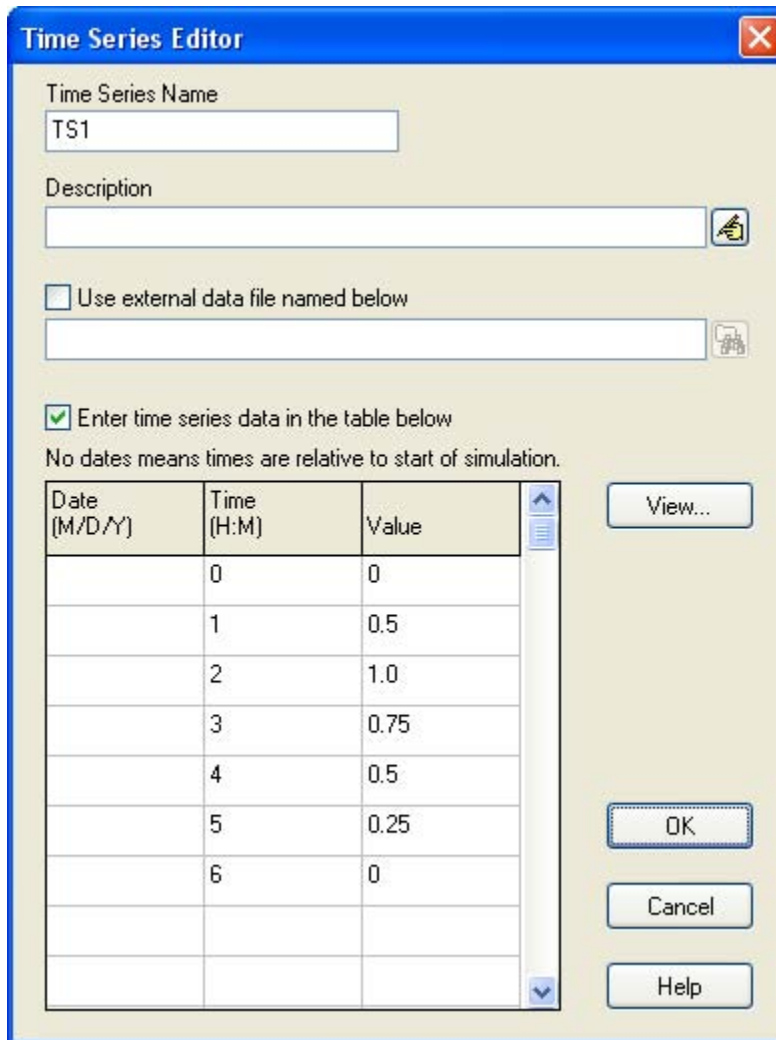


Figure 2-7. Time Series Editor.

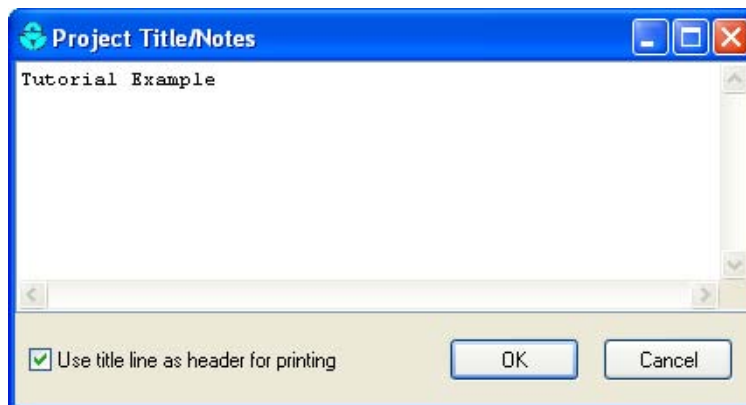



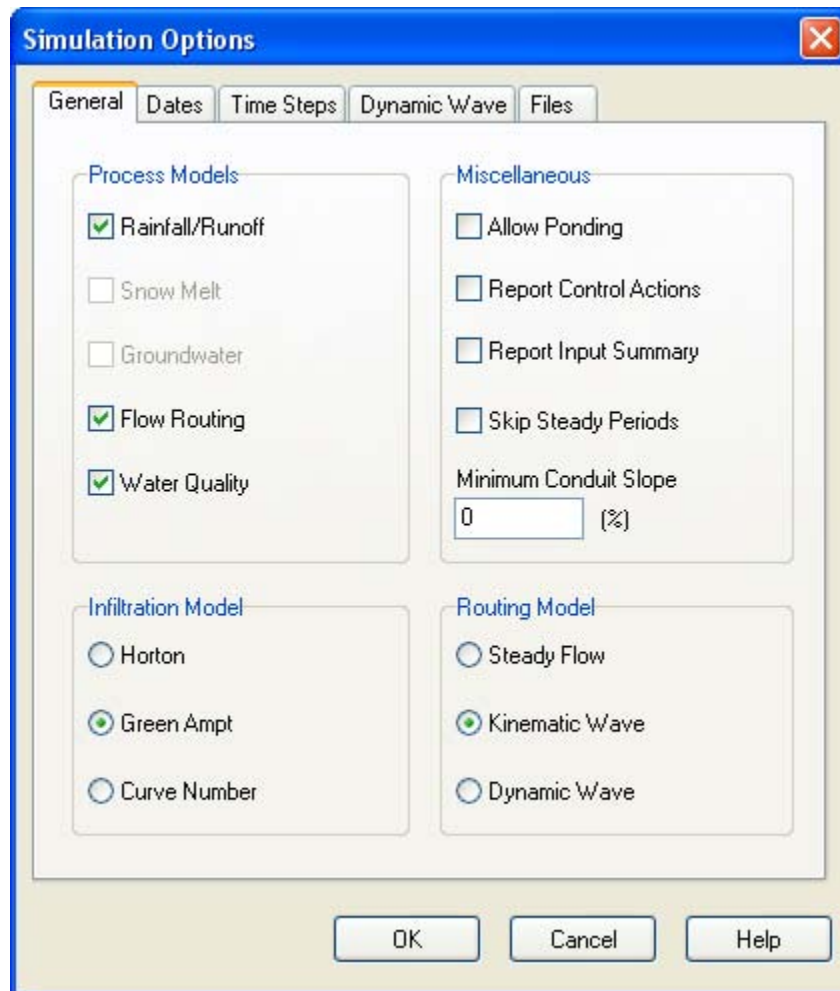
Figure 2-8. Title/Notes Editor.

## 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 2-9), select Kinematic Wave as the flow routing method. The infiltration method should already be set 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 60 seconds.
5. Click **OK** to close the Simulation Options dialog.



**Figure 2-9. 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 2-10. 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 (-0.22% 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 Flooding Summary table (not shown in Figure 2-11) indicates there was internal flooding in the system at node *J2*<sup>9</sup>.
- The Conduit Surge 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.

## Viewing Results on the Map

Simulation results (as well as some design parameters, such as subcatchment area, node invert elevation, and 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 appearing in the Themes panel. In Figure 2-11, 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.

---

<sup>9</sup> In SWMM, flooding will occur whenever the water surface at a node exceeds the maximum defined depth or if more flow volume enters a node than can be stored or released during a given time step. 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.

```

EPA STORM WATER MANAGEMENT MODEL - VERSION 5.0 (Build 5.0.015)
-----

Tutorial Example

*****
NOTE: The summary statistics displayed in this report are
based on results found at every computational time step,
not just on results from each reporting time step.
*****

*****
Analysis Options
*****
Flow Units ..... CFS
Process Models:
  Rainfall/Runoff ..... YES
  Snowmelt ..... NO
  Groundwater ..... NO
  Flow Routing ..... YES
  Water Quality ..... NO
Infiltration Method ..... GREEN_AMPT
Flow Routing Method ..... KINWAVE
Starting Date ..... JAN-01-1998 00:00:00
Ending Date ..... JAN-01-1998 12:00:00
Antecedent Dry Days ..... 5.0
Report Time Step ..... 00:15:00
Wet Time Step ..... 00:15:00
Dry Time Step ..... 01:00:00
Routing Time Step ..... 60.00 sec


*****
Runoff Quantity Continuity          Volume          Depth
*****                          acre-feet        inches
*****                          -----          -----
Total Precipitation .....          3.000          3.000
Evaporation Loss .....            0.000          0.000
Infiltration Loss .....           1.750          1.750
Surface Runoff .....              1.241          1.241
Final Surface Storage ....         0.016          0.016
Continuity Error (%) .....        -0.218

*****
Flow Routing Continuity          Volume          Volume
*****                          acre-feet        10^6 gal
*****                          -----          -----
Dry Weather Inflow .....          0.000          0.000
Wet Weather Inflow .....          1.246          0.406
Groundwater Inflow .....          0.000          0.000
RDII Inflow .....                 0.000          0.000
External Inflow .....              0.000          0.000
External Outflow .....             1.191          0.388
Internal Outflow .....              0.055          0.018
Evaporation Loss .....             0.000          0.000
Initial Stored Volume ....         0.000          0.000
Final Stored Volume .....          0.000          0.000
Continuity Error (%) .....        -0.032

```


**Figure 2-10. Portion of the Status Report for initial simulation run.**

- 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 **Tools >> Map Display Options** and then select the Annotation page of the Map Options dialog. Use the check boxes for Subcatchment Values, Node Values, and Link Values to specify what kind of annotation to add.


6. The Date / Time of Day / Elapsed Time controls on the Map Browser can be used to move through the simulation results in time. Figure 2-11 depicts results at 5 hours and 45 minutes into the simulation.
7. You can use the controls in the Animator panel of the Map Browser (see Figure 2-11) to animate the map display through time. 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.
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 2-12):

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 2-13.

After a plot is created you can:

- customize its appearance by selecting **Report >> Customize** 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.).

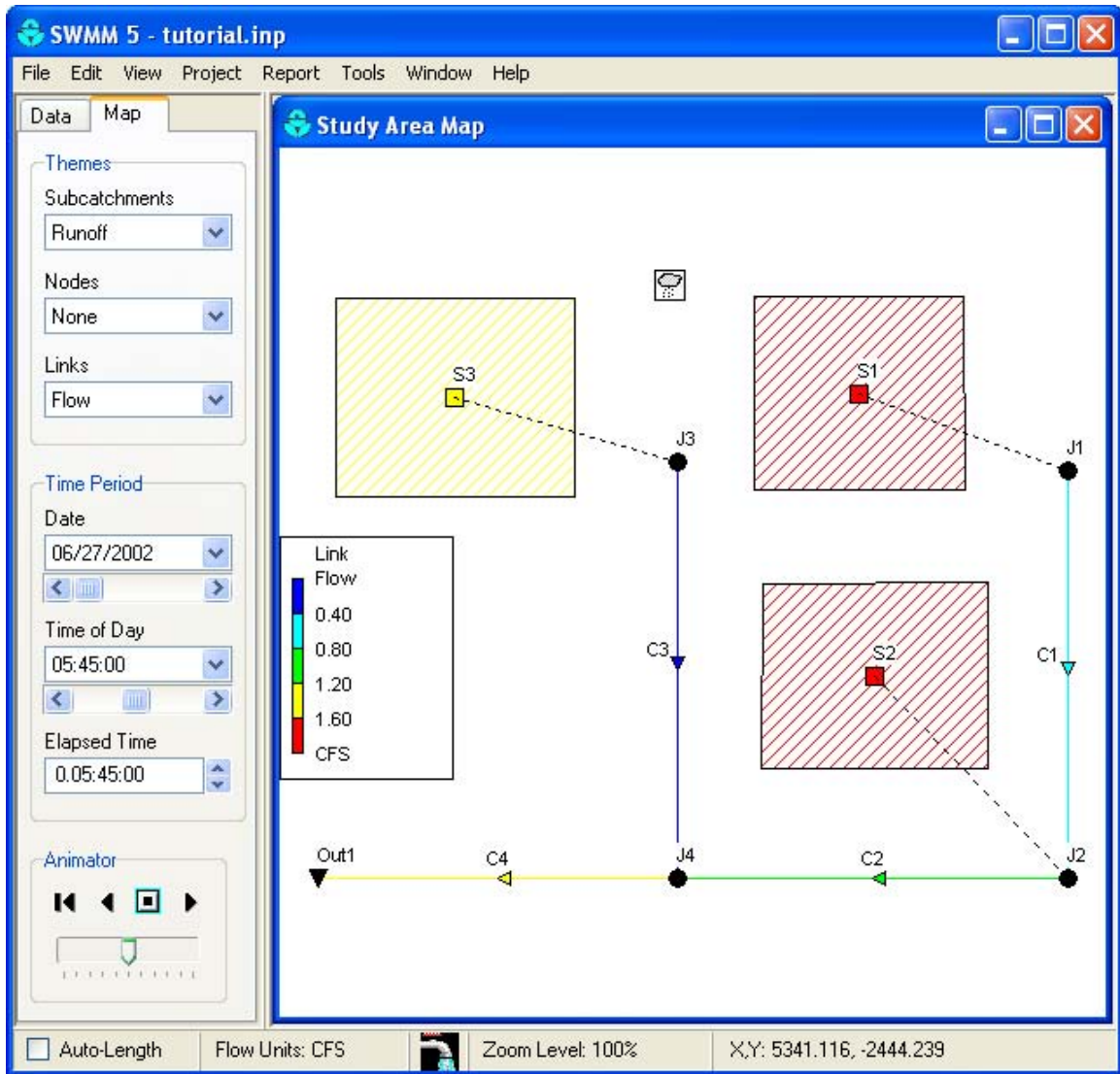


Figure 2-11. Example of viewing color-coded results on the Study Area Map.

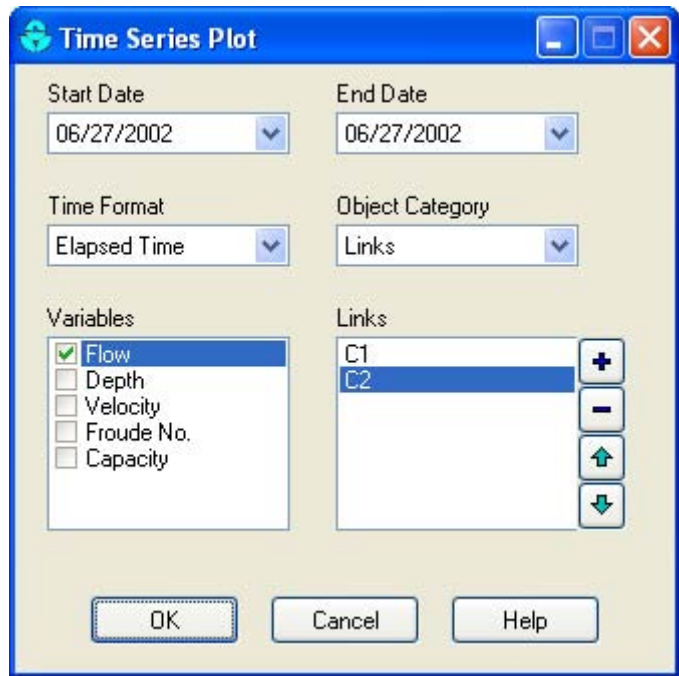


Figure 2-12. Time Series Plot dialog.

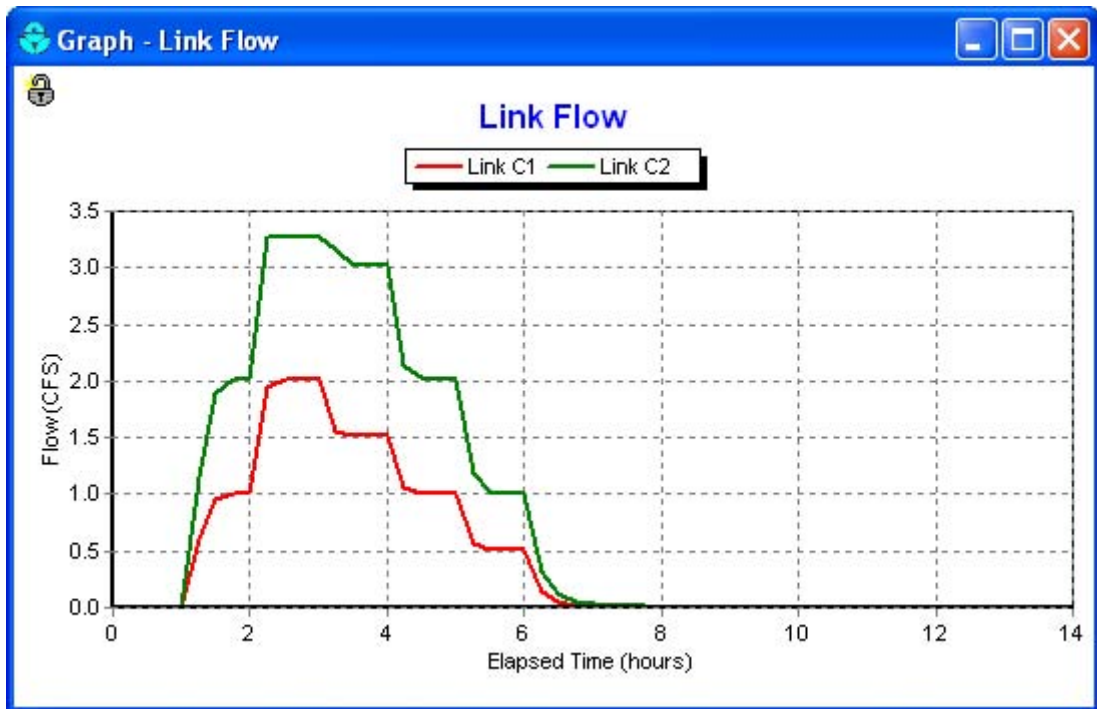




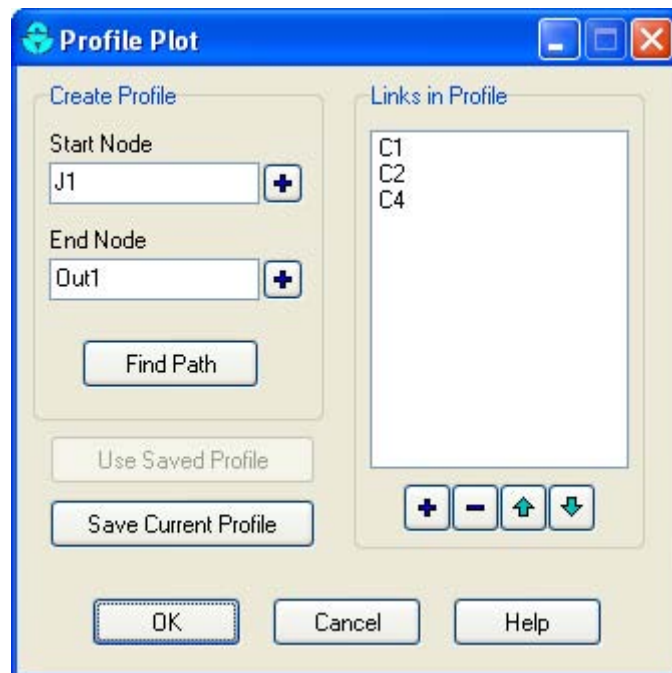
Figure 2-13. Time series plot of results from initial simulation run.



## Viewing a Profile Plot

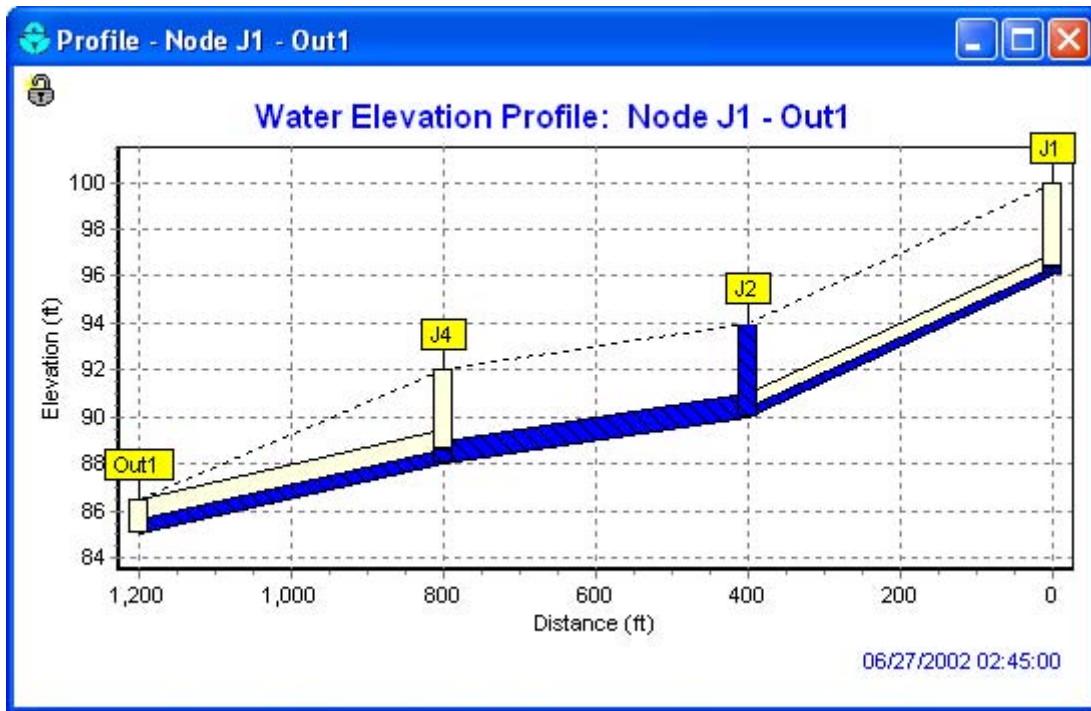
SWMM can generate profile plots showing how water surface depth varies across a path of connected nodes and links. Let's create such a plot for the conduits connecting junction *J1* to the outfall *Out1* of our example drainage system. To do this:

1. Select **Report >> Graph >> Profile** or simply click  on the Standard Toolbar.
2. Either enter *J1* in the Start Node field of the Profile Plot dialog that appears (see Figure 2-14) or select it on the map or from the Data Browser and click the  button next to the field.



**Figure 2-14. 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 2-15).



**Figure 2-15. 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

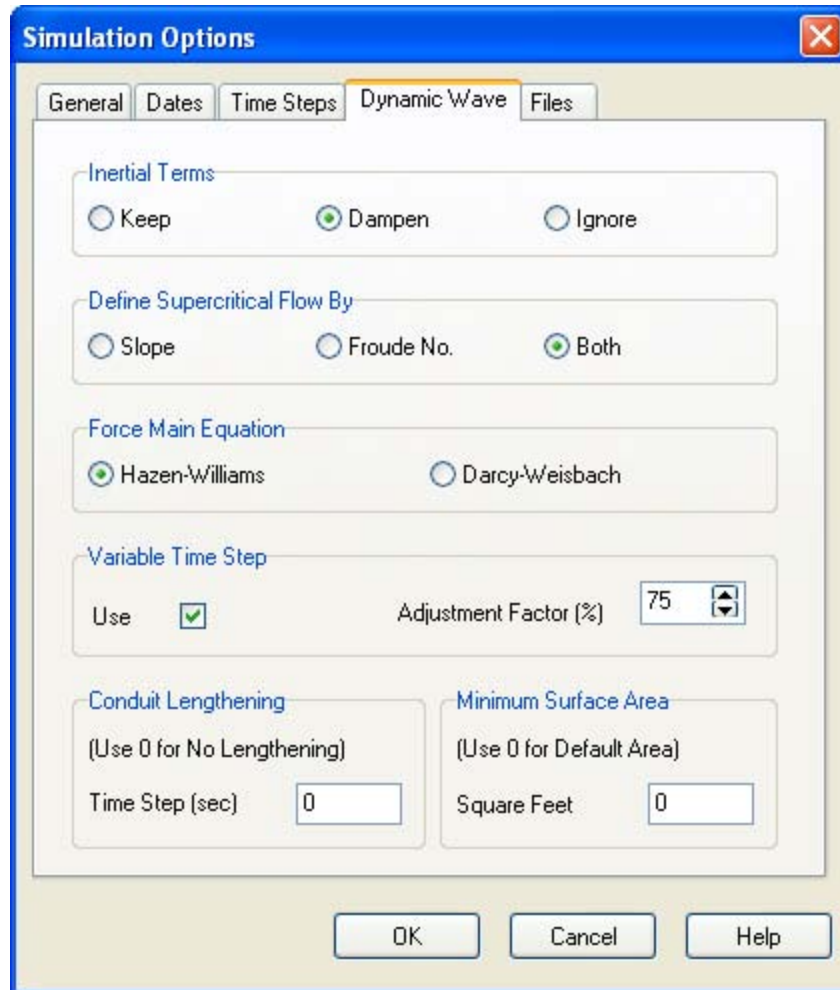
In the analysis just run we chose to use the Kinematic Wave method of routing flows through our drainage system. This is an efficient but simplified approach that cannot deal with such phenomena as backwater effects, pressurized flow, flow reversal, and non-dendritic layouts. SWMM also includes a Dynamic 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 2-16<sup>10</sup>.



The screenshot shows the 'Simulation Options' dialog box with the 'Dynamic Wave' tab selected. The dialog has five tabs: 'General', 'Dates', 'Time Steps', 'Dynamic Wave', and 'Files'. The 'Dynamic Wave' tab contains the following settings:

- Inertial Terms:** Radio buttons for 'Keep', 'Dampen' (selected), and 'Ignore'.
- Define Supercritical Flow By:** Radio buttons for 'Slope', 'Froude No.', and 'Both' (selected).
- Force Main Equation:** Radio buttons for 'Hazen-Williams' (selected) and 'Darcy-Weisbach'.
- Variable Time Step:** A checked 'Use' checkbox and an 'Adjustment Factor (%)' spinner set to 75.
- Conduit Lengthening:** A text box for 'Time Step (sec)' containing the value 0.
- Minimum Surface Area:** A text box for 'Square Feet' containing the value 0.

At the bottom of the dialog are three buttons: 'OK', 'Cancel', and 'Help'.

Figure 2-16. Dynamic Wave simulation options.

4. Click **OK** to close the form and select **Project >> Run Simulation** (or click the  button) to re-run the analysis.

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 3.52 cfs to 4.04 cfs.

---

<sup>10</sup> 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.

## 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<sup>11</sup>.

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:

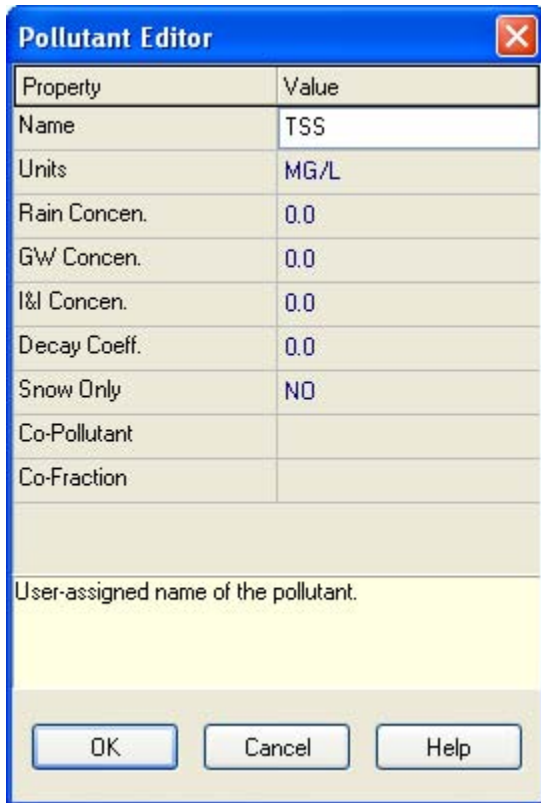
1. Under the Quality category in the Data Browser, select the Pollutants sub-category beneath it.
2. Click the **+** button to add a new pollutant to the project.
3. In the Pollutant Editor dialog that appears (see Figure 2-17), enter TSS for the pollutant name and leave the other data fields at their 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:

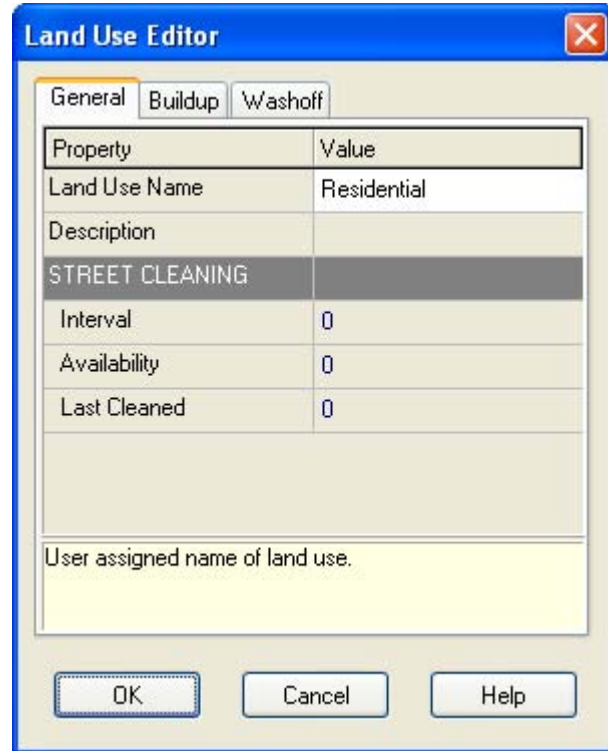
1. Under the Quality category in the Data Browser, select the Land Uses sub-category and click the **+** button.
2. In the Land Use Editor dialog that appears (see Figure 2-18), enter Residential in the Name field and then click the OK button.
3. Repeat steps 1 and 2 to create the Undeveloped land use category.

---

<sup>11</sup> 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 dependent inflow/infiltration




**Figure 2-17. Pollutant Editor dialog.**



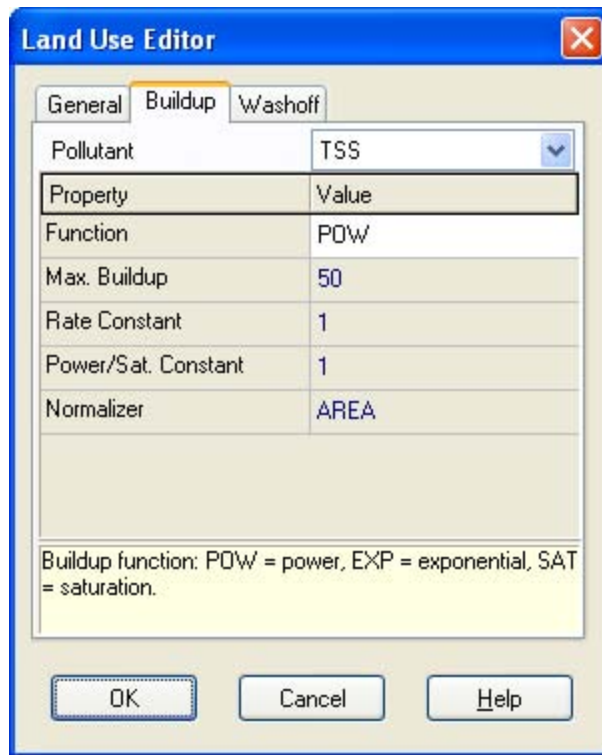
**Figure 2-18. Land Use Editor dialog.**

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 2-19).
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.

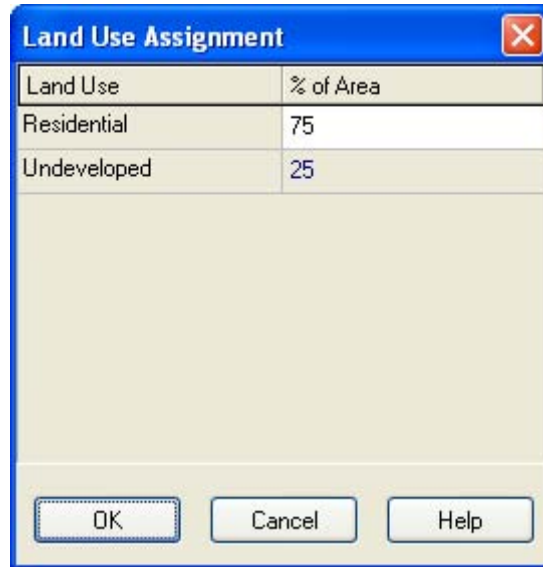
Now do the same for the Undeveloped land use category, except use a maximum buildup of 25, a buildup rate constant of 0.5, a buildup power of 1, and a washoff EMC of 50.



**Figure 2-19. 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 2-20). 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.



**Figure 2-20. Land Use Assignment dialog.**

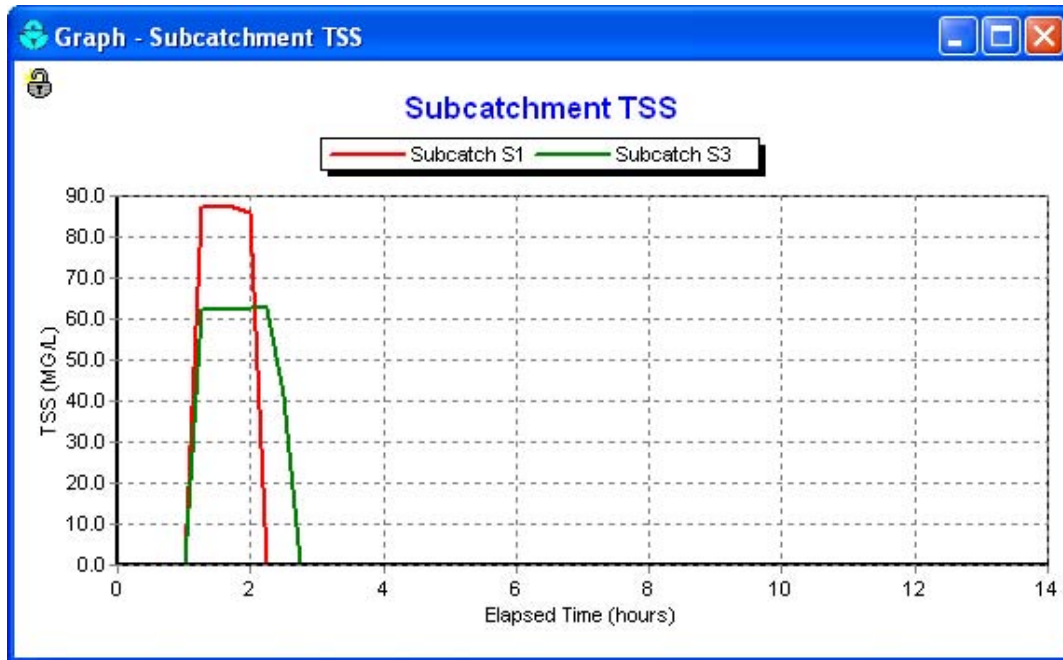
Before we simulate the runoff quantities of TSS and Lead from our study area, an initial buildup of TSS should be defined so it can be washed off during our single rainfall event. We can either specify the number of antecedent dry days prior to the simulation or directly specify the initial buildup mass 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 2.5 lbs of buildup added during the dry periods of the simulation. About 47.9 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 2-21, 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.




**Figure 2-21. 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:

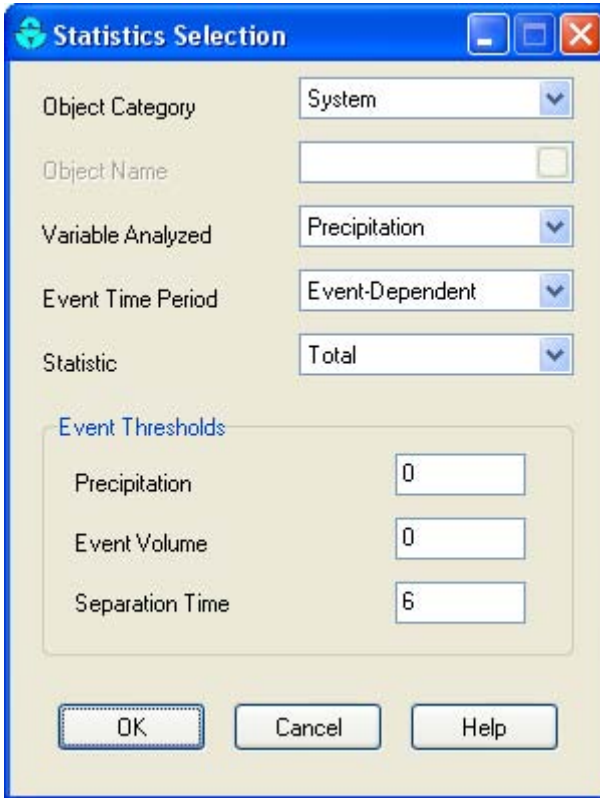
1. Select the rain gage *Gage1* into the Property Editor.
2. Change the selection of Data Source to FILE.
3. Select the File Name data field and click the ellipsis button (or press the **Enter** key) to bring up a standard Windows File Selection dialog.
4. Navigate to the folder where the SWMM example 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 300 seconds.
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).

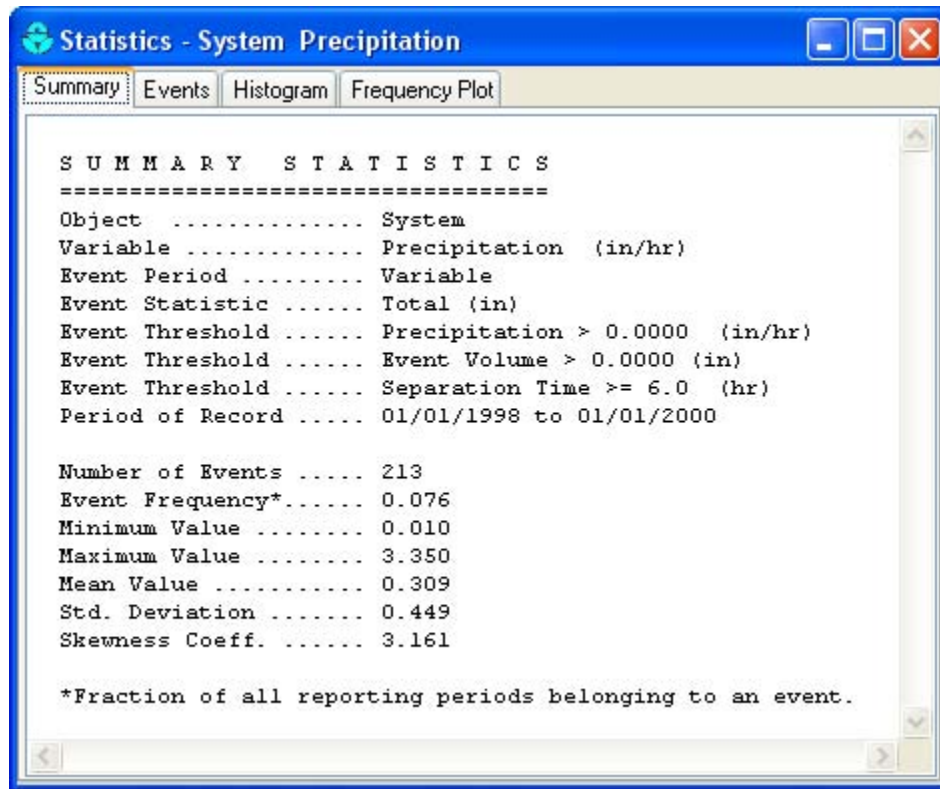
After our continuous simulation is completed we can perform a statistical frequency analysis on any of the variables produced as output. For example, to determine the distribution of rainfall volumes within each storm event over the two-year period simulated:

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 2-22.
3. Click the **OK** button to close the form.



**Figure 2-22. Statistics Selection dialog.**

The results of this request will be a Statistics Report form (see Figure 2-23) containing four tabbed pages: a Summary page, an Events page containing a rank-ordered listing of each event, a Histogram page containing a plot of the occurrence frequency versus event magnitude, and a Frequency Plot page that plots event magnitude versus cumulative frequency.



**Figure 2-23. Statistical Analysis report.**

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.

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

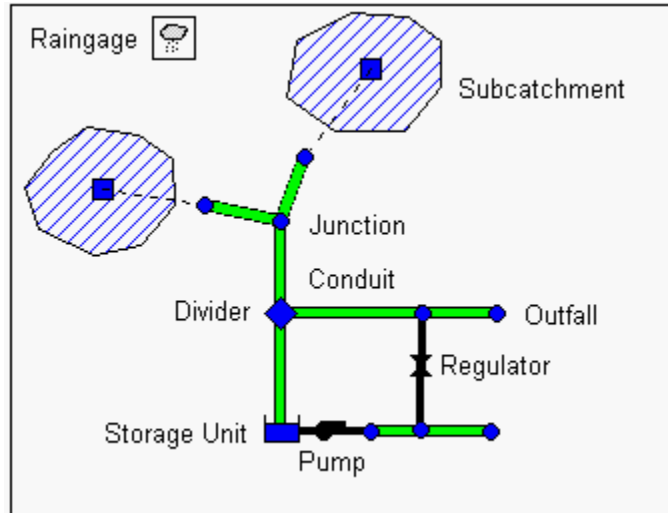
SWMM conceptualizes a drainage system as a series of water and material flows between several major environmental compartments. These compartments and the SWMM objects they contain include:

- 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.



**Figure 3-1. Example of physical objects used to model a drainage system.**

### 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

$$Q_{div} = C_w (fH_w)^{1.5}$$

where  $Q_{div}$  = diverted flow,  $C_w$  = weir coefficient,  $H_w$  = weir height and  $f$  is computed as

$$f = \frac{Q_{in} - Q_{min}}{Q_{max} - Q_{min}}$$

where  $Q_{in}$  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 catch basin 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:

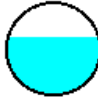
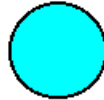







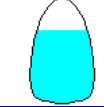





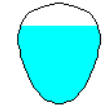

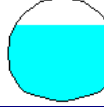





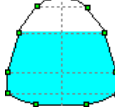
- invert elevation
- maximum depth
- depth-surface area data
- evaporation potential
- ponded surface area when flooded (optional)
- external inflow data (optional).

### 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.

Most open channels can be represented with a rectangular, trapezoidal, or user-defined irregular cross-section shape. For the latter, a Transect object is used to define how depth varies with distance across the cross-section (see Section 3.3.5 below). The most common shapes for new drainage and sewer pipes are circular, elliptical, and arch pipes. They come in standard sizes that are published by the American Iron and Steel Institute in *Modern Sewer Design* and by the American Concrete Pipe Association in the *Concrete Pipe Design Manual*. The Filled Circular shape allows the bottom of a circular pipe to be filled with sediment and thus limit its flow capacity. The Custom Closed Shape allows any closed geometrical shape that is symmetrical about the center line to be defined by supplying a Shape Curve for the cross section (see Section 3.3.11 below).

**Table 3-1. Available cross section shapes for conduits**

Name	Parameters	Shape	Name	Parameters	Shape
Circular	Full Height		Circular Force Main	Full Height, Roughness	
Filled Circular	Full Height, Filled Depth		Rectangular - Closed	Full Height, Width	
Rectangular – Open	Full Height, Width		Trapezoidal	Full Height, Base Width, Side Slopes	
Triangular	Full Height, Top Width		Horizontal Ellipse	Full Height, Max. Width	
Vertical Ellipse	Full Height, Max. Width		Arch	Full Height, Max. Width	
Parabolic	Full Height, Top Width		Power	Full Height, Top Width, Exponent	
Rectangular-Triangular	Full Height, Top Width, Triangle Height		Rectangular-Round	Full Height, Top Width, Bottom Radius	
Modified Baskethandle	Full Height, Bottom Width, Top Radius		Egg	Full Height	
Horseshoe	Full Height		Gothic	Full Height	
Catenary	Full Height		Semi-Elliptical	Full Height	
Baskethandle	Full Height		Semi-Circular	Full Height	
Irregular Natural Channel	Transect Coordinates		Custom Closed Shape	Full Height, Shape Curve Coordinates	



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

$$Q = \frac{1.49}{n} AR^{2/3} S^{1/2}$$

where  $n$  is the Manning roughness coefficient. The slope  $S$  is interpreted as either the conduit slope or the friction slope (i.e., head loss per unit length), depending on the flow routing method used.

For pipes with Circular Force Main cross-sections either the Hazen-Williams or Darcy-Weisbach formula is used in place of the Manning equation for fully pressurized flow. For U.S. units the Hazen-Williams formula is:

$$Q = 1.318C AR^{0.63} S^{0.54}$$

where  $C$  is the Hazen-Williams C-factor which varies inversely with surface roughness and is supplied as one of the cross-section's parameters. The Darcy-Weisbach formula is:

$$Q = \sqrt{\frac{8g}{f}} AR^{1/2} S^{1/2}$$

where  $g$  is the acceleration of gravity and  $f$  is the Darcy-Weisbach friction factor. For turbulent flow, the latter is determined from the height of the roughness elements on the walls of the pipe (supplied as an input parameter) and the flow's Reynolds Number using the Colebrook-White equation. The choice of which equation to use is a user-supplied option.



A conduit does not have to be assigned a Force Main shape for it to pressurize. Any of the closed cross-section shapes can potentially pressurize and thus function as force mains that use the Manning equation to compute friction losses.

A conduit can also be designated to act as a culvert if a Culvert Inlet Geometry code number is assigned to it. These code numbers are listed in Table A10 of Appendix A. Culvert conduits are checked continuously during dynamic wave flow routing to see if they operate under Inlet Control as defined in the Federal Highway Administration's publication *Hydraulic Design of Highway Culverts* (Publication No. FHWA-NHI-01-020, May 2005). Under inlet control a culvert obeys a particular flow versus inlet depth rating curve whose shape depends on the culvert's shape, size, slope, and inlet geometry.

The principal input parameters for conduits are:

- names of the inlet and outlet nodes
- offset height or elevation above the inlet and outlet node inverts
- conduit length
- Manning's roughness
- cross-sectional geometry

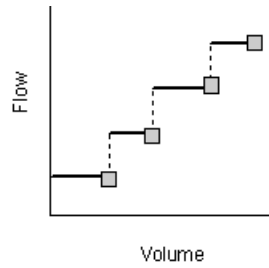
- entrance/exit losses (optional)
- presence of a flap gate to prevent reverse flow (optional).

### 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:

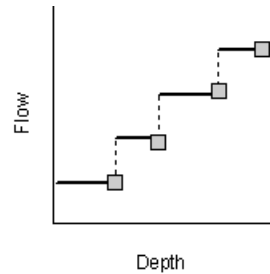
#### Type1

An off-line pump with a wet well where flow increases incrementally with available wet well volume



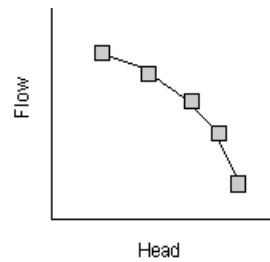
#### Type2

An in-line pump where flow increases incrementally with inlet node depth.



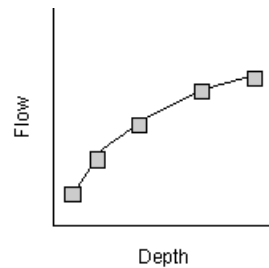
#### Type3

An in-line pump where flow varies continuously with head difference between the inlet and outlet nodes.



#### Type4

A variable speed in-line pump where flow varies continuously with inlet node depth.



#### Ideal

An "ideal" transfer pump whose flow rate equals the inflow rate at its inlet node. No curve is required. The pump must be the only outflow link from its inlet node. Used mainly for preliminary design.

The on/off status of pumps can be controlled dynamically by specifying startup and shutoff water depths at the inlet node or through user-defined Control Rules. Rules can also be used to simulate variable speed drives that modulate pump flow.

The principal input parameters for a pump include:

- names of its inlet and outlet nodes
- name of its pump curve
- initial on/off status
- startup and shutoff depths.

### 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, and 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 height of an orifice's opening can be controlled dynamically through user-defined Control Rules. This feature can be used to model gate openings and closings.

The principal input parameters for an orifice include:

- names of its inlet and outlet nodes
- configuration (bottom or side)
- shape (circular or rectangular)
- height or elevation above the inlet node invert
- discharge coefficient

- time to open or close.

### 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.**

Weir Type	Cross Section Shape	Flow Formula
Transverse	Rectangular	$C_w L h^{3/2}$
Side flow	Rectangular	$C_w L h^{5/3}$
V-notch	Triangular	$C_w S h^{5/2}$
Trapezoidal	Trapezoidal	$C_w L h^{3/2} + C_{ws} S h^{5/2}$
$C_w$ = 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 or elevation 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 rating curve determines an outlet's discharge flow as a function of either the freeboard depth above the outlet's opening or the head difference across it. Control Rules can be used to dynamically adjust this flow when certain conditions exist.

The principal input parameters for an outlet include:

- names of its inlet and outlet nodes
- height or elevation above the inlet node invert
- function or table containing its head (or depth) - 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.

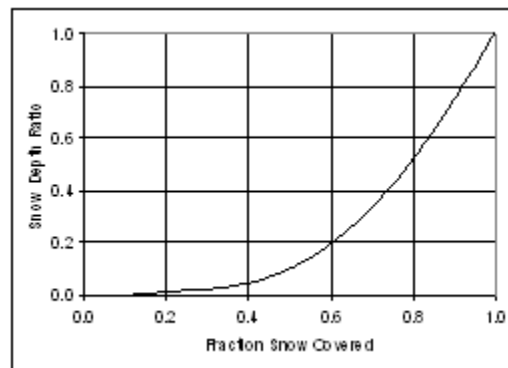
### Snowmelt

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.**

### 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 un-saturated 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.

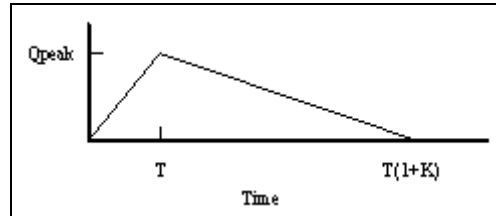
### 3.3.4 Unit Hydrographs

Unit Hydrographs (UHs) estimate rainfall-dependent 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**

Each unit hydrograph can also have a set of Initial Abstraction (IA) parameters associated with it. These determine how much rainfall is lost to interception and depression storage before any excess rainfall is generated and transformed into RDII flow by the hydrograph. The IA parameters consist of:

- a maximum possible depth of IA (inches or mm),
- a recovery rate (inches/day or mm/day) at which stored IA is depleted during dry periods,
- an initial depth of stored IA (inches or mm).

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.

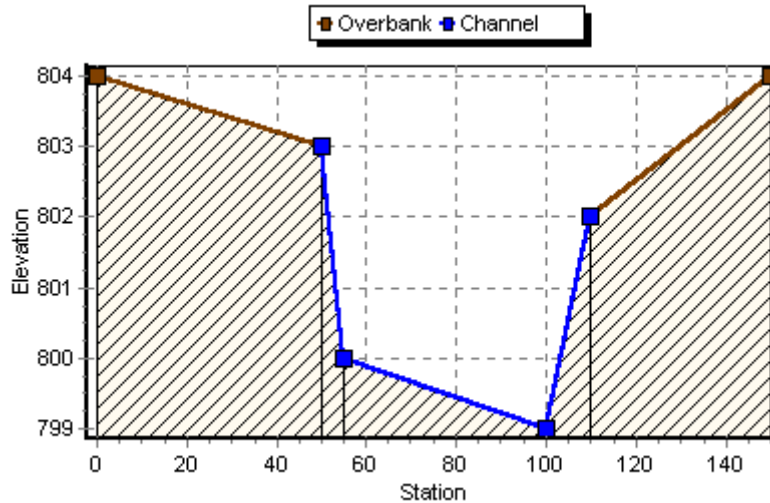


An alternative to using unit hydrographs to define RDII flow is to create an external RDII interface file, which contains RDII time series data.

### 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 transect for a natural channel.





**Figure 3-4. Example of a natural channel transect.**

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-Dependent 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 11.7 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
```

### Modulated weir height control:

```
RULE R4  
IF NODE N2 DEPTH >= 0  
THEN WEIR W25 SETTING = CURVE C25
```

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 units. The buildup here is the total mass (not per area or curb length) and both buildup and 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<sup>2</sup> or m<sup>2</sup>)
  - **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:

$$C = BOD * exp(-0.05 * HRT)$$

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

$$R = 0.75 * R\_TSS$$

### 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.
- **Shape** - describes how the width of a customized cross-sectional shape varies with height for a Conduit link.
- **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.
- **Control** - determines how the control setting of a pump or flow regulator varies as a function of some control variable (such as water level at a particular node) as specified in a Modulated Control rule.

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
- control settings for pumps and flow regulators..

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. Time series data can either be entered directly into the program or be accessed from a user-supplied Time Series file.



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
- Groundwater
- Flow Routing
- Water Quality Routing
- Infiltration
- Snowmelt
- Surface Ponding

### 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,  $d_p$ , 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.

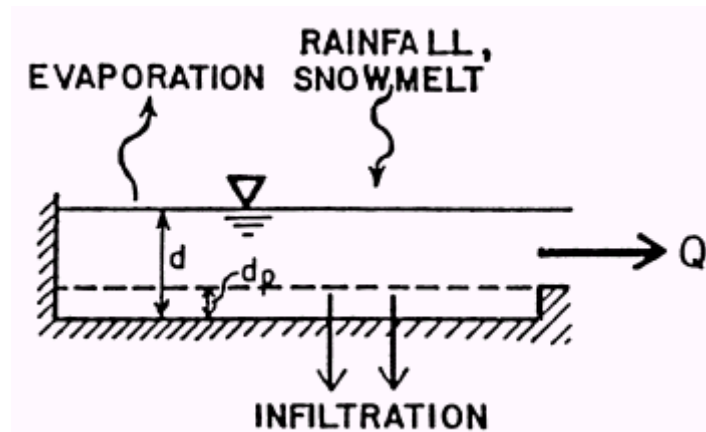


Figure 3-5. Conceptual view of surface runoff.

### 3.4.2 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

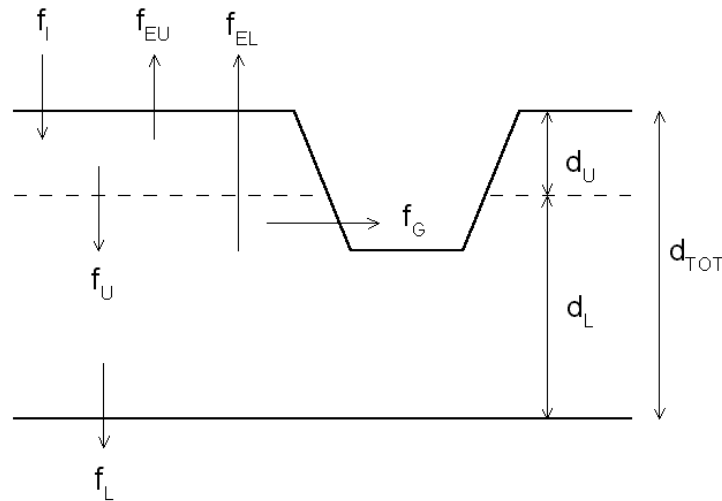


remaining capacity. The input parameters for this method are the curve number and the time it takes a fully saturated soil to completely dry.

SWMM also allows the infiltration recovery rate to be adjusted by a fixed amount on a monthly basis to account for seasonal variation in such factors as evaporation rates and groundwater levels. This optional monthly soil recovery pattern is specified as part of a project's Evaporation data.

### 3.4.3 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:



**Figure 3-6. Two-zone groundwater model.**

$f_I$  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.

#### 3.4.4 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. a heat budget equation for periods with rainfall, where melt rate increases with increasing air temperature, wind speed, and rainfall intensity
  - b. 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.

#### 3.4.5 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 normal flow 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 normal flow 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 normal flow 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).

Each of these routing methods employs the Manning equation to relate flow rate to flow depth and bed (or friction) slope. The one exception is for circular Force Main shapes under pressurized flow, where either the Hazen-Williams or Darcy-Weisbach equation is used instead.

### 3.4.6 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.

### 3.4.7 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.

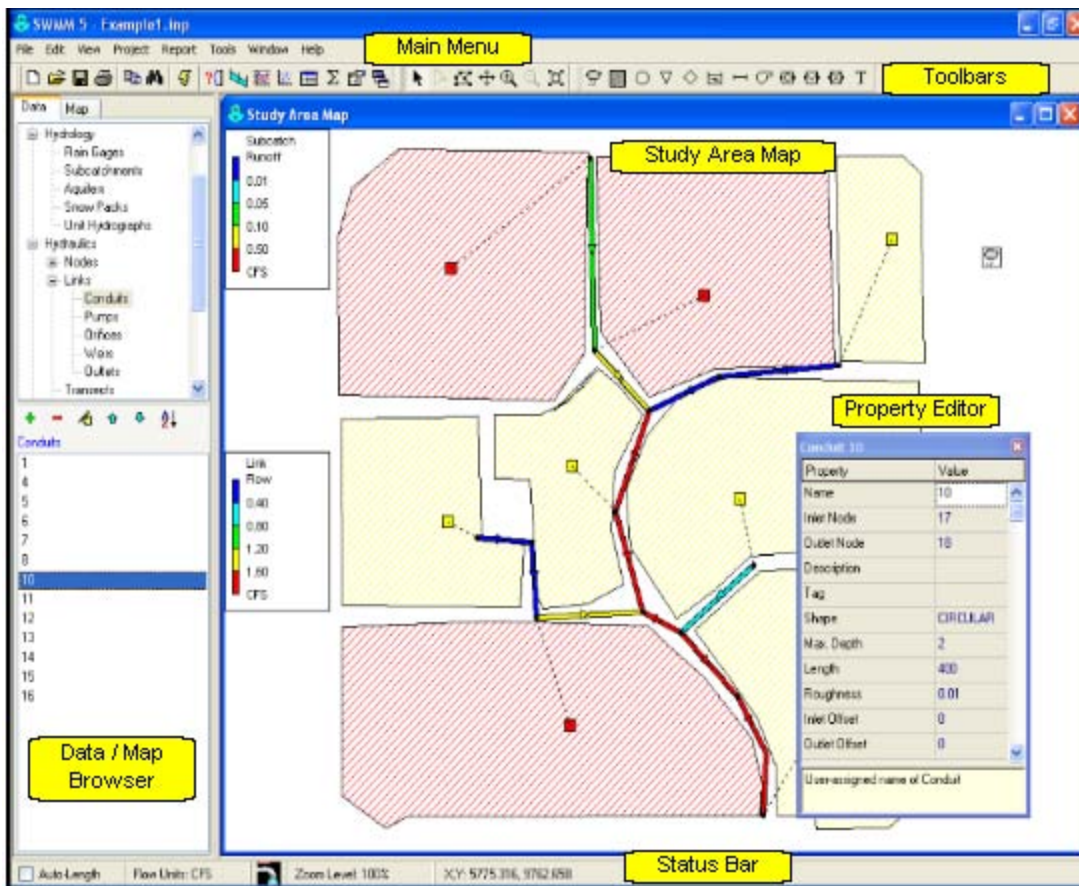
Water quality modeling within storage unit nodes follows the same approach used for conduits. For other types of nodes that have no volume, the quality of water exiting the node is simply the mixture concentration of all water entering the node.

## CHAPTER 4 - SWMM'S MAIN WINDOW

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
- Tools Menu
- Window Menu
- Help Menu

### File Menu

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

<i>Command</i>	<i>Description</i>
New	Creates a new SWMM project
Open	Opens an existing project
Reopen	Reopens a recently used project
Save	Saves the current project
Save As	Saves the current project under a different name
Export	Exports study area map to a file in a variety of formats; Exports current results to a Hot Start file
Combine	Combines two Routing Interface files together
Page Setup	Sets page margins and orientation for printing
Print Preview	Previews a printout of the currently active view (map, report, graph, or table)
Print	Prints the current view
Exit	Exits EPA SWMM

## Edit Menu

The Edit Menu contains commands for editing and copying:

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

## View Menu

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

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

## Project Menu

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

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

## Report Menu

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

<i>Command</i>	<i>Description</i>
Status	Displays a status report for the most recent simulation run
Graph	Displays simulation results in graphical form
Table	Displays simulation results in tabular form
Statistics	Displays a statistical analysis of simulation results
Customize	Customizes the display style of the currently active graph

## Tools Menu

The Tools menu contains commands used to configure program preferences, study area map display options, and external add-in tools:

<i>Command</i>	<i>Description</i>
Program Preferences	Sets program preferences, such as font size, confirm deletions, number of decimal places displayed, etc.
Map Display Options	Sets appearance options for the Map, such as object size, annotation, flow direction arrows, and back-ground color
Configure Tools	Adds, deletes, or modifies add-in tools



## Window Menu

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

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

## Help Menu

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

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

### 4.3 Toolbars

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
















- Standard Toolbar
- Map Toolbar
- Object 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 object on the Study Area Map (Edit >> Find Object) or specific text in the Status Report (Edit >> Find Text)
-  Runs a simulation (Project >> Run Simulation)
-  Makes a visual query of the study area map (View >> Query)
-  Creates a new profile plot of simulation results (Report >> Graph >> Profile)
-  Creates a new time series plot of simulation results (Report >> Graph >> Time Series)
-  Creates a new scatter plot of simulation results (Report >> Graph >> Scatter)
-  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 (Tools >> Map Display Options or Report >> Customize)
-  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**)
-  Measures a length or area on the map

## 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

## 4.4 Status Bar

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



### **Auto-Length**

Indicates whether the automatic computation of conduit lengths and subcatchment areas is turned on or off. The setting can be changed by clicking the drop down arrow.

### **Offsets**

Indicates whether the positions of links above the invert of their connecting nodes are expressed as a Depth above the node invert or as the Elevation of the offset. Click the drop down arrow to change this option. If changed, a dialog box will appear asking if all existing offsets in the current project should be changed or not (i.e., convert Depth offsets to Elevation offsets or Elevation offsets to Depth offsets, depending on the option selected)

### **Flow Units**

Displays the current flow units that are in effect. Click the drop down arrow to change the 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 units of previously entered data are not automatically adjusted if the unit system is changed.

### 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

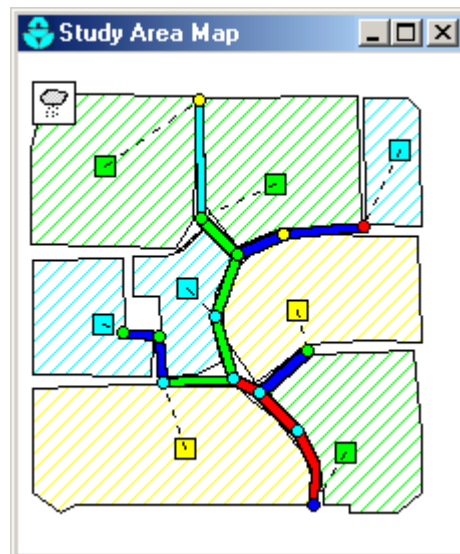
Displays the current zoom level for the map (100% is full-scale).

### 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:

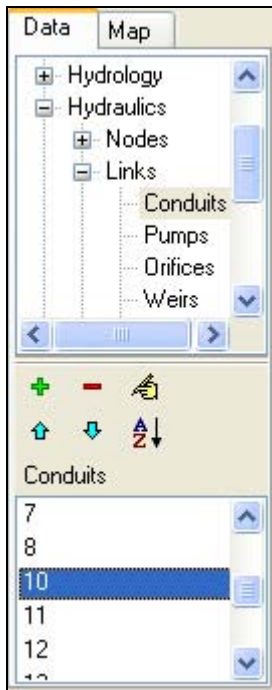


- The location of objects and the distances between them do not necessarily have to conform to their actual physical scale.
- Selected properties of these objects, such as water quality at nodes or flow velocity in links, can be displayed by using different colors. The color-coding is described in a 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 workspace is selected. It provides access to all of the data objects in a project. The vertical sizes of the list boxes in the browser can be adjusted by using the splitter bar located just below the upper list box. The width of the Data Browser panel can be adjusted by using the splitter bar located along its right edge.



The upper 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 workspace is selected. It controls the mapping themes and time periods viewed on the Study Area Map. The width of the Map Browser panel can be adjusted by using the splitter bar located along its right edge.



The Map Browser consists of the following three panels that control what results are displayed on the map:

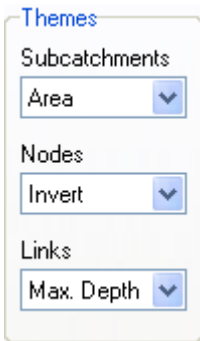
The Themes panel selects a set of variables to view in color-coded fashion on the Map.

The Time Period panel selects which time period of the simulation results are viewed on the Map.

The Animator panel controls the animated display of the Study Area Map and all Profile Plots over time.

The width of the Map Browser panel can be adjusted by using the splitter bar located along its right edge.

The Themes panel of the Map Browser is used to select a thematic variable to view in color-coded fashion on the Study Area Map.

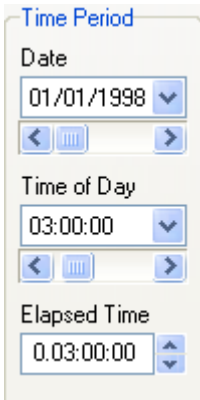


Subcatchments - selects the theme to display for the subcatchment areas shown on the Map.

Nodes - selects the theme to display for the drainage system nodes shown on the Map.

Links - selects the theme to display for the drainage system links shown on the Map.

The Time Period panel of the Map Browser allows is used to select a time period in which to view computed results in thematic fashion on the Study Area Map.

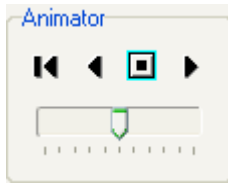


Date - selects the day for which simulation results will be viewed.

Time of Day - 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.

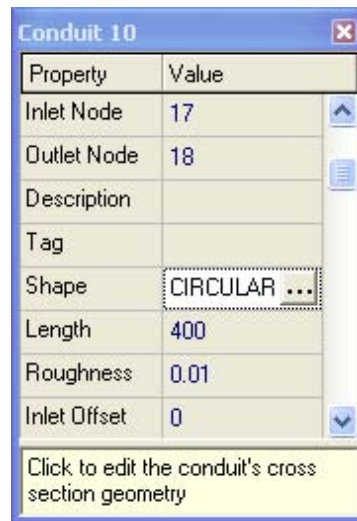
The Animator panel of the Map Browser contains controls for animating the Study Area Map and all Profile Plots through time i.e., updating map color-coding and hydraulic grade line profile depths as the simulation time clock is automatically moved forward or back. The meaning of the control buttons are as follows:



- ◀ Returns to the starting period.
  - ◀ Starts animating backwards in time
  - ◻ Stops the animation
  - ▶ Starts animating forwards in time
- The slider bar is used to adjust the animation speed.

## 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.



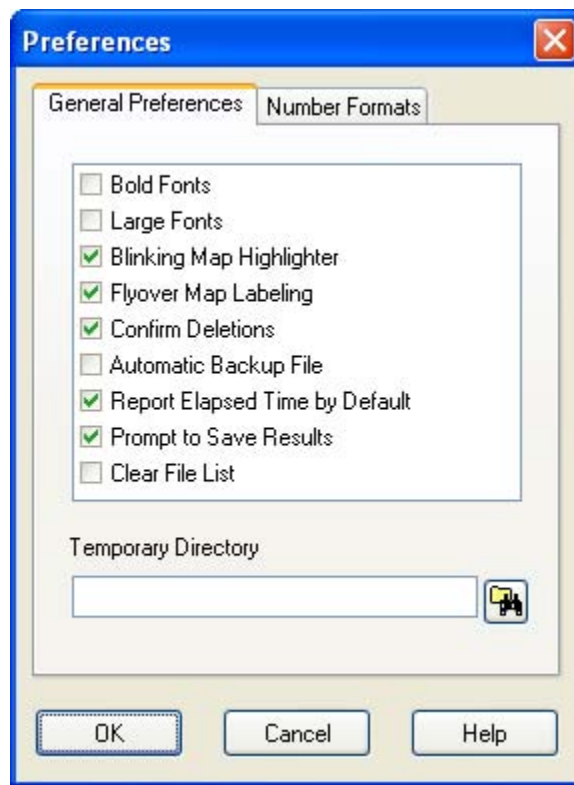
Key features of the Property Editor include:

- 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.
- 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 Program Preferences from the Tools 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:

<i>Preference</i>	<i>Description</i>
Bold Fonts	Check to use bold fonts in all windows
Large Fonts	Check to use large size fonts in all windows
Blinking Map Highlighter	Check to make the selected object on the study area map blink on and off
Flyover Map Labeling	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
Confirm Deletions	Check to display a confirmation dialog box before deleting any object
Automatic Backup File	Check to save a backup copy of a newly opened project to disk named with a .bak extension
Report Elapsed Time by Default	Check to use elapsed time (rather than date/time) as the default for time series graphs and tables.
Prompt to Save Results	If left unchecked then simulation results are automatically saved to disk when the current project is closed. Otherwise the user will be asked if results should be saved.
Clear File List	Check to clear the list of most recently used files that appears when File >> Reopen is selected from the Main Menu
Temporary Directory	Name of the directory (folder) where EPA SWMM writes its temporary files



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.

(This page intentionally left blank.)


## 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:

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

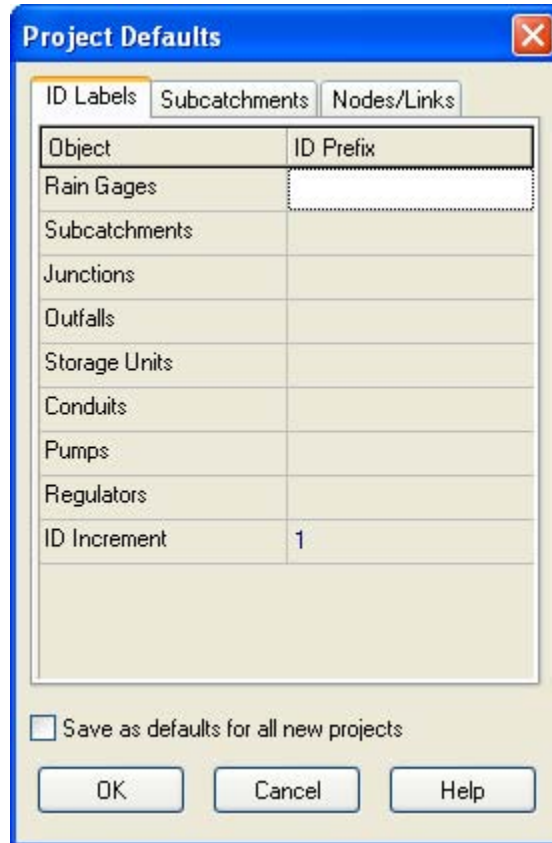
To set default values for a project:

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 left 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 Roughness
- Pervious Area Roughness
- 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
- Node Pondered Area
- Conduit Length
- Conduit Shape and Size
- Conduit Roughness
- Flow Units
- Link Offsets Convention
- Routing Method
- Force Main Equation

The defaults automatically assigned to individual objects can be changed by using the object's Property Editor. The choice of Flow Units and Link Offsets Convention can be changed directly on the main window's Status Bar.

## 5.5 Measurement Units

SWMM can use either US units or SI metric units. The choice of flow units determines what unit system is used for all other quantities:

- selecting CFS (cubic feet per second), GPM (gallons per minutes), or MGD (million gallons per day) for flow units implies that US units will be used throughout
- selecting CMS (cubic meters per second), LPS (liters per second), or MLD (million liters per day) as flow units implies that SI units will be used throughout.

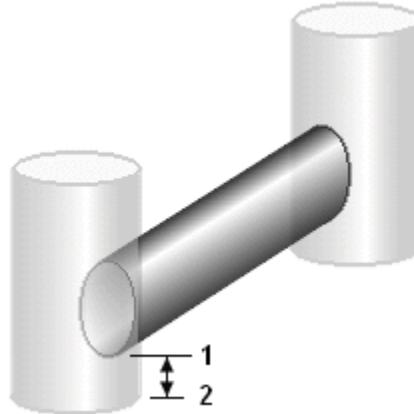
Flow units can be selected directly on the main window's Status Bar or by setting a project's default values. In the latter case the selection can be saved so that all new future projects will automatically use those units.



The units of previously entered data are not automatically adjusted if the unit system is changed.

## 5.6 Link Offset Conventions

Conduits and flow regulators (orifices, weirs, and outlets) can be offset some distance above the invert of their connecting end nodes as depicted below:



There are two different conventions available for specifying the location of these offsets. The Depth convention uses the offset distance from the node's invert (distance between ① and ②, in the figure above). The Elevation convention uses the absolute elevation of the offset location (the elevation of point ① in the figure). The choice of convention can be made on the Status Bar of SWMM's main window or on the Node/Link Properties page of the Project Defaults dialog. When this convention is changed, a dialog will appear giving one the option to automatically recalculate all existing link offsets in the current project using the newly selected convention

## 5.7 Calibration Data

SWMM can compare the results of a simulation with measured field data in its Time Series Plots, which are discussed in section 9.4. 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
- Groundwater Flow
- Groundwater Elevation
- Snow Pack Depth
- Node Depth
- Node Lateral Inflow

- Node Flooding
- Node Water Quality
- Link Flow Rate

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.

Calibration Variable	Name of Calibration File
Subcatchment Runoff	
Subcatchment Washoff	
Node Water Depth	
Link Flow Rate	C:\My Projects\SWMM5\QA_Report\Extran\extran1.dat
Node Water Quality	
Node Lateral Inflow	
Node Flooding	
Groundwater Flow	
Groundwater Elevation	
Snow Pack Depth	
Link Flow Depth	
Link Flow Velocity	

Buttons: Browse, Edit, OK, Cancel, Help



## 5.8 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.

(This page intentionally left blank.)

## 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	Links
Simulation Options	Transects
Climatology	Controls
Rain Gages	Pollutants
Subcatchments	Land Uses
Aquifers	Curves
Snow Packs	Time Series
Unit Hydrographs	Time Patterns
Nodes	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:
3. left-click at each vertex
4. right-click or press **<Enter>** to close the polygon
5. 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 <Enter> 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 either be 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 directly from the main window's Status Bar (see Section 4.4). 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 either by 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.

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. 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, or right-click the object on the map and select **Delete** from the pop-up menu that appears.



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, Infiltration, Junctions, Storage Units, 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. (For Infiltration, the Tag will be that of the subcatchment to which the infiltration parameters belong.)
3. Enter a Tag value to filter on if you have selected that option.
4. Select the property to edit.
5. Select whether to replace, multiply, or add to the existing value of the property. Note that for some non-numerical properties the only available choice is to replace the value.

6. In the lower-right edit box, enter the value that should replace, multiply, or be added to the existing value for all selected objects. Some properties will have an ellipsis button displayed in the edit box which should be clicked to bring up a specialized editor for the property.
7. 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. As an option, you can specify that only objects within the selected area that have a specific Tag property should be deleted. Keep in mind that deleting a node will also delete any links connected to the node.

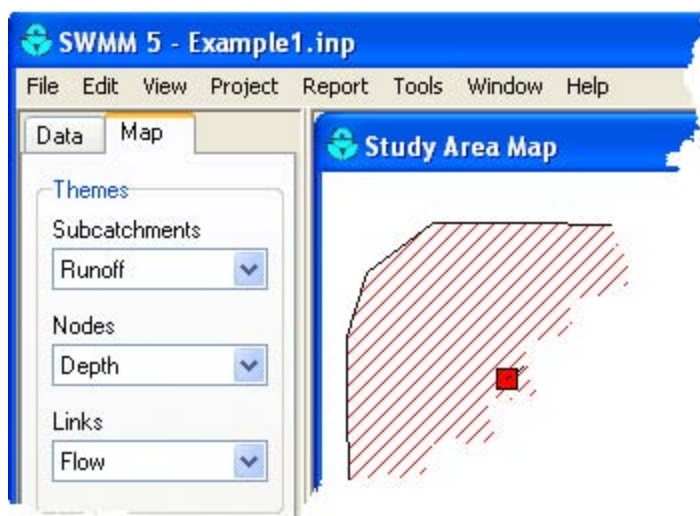
## CHAPTER 7 - WORKING WITH THE MAP

---

*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.



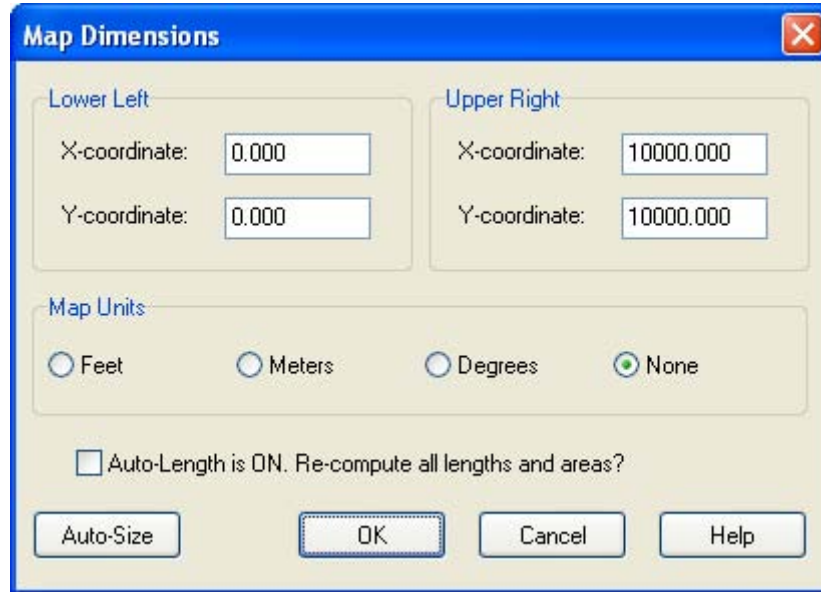
Methods for changing the color-coding associated with a theme are discussed in Section 7.9 below.

### 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. If the **Auto-Length** option is in effect, check the “Re-compute all lengths and areas” box if you would like SWMM to re-calculate all conduit lengths and subcatchment areas under the new set of map dimensions.
5. 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.

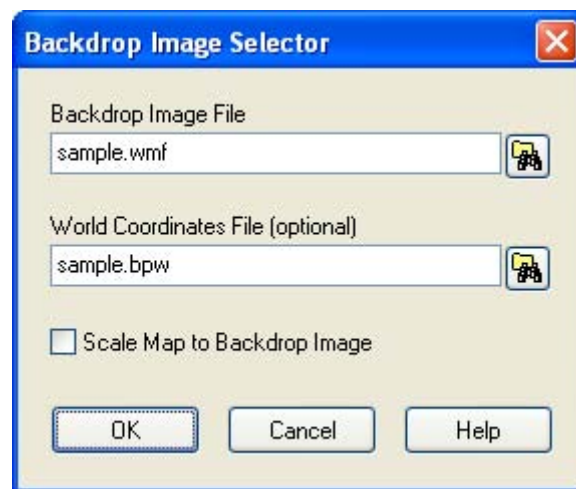


The backdrop image must be a Windows metafile, bitmap, or JPEG image created outside of SWMM. Once imported, its features cannot be edited, although its scale and viewing area will change as the map window is zoomed and panned. For this reason metafiles work better than bitmaps or JPEGs since they will not lose 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**

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.

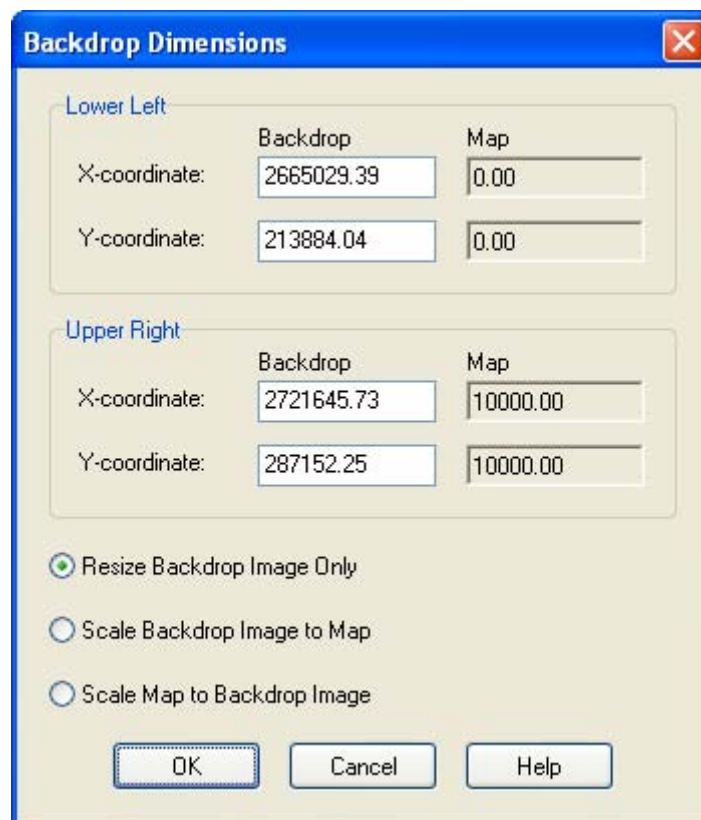
If no world file is specified, then the backdrop will be scaled to fit into the center of the map display window.

### **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.



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 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 Measuring Distances

To measure a distance or area on the Study Area Map:


1. Click  on the Map Toolbar.
2. Left-click on the map where you wish to begin measuring from.
3. Move the mouse over the distance being measured, left-clicking at each intermediate location where the measured path changes direction.
4. Right-click the mouse or press **<Enter>** to complete the measurement.
5. The distance measured in project units (feet or meters) will be displayed in a dialog box. If the last point on the measured path coincides with the first point then the area of the enclosed polygon will also be displayed.

## 7.5 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. The map will be returned to the view in effect at the previous zoom level.

## 7.6 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.

## 7.7 Viewing at Full Extent


To view the Study Area Map at full extent, either:

- select **View >> Full Extent** from the Main Menu, or
- press  on the Map Toolbar.



## 7.8 Finding an Object

To find an object on the Study Area Map whose name is known:

1. Select **View >> Find Object** 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*.

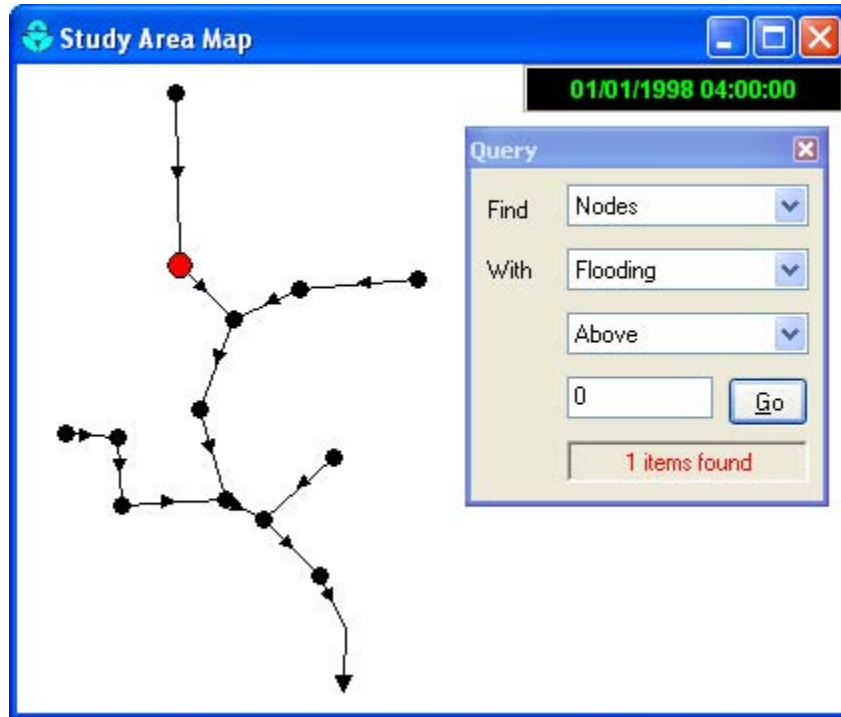
After an object is found, the Map Finder dialog will also list:

- the outlet connections for a subcatchment
- the connecting links for a node
- the connecting nodes for a link.

## 7.9 Submitting a Map Query

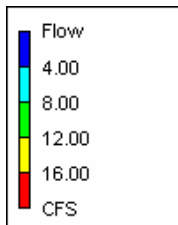
A Map Query identifies objects on the study area map that meet a specific criterion (e.g., nodes which flood, links with velocity below 2 ft/sec, etc.). To submit a map query:

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.

## 7.10 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.

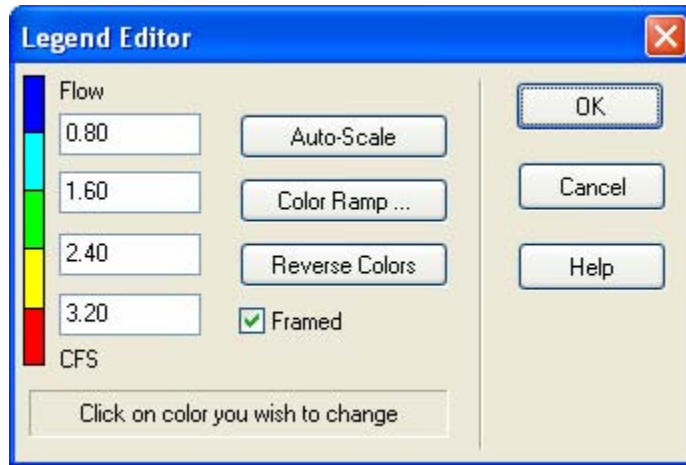
To display or hide a map legend:

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.

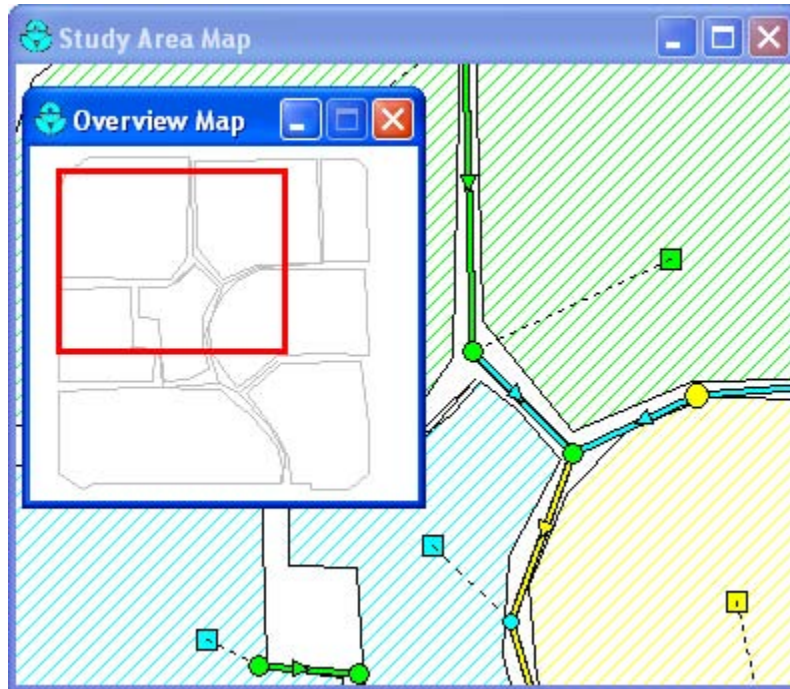


The Legend Editor is used to set numerical ranges to which different colors are assigned for viewing a particular parameter on the network map. It works as follows:

- 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.

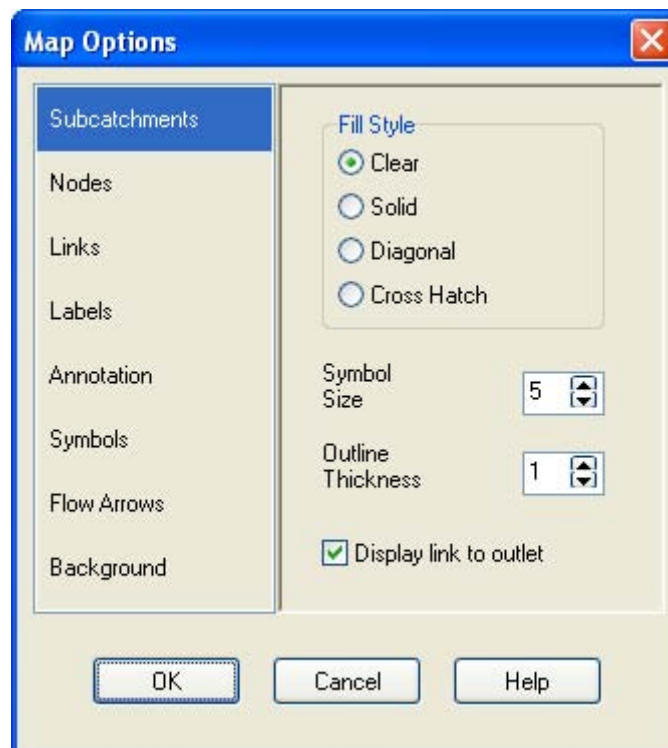
### 7.11 Using the Overview Map


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.



## 7.12 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 **Tools >> Map Display 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.

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

## Node Options

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

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

## Link Options

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

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

## Label Options

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

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

## 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.

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

## Symbol Options

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

<i>Option</i>	<i>Description</i>
Display Node Symbols	If checked then special node symbols will be used
Display Link Symbols	If checked then special link symbols will be used
At Zoom Of	Selects minimum zoom at which symbols should be displayed; symbols will be hidden at zooms smaller than this

## Flow Arrow Options

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

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



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.

## 7.13 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.





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.

After choosing a format, click **OK** and enter a name for the file in the Save As dialog that appears.

(This page intentionally left blank.)

## 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.
2. Select one of the following categories of options to edit:
  - a. General Options
  - b. Date Options
  - c. Time Step Options
  - d. Dynamic Wave Routing Options
  - e. Interface File Options

and click the  button to invoke the Simulation Options dialog.

The Simulations Options dialog contains a separate tabbed page for each of these option categories which are described in more detail below.

#### General Options

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

##### **Process Models**

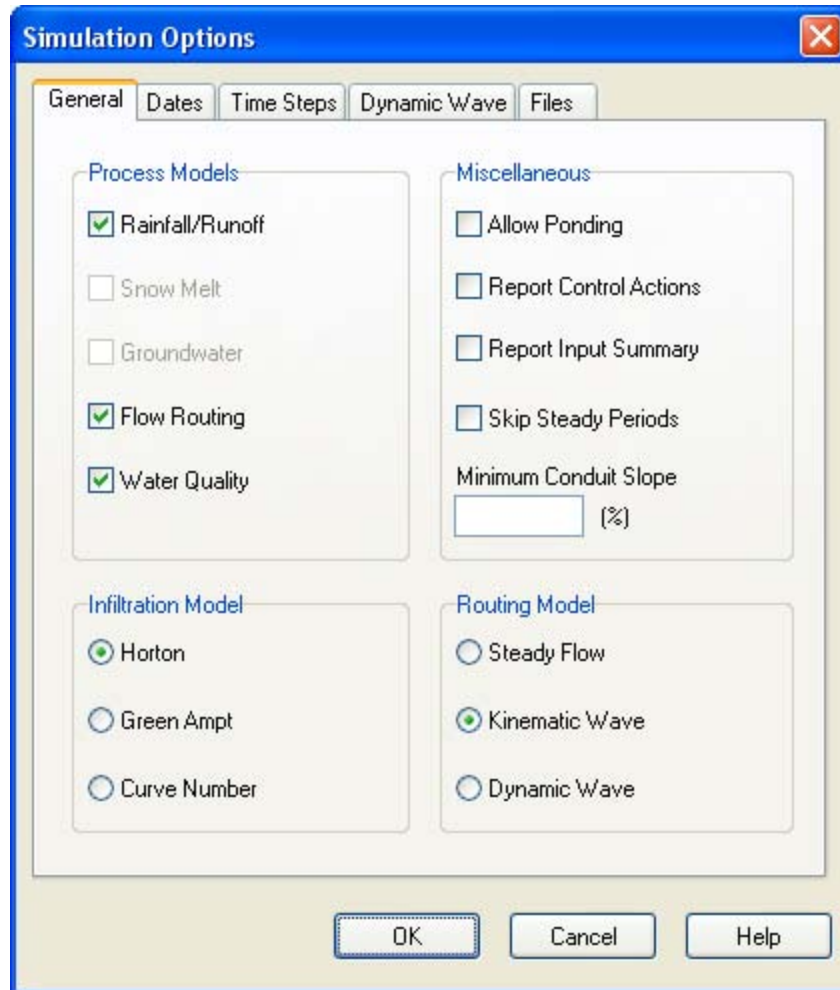
This section allows you to select which of SWMM's process models will be applied to the current project. For example, a model that contained Aquifer and Groundwater elements could be run first with the groundwater computations turned on and then again with them turned off to see what effect this process had on the site's hydrology. Note that if there are no elements in the project needed to model a given process then that process option is disabled (e.g., if there were no Aquifers defined for the project then the Groundwater check box will appear disabled in an unchecked state).

##### **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

Each of these models is briefly described in section 3.4.2. 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

Review section 3.4.5 for a brief description of each of these alternatives.

### **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 Pondered Area attribute must be used.

### **Report Control Actions**

Check this option if you want the simulation's Status Report to list all discrete control actions taken by the Control Rules associated with a project (continuous modulated control actions are not listed). This option should only be used for short-term simulation.

### **Report Input Summary**

Check this option if you want the simulation's Status Report to list a summary of the project's input data.

### **Skip Steady Periods**

Checking this option will make the simulation use the most recently computed conveyance system flows during a steady state period instead of computing a new flow routing solution. A time step is considered to be in steady state if the change in external inflows at each node is below 0.5 cfs and the relative difference between total system inflow and outflow is below 5%.

### **Minimum Conduit Slope**

The minimum value allowed for a conduit's slope (%). If blank or zero (the default) then no minimum is imposed (although SWMM uses a lower limit on elevation drop of 0.001 ft (0.00035 m) when computing a conduit slope).

## 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 except for flow routing which is entered as decimal seconds.

### **Reporting Time Step**

Enter the time interval for reporting of computed results.

### **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 in decimal seconds 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.

## 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.

### **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.

### **Define Supercritical Flow By**

Selects the basis used to determine when supercritical flow occurs in a conduit. The choices are:

- water surface slope only (i.e., water surface slope > conduit slope)
- Froude number only (i.e., Froude number > 1.0)
- both water surface slope and Froude number.

The first two choices were used in earlier versions of SWMM while the third choice, which checks for either condition, is now the recommended one.

### **Force Main Equation**

Selects which equation will be used to compute friction losses during pressurized flow for conduits that have been assigned a Circular Force Main cross-section. The choices are either the Hazen-Williams equation or the Darcy-Weisbach equation.

### **Use Variable Time Step**

Check the box if an internally computed variable time step should be used at each routing time period and select an adjustment (or safety) factor to apply to this time step. The variable time step is computed so as to satisfy the Courant condition within each conduit. A typical adjustment factor would be 75% to provide some margin of conservatism. The computed variable time step will not be less than 0.5 seconds nor be greater than the fixed time step specified on the Time Steps page of the dialog. If the latter was set lower than 0.5 seconds then the variable time step option is ignored.

### **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. The ratio of the artificial length to the original length for each conduit is listed in the Flow Classification table that appears in the simulation's Status Report (see Section 9.1).

### **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 ft<sup>2</sup> (1.167 m<sup>2</sup>) is used. This is the area of a 4-ft diameter manhole. The value entered should be in square feet for US units or square meters for SI units.

## File Options

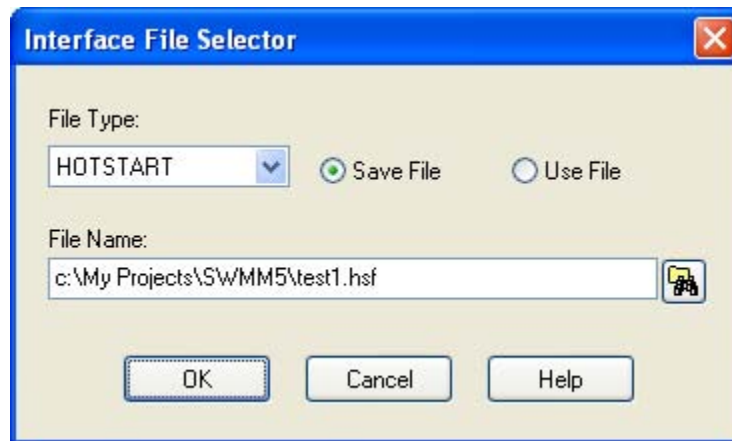
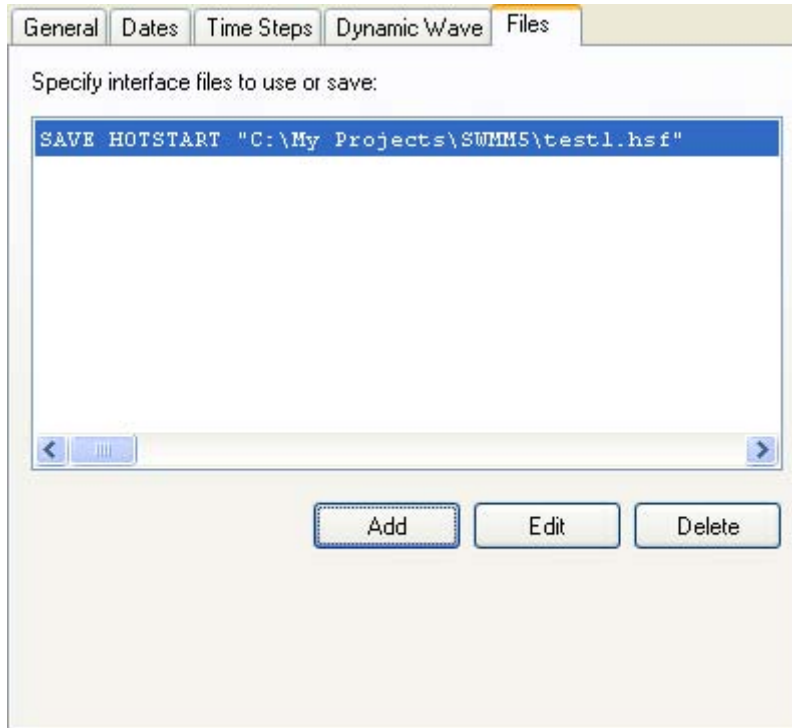
The Interface Files page of the Simulation Options dialog is used to specify which interface files will be used or saved during the simulation. (Interface files are described in Chapter 11.) The page contains a list box with three buttons underneath it. The list box lists the currently selected files,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

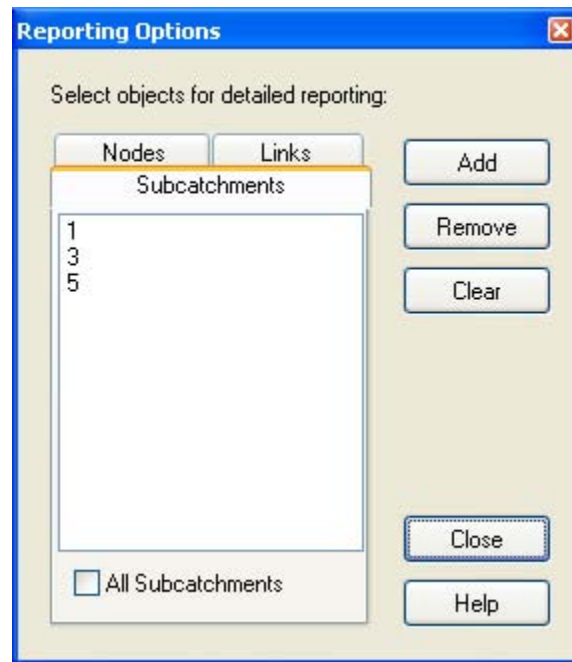
**File Name**

Enter the name of the interface file or click the Browse button  to select from a standard Windows file selection dialog box.



## 8.2 Setting Reporting Options

The Reporting Options dialog shown below is used to select individual subcatchments, nodes, and links that will have detailed time series results saved for viewing after a simulation has been run. The default for new projects is that all objects will have detailed results saved for them. The dialog is invoked by selecting the Reporting category of Options from the Data Browser and clicking the  button.



The dialog contains three tabbed pages - one each for subcatchments, nodes, and links. It is a stay-on-top form which means that you can select items directly from the Study Area Map or Data Browser while the dialog remains visible.

To include an object in the set that is reported on:

1. Select the tab to which the object belongs (**Subcatchments**, **Nodes** or **Links**).
2. Unselect the "All" check box if it is currently checked.
3. Select the specific object either from the **Study Area Map** or from the listing in the Data Browser.
4. Click the **Add** button on the dialog.
5. Repeat the above steps for any additional objects.

To remove an item from the set selected for reporting:


1. Select the desired item in the dialog's list box.
2. Click the **Remove** button to remove the item.

To remove all items from the reporting set of a given object category, select the object category's page and click the **Clear** button.

To include all objects of a given category in the reporting set, check the "**All**" box on the page for that category (i.e., subcatchments, nodes, or links). This will override any individual items that may be currently listed on the page.

To dismiss the dialog click the **Close** button.


### 8.3 Starting a Simulation

To start a simulation 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.

### 8.4 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. Even if a run completes successfully, one should check to insure that the results are reasonable. 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:

- 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 + total inflow and final storage + 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 reasons for an excessive continuity error are computational time steps that are too long or conduits that are too short.



In addition to the system continuity error, the Status Report produced by a run (see Section 9.1) will list those nodes of the drainage network that have the largest flow continuity errors. If the error for a node is excessive, then one should first consider if the node in question is of importance to the purpose of the simulation. If it is, then further study is warranted to determine how the error might be reduced.

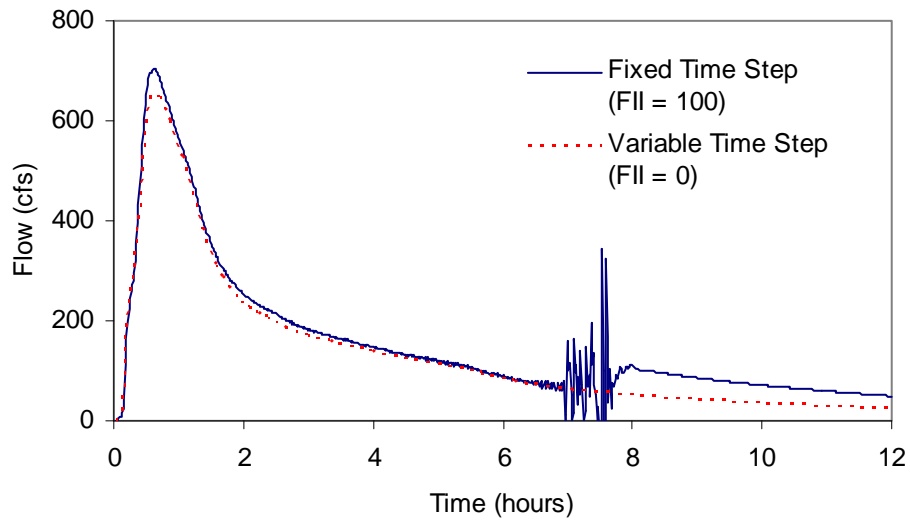
#### Unstable Flow Routing Results

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.

The run's Status Report lists the links having the five highest values of a Flow Instability Index (FII). This index counts the number of times that the flow value in a link is higher (or lower) than the flow in both the previous and subsequent time periods. The index is normalized with respect to the expected number of such 'turns' that would occur for a purely random series of values and can range from 0 to 150.

As an example of how the Flow Instability Index can be used, consider the figure shown below. The solid line plots the flow hydrograph for the link identified as having the highest FII value (100) in a dynamic wave flow routing run that used a fixed time step of 30 seconds. The dashed line shows the hydrograph that results when a variable time step was used instead, which is now completely stable.



Flow time series plots for the links having the highest FII's should be inspected to insure that flow routing results are acceptably stable.

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.

(This page intentionally left blank.)

## 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 summary of the main Simulation Options that are in effect
- a list of any error conditions encountered during the run
- a summary listing of the project's input data (if requested in the Simulation Options)
- a summary of the data read from each rainfall file used in the simulation
- a description of each control rule action taken during the simulation (if requested in the Simulation Options)
- the system-wide mass continuity errors for:
  - runoff quantity and quality
  - groundwater flow
  - conveyance system flow and water quality
- the names of the nodes with the highest individual flow continuity errors
- the names of the conduits that most often determined the size of the time step used for flow routing (only when the Variable Time Step option is used)
- the names of the links with the highest Flow Instability Index values
- information on the range of routing time steps taken and the percentage of these that were considered steady state.


In addition, the report contains several tables that display summary results for the quantities of most interest for each subcatchment, node, and link. The tables and the information they display are listed below.


Table	Columns
Subcatchment Runoff	Total precipitation (in or mm); Total run-on from other subcatchments (in or mm); Total evaporation (in or mm); Total infiltration (in or mm); Total runoff depth (in or mm); Total runoff volume (million gallons or million liters); Runoff coefficient (ratio of total runoff to total precipitation).
Subcatchment Washoff	Total mass of each pollutant washed off the subcatchment (lbs or kg).
Node Depths	Average water depth (ft or m); Maximum water depth (ft or m); Maximum hydraulic head (HGL) elevation (ft or m); Time of maximum depth.
Node Inflows	Maximum lateral inflow (flow units); Maximum total inflow (flow units); Time of maximum total inflow; Total lateral inflow volume (million gallons or million liters); Total inflow volume (million gallons or million liters).  <i>Note: Total inflow consists of lateral inflow plus inflow from connecting links.</i>
Node Surcharging	Hours surcharged; Maximum height of surcharge above node's crown (ft or m); Minimum depth of surcharge below node's top rim (ft or m).  <i>Note: surcharging occurs when water rises above the crown of the highest conduit and only those conduits that surcharge are listed.</i>
Node Flooding	Hours flooded; Maximum flooding rate (flow units); Time of maximum flooding; Total flood volume (million gallons or million liters); Maximum ponded volume (acre-in or ha-mm).  <i>Note: flooding refers to all water that overflows a node, whether it ponds or not, and only those nodes that flood are listed.</i>
Storage Volumes	Average volume of water in the facility (1000 ft <sup>3</sup> or 1000 m <sup>3</sup> ); Average percent of full storage capacity utilized; Maximum volume of water in the facility (1000 ft <sup>3</sup> or 1000 m <sup>3</sup> ); Maximum percent of full storage capacity utilized; Time of maximum water stored; Maximum outflow rate from the facility (flow units).
Outfall Loading	Percent of time that outfall discharges; Average discharge flow (flow units); Maximum discharge flow (flow units); Total volume of flow discharged (million gallons or million liters); Total mass discharged of each pollutant (lbs or kg).



Link Flows	Maximum flow (flow units); Time of maximum flow; Maximum velocity (ft/sec or m/sec) Ratio of maximum flow to full normal flow; Ratio of maximum flow depth to full depth.
Flow Classification	Ratio of adjusted conduit length to actual length; Fraction of time spent in the following flow categories: <ul style="list-style-type: none"> <li>▪ dry on both ends</li> <li>▪ dry on the upstream end</li> <li>▪ dry on the downstream end</li> <li>▪ subcritical flow</li> <li>▪ supercritical flow</li> <li>▪ critical flow at the upstream end</li> <li>▪ critical flow at the downstream end</li> </ul> Average Froude number; Average change in flow between each time step (flow units).
Conduit Surcharging	Hours that conduit is full at: <ul style="list-style-type: none"> <li>▪ both ends</li> <li>▪ upstream end</li> <li>▪ downstream end</li> </ul> Hours that conduit flows above full normal flow; Hours that conduit is capacity limited  <i>Note: only conduits with one or more non-zero entries are listed and a conduit is considered capacity limited if its upstream end is full and the HGL slope is greater than the conduit slope.</i>
Pumping Summary	Percent of time that the pump is on line; Maximum flow pumped (flow units); Average flow pumped (flow units); Total energy consumed assuming 100% efficiency (kwatt-hours); Percent of time that the pump operates off of its pump curve.

The Status Report can be viewed by selecting **Report >> Status** from the Main Menu. Its window includes a **Bookmarks** panel that makes it easy to navigate between the topics listed above.

To copy selected text from the Status Report to a file or to the Windows Clipboard, first select the text to copy with the mouse and then choose **Edit >> Copy To** from the Main Menu (or press the  button on the Standard Toolbar). If the entire report is to be copied then it is not necessary to first select text with the mouse.

To locate an object that is listed in one of the Status Report's tables, first select the object's name with the mouse and choose **Edit >> Find Object** from the Main Menu (or press the  button on the Standard Toolbar and select **Find Object** from the dropdown menu). Then in the Map Finder dialog that appears, select the type of object to look for (Subcatchment, Node or Link) and press the **Go** button (the object's name will have already been entered in the form). The object will appear highlighted in both the Data Browser and on the Study Area Map.

## 9.2 Variables That Can Be Viewed

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

### Subcatchment Variables

- rainfall rate (in/hr or mm/hr)
- snow depth (inches or millimeters)
- losses (infiltration + evaporation in in/hr or mm/hr)
- runoff flow (flow units)
- groundwater flow into the drainage network (flow units)
- groundwater elevation (ft or m)
- washoff concentration of each pollutant (mass/liter)

### Link Variables

- flow rate (flow units)
- average water depth (ft or m)
- flow velocity (ft/sec or m/sec)
- Froude number (dimensionless)
- capacity (ratio of depth to full depth)
- concentration of each pollutant (mass/liter)

### 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, ft<sup>3</sup> or m<sup>3</sup>)
- lateral inflow (runoff + all other external inflows, in flow units)
- total inflow (lateral inflow + upstream inflows, in flow units)
- surface flooding (flow lost from the system when the node's inflow exceeds its available storage and outflow capacity, flow units)
- concentration of each pollutant after any treatment applied at the node (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 surface flooding (flow units)
- total outflow from outfalls (flow units)
- total nodal storage volume (ft<sup>3</sup> or m<sup>3</sup>)

## 9.3 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:

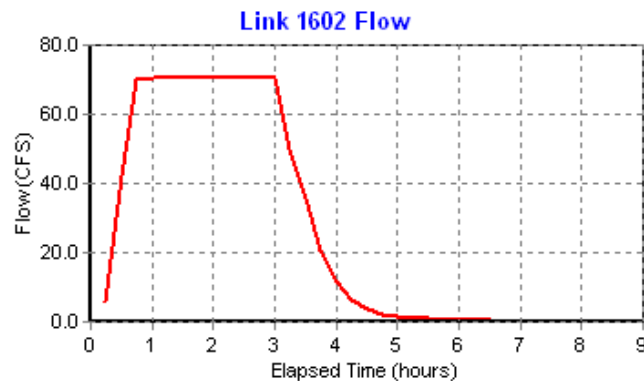
- For the current settings on the Map Browser, the subcatchments, nodes and links of the map will be colored according to their respective Map Legends. The map's color coding will be updated as a new time period is selected in the Map Browser.

- 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 panel of the Map Browser (see Section 4.7).
- The map can be printed, copied to the Windows clipboard, or saved as a DXF file or Windows metafile (see Section 7.12).

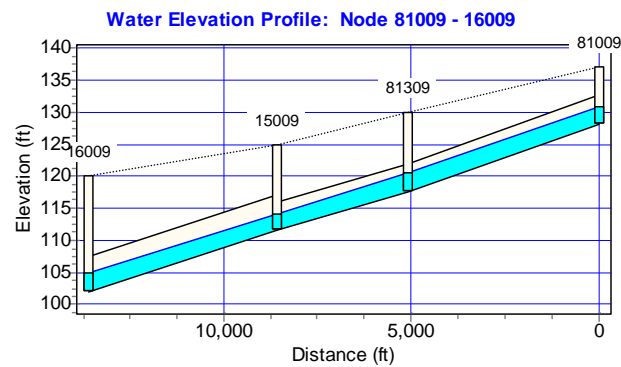
## 9.4 Viewing Results with a Graph

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:

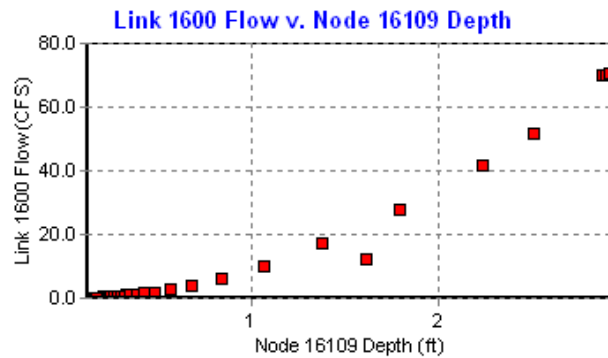


- Profile Plot:




01/01/2002 01:30:00

- Scatter Plot:




You 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 it 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

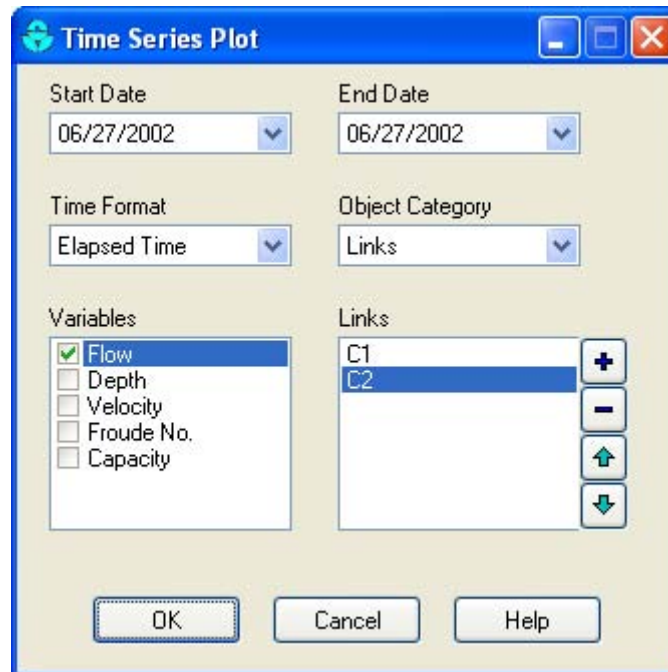
An opened graph will normally be redrawn when a new simulation is run. To prevent the automatic updating of a graph once a new set of results is computed you can lock the current graph by clicking the  icon in the upper left corner of the graph. To unlock the graph, click the icon again.

### 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 >> Time Series** from the Main Menu or click  on the Standard Toolbar.
2. A Time Series Plot dialog will appear. Use it to describe what objects and quantities should be plotted.



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.

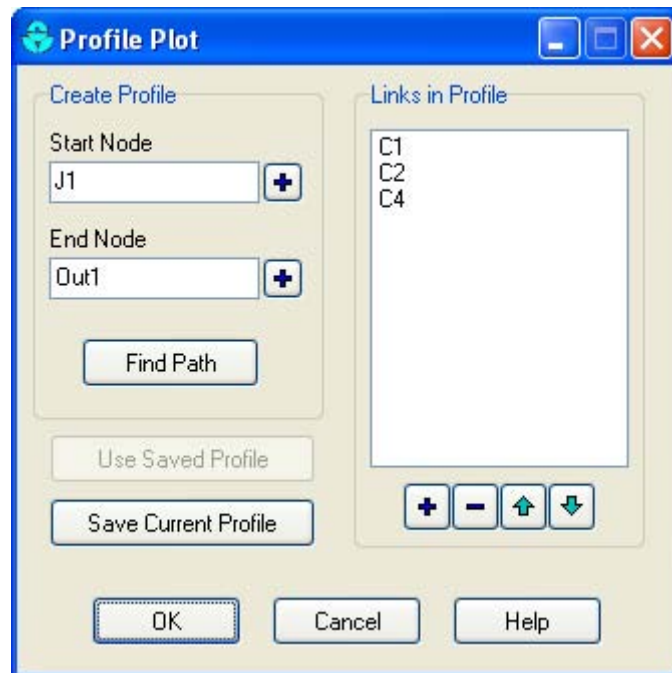
A maximum of 6 objects can be selected for a single plot. Objects already selected can be deleted, moved up in the order or moved down in the order by clicking the , , and  buttons, respectively.

## 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.




To create a Profile Plot:

1. Select **Report >> Graph >> Profile** from the main menu or press  on the Standard Toolbar
2. A Profile Plot dialog will appear (see below). 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 with the smallest number of links between the start and end nodes. These will be listed in the Links in Profile box.
4. You can insert a new link into the Links in Profile list by selecting the new link either on the Study Area Map or in the Data Browser and then clicking the  button underneath the Links in Profile list box.


5. Entries in the Links in Profile list can be deleted or rearranged by using the , , and  buttons underneath the list box.
6. 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.


Profile plots can also be created before any simulation results are available, to help visualize and verify the vertical layout of a drainage system. Plots created in this manner will contain a refresh button  in the upper left corner that can be used to redraw the plot after edits are made to any elevation data appearing in the plot.

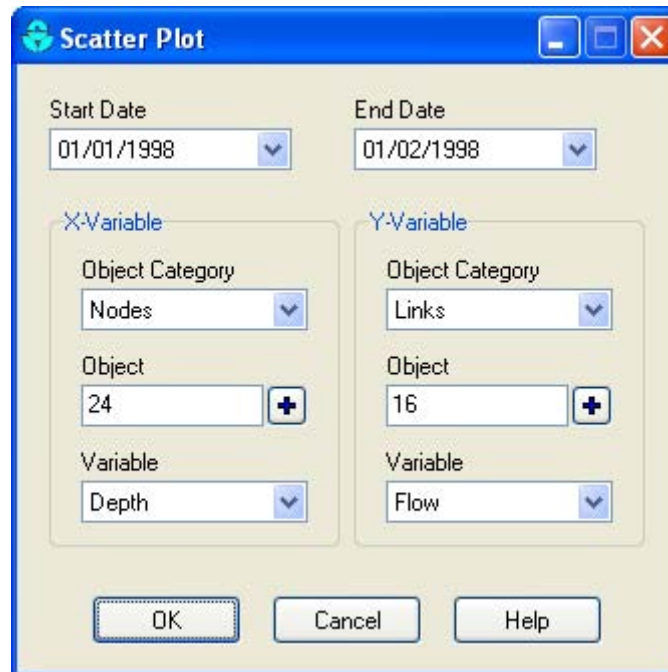
## Scatter Plots

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 >> Scatter** from the main menu or press  on the Standard Toolbar
2. 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):
  - a. Object Category (Subcatchment, Node or Link)
  - b. 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)
  - c. 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:

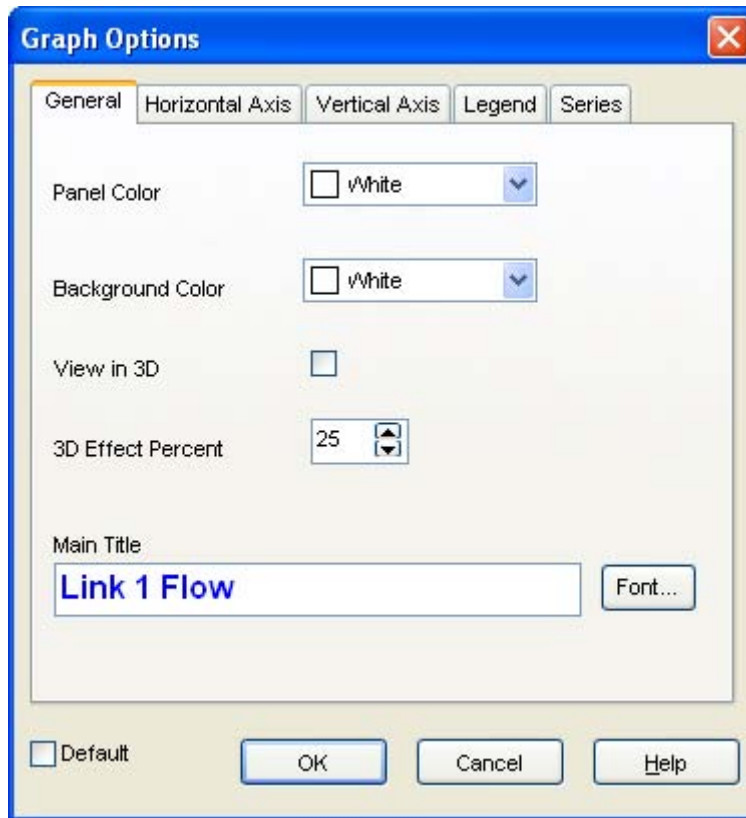
1. Make the graph the active window (click on its title bar).
2. Select **Report >> Customize** from the Main Menu, or click  on the Standard Toolbar, or right-click on the graph.
3. Use the Graph Options dialog that appears to customize the appearance of a Time Series or Scatter Plot, or use the Profile 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, and 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:

---

<i>Panel Color</i>	Color of the panel that contains the graph
<i>Background Color</i>	Color of graph's plotting area
<i>View in 3D</i>	Check if graph should be drawn in 3D
<i>3D Effect Percent</i>	Degree to which 3D effect is drawn
<i>Main Title</i>	Text of graph's main title
<i>Font</i>	Click to set the font used for the main title

---

## 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.

---

<i>Minimum</i>	Sets minimum axis value (minimum data value is shown in parentheses). Can be left blank.
<i>Maximum</i>	Sets maximum axis value (maximum data value is shown in parentheses). Can be left blank.
<i>Increment</i>	Sets increment between axis labels. Can be left blank.
<i>Auto Scale</i>	If checked then Minimum, Maximum, and Increment settings are ignored.
<i>Gridlines</i>	Toggles the display of grid lines on and off.
<i>Axis Title</i>	Text of axis title.
<i>Font</i>	Click to select a font for the axis title.

---

## Graph Options - Legend

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

---

<i>Position</i>	Selects where to place the legend.
<i>Color</i>	Selects color to use for legend background.
<i>Symbol Width</i>	Selects width to use (in pixels) to draw the symbol portion of the legend.
<i>Framed</i>	Places a frame around the legend.
<i>Visible</i>	Makes the legend visible.

---

## 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 three pages:

### Colors:

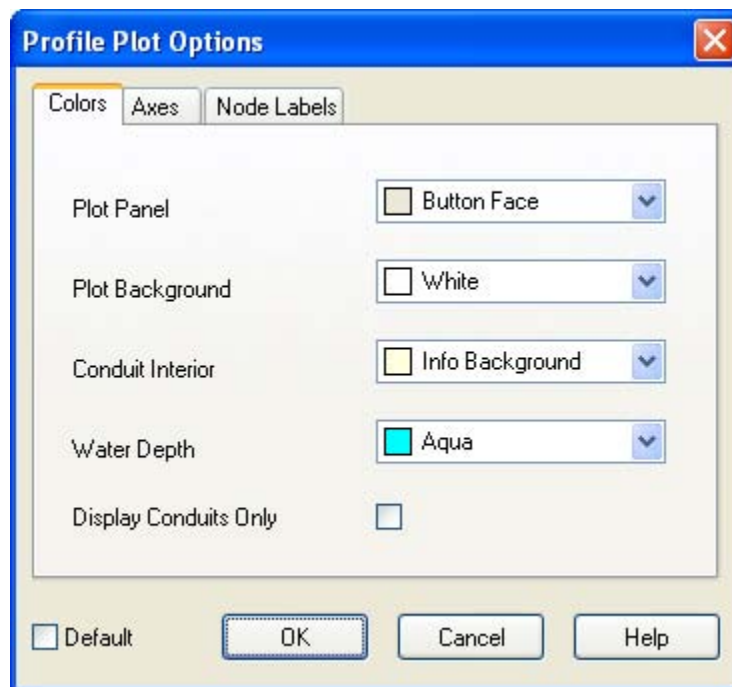
- selects the color to use for the plot window panel, the plot background, a conduit's interior, and the depth of filled water
- includes a "Display Conduits Only" check box that provides a closer look at the water levels within conduits by removing all other details from the plot.

### Axes:

- edits the main and axis titles, including their fonts
- selects to display horizontal and vertical axis grid lines.

### Node Labels:

- selects to display node ID labels either along the plot's top axis, directly on the plot above the node's crown height, or both.
- selects the length of arrow to draw between the node label and the node's crown on the plot (use 0 for no arrows).
- selects the font size of the node ID labels.



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

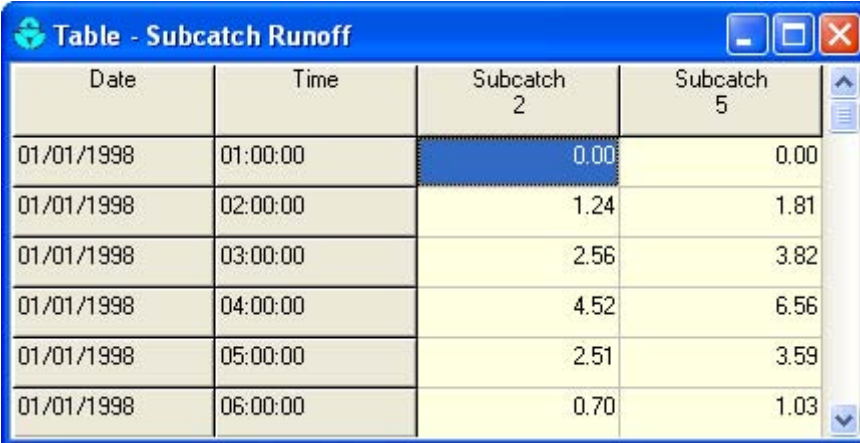
Time series results for selected variables and objects can also be viewed in a tabular format. There are two types of formats available:

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




Date	Time	Flow (CFS)	Depth (ft)
01/01/1998	01:00:00	0.00	0.00
01/01/1998	02:00:00	2.58	0.40
01/01/1998	03:00:00	4.87	0.55
01/01/1998	04:00:00	5.41	0.58
01/01/1998	05:00:00	5.23	0.57
01/01/1998	06:00:00	1.74	0.33

- 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).

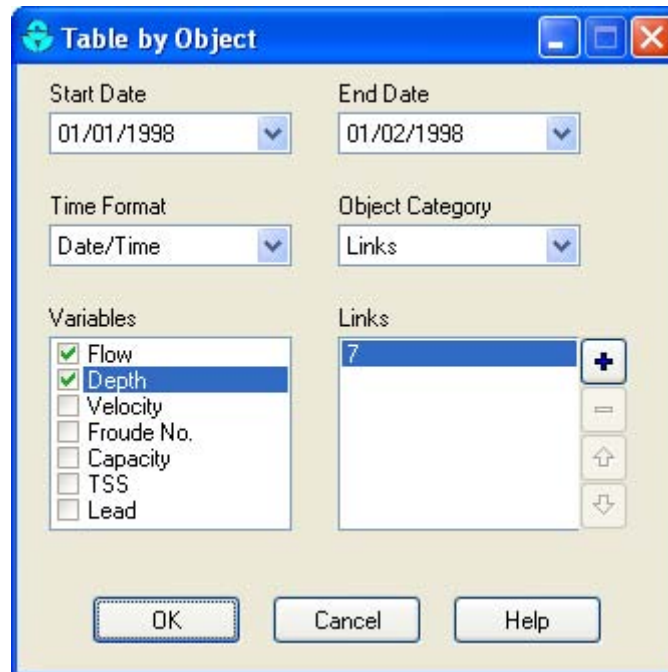



Date	Time	Subcatch 2	Subcatch 5
01/01/1998	01:00:00	0.00	0.00
01/01/1998	02:00:00	1.24	1.81
01/01/1998	03:00:00	2.56	3.82
01/01/1998	04:00:00	4.52	6.56
01/01/1998	05:00:00	2.51	3.59
01/01/1998	06:00:00	0.70	1.03

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.

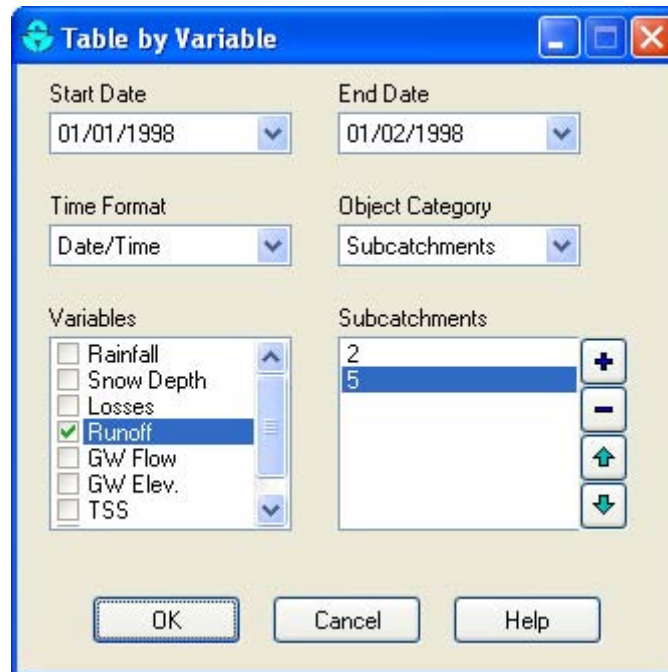
The Table by Object dialog is used when creating a time series table of several variables for a single object. Use the dialog as follows:






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  $\Sigma$  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:

The screenshot shows the 'Statistics Selection' dialog box. It has a title bar with a green icon and standard window controls (minimize, maximize, close). The main area contains the following fields:

- Object Category: Subcatchment (dropdown)
- Object Name: S1 (text field with a '+' button)
- Variable Analyzed: Rainfall (dropdown)
- Event Time Period: Event-Dependent (dropdown)
- Statistic: Mean (dropdown)

Below these fields is a section titled 'Event Thresholds' with three sub-fields:


- Rainfall: 0 (text field)
- Event Volume: 0 (text field)
- Separation Time: 6 (text field)

At the bottom of the dialog are three buttons: 'OK', 'Cancel', and 'Help'.

#### Object Category

Select the category of object to analyze (Subcatchment, Node, Link or System).

#### Object Name

Enter the ID name of the object to analyze. Instead of typing in an ID name, you can select the object on the Study Area Map or in the Data Browser and then click the  button to select it into the Object Name field.

#### Variable Analyzed

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

#### Event Time Period

Select 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.

### Statistic

Choose an 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 inter-event time (i.e., the time interval between the midpoints of successive events). For water quality variables the choices include mean concentration, peak concentration, mean loading, peak loading, and event total load.

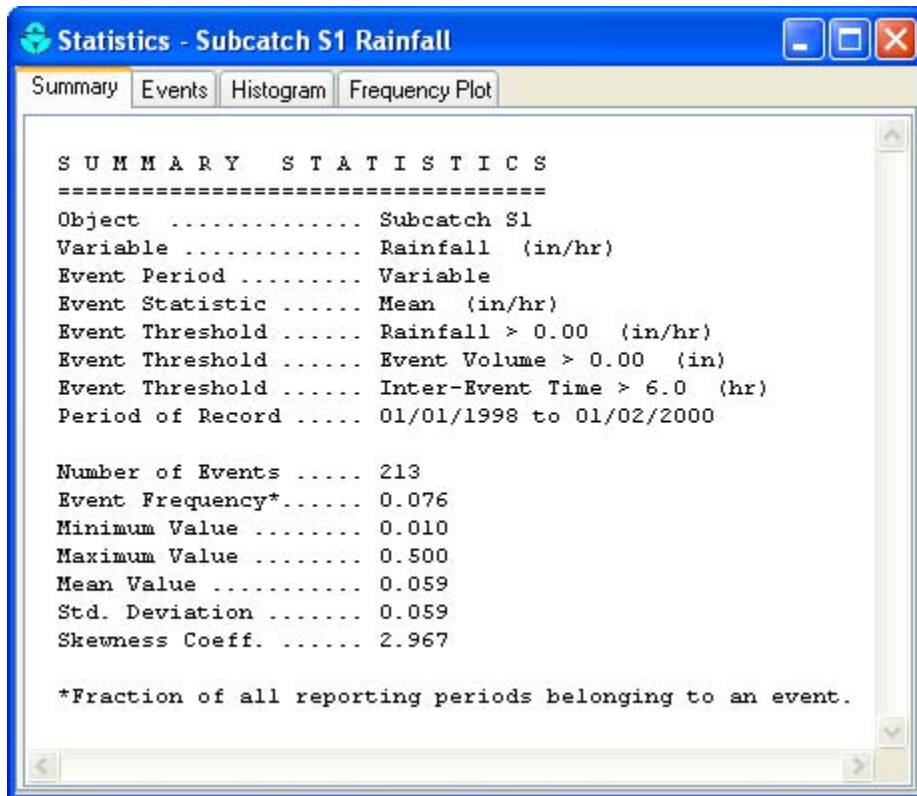
### Event Thresholds

These define minimum values that must be exceeded for an event to occur:

- The Analysis Variable threshold specifies the minimum value of the variable being analyzed that must be exceeded for a time period to be included in an event.
- The Event Volume threshold specifies a minimum flow volume (or rainfall volume) that must be exceeded for a result to be counted as part of an event.
- Separation Time sets the minimum number of hours that must occur between the end of one event and the start of the next event. Events with fewer hours are combined together. This value applies only to event-dependent time periods (not to daily or monthly event periods).

If a particular type of threshold does not apply, then leave the field blank.

After the choices made on the Statistics Selection dialog form are processed, a Statistics Report is produced as shown below.





The report consists of four 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
- an exceedance frequency plot of the event values.

Note that the exceedance frequencies included in the report are computed with respect to the number of events that occur, not the total number of reporting periods.

(This page intentionally left blank.)

## CHAPTER 10 - PRINTING AND COPYING

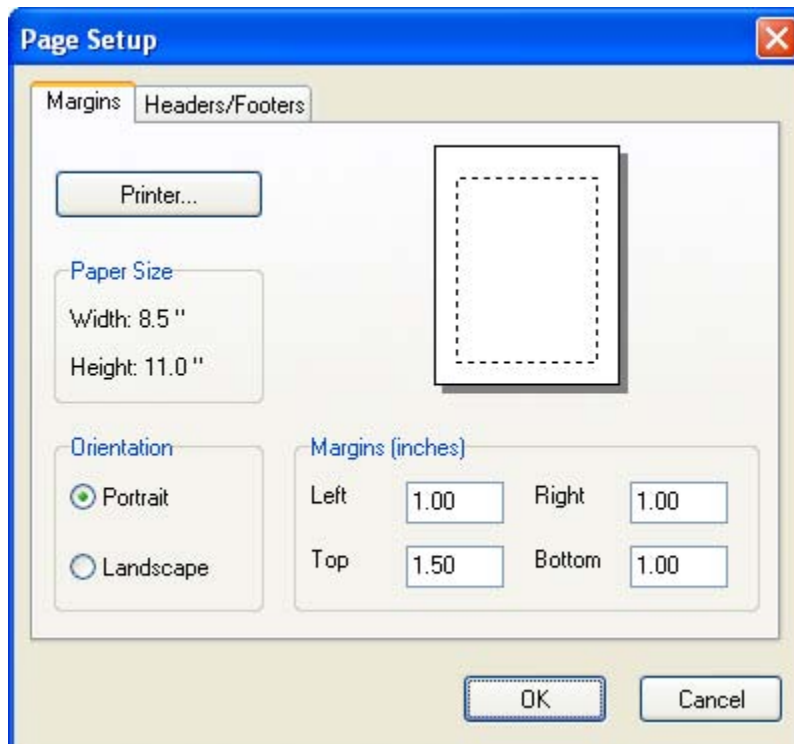
---

*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.

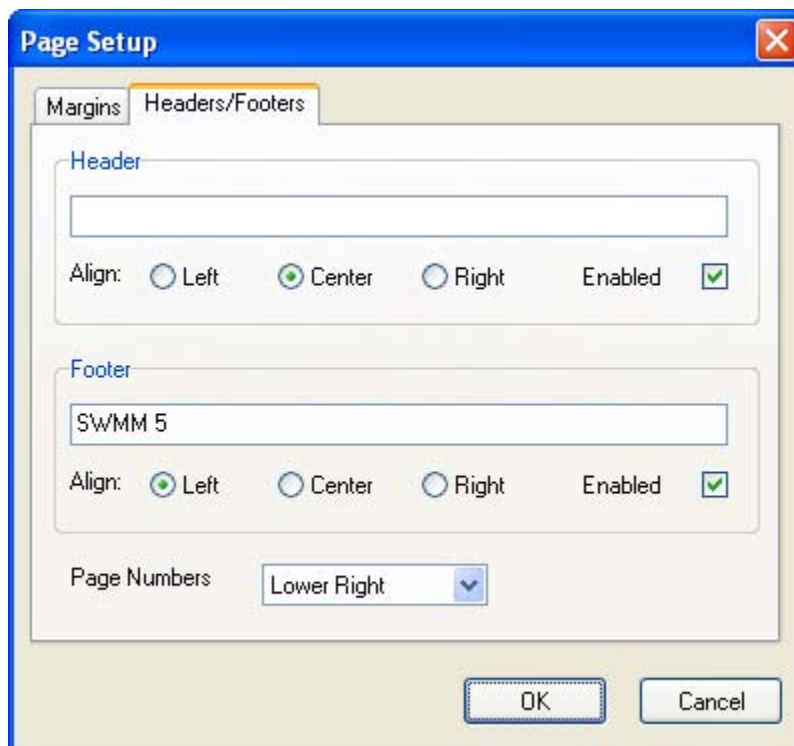


**Figure 10-1.** The Margins page of the Page Setup dialog.

## 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:
3. Select a printer.
4. Select the paper orientation (Portrait or Landscape).
5. Set left, right, top, and bottom margins.
6. Use the Headers/Footers page of the dialog box (Figure 10-2) to:
7. Supply the text for a header that will appear on each page.
8. Indicate whether the header should be printed or not and how its text should be aligned.
9. Supply the text for a footer that will appear on each page.
10. Indicate whether the footer should be printed or not and how its text should be aligned.
11. Indicate whether pages should be numbered.
12. 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.

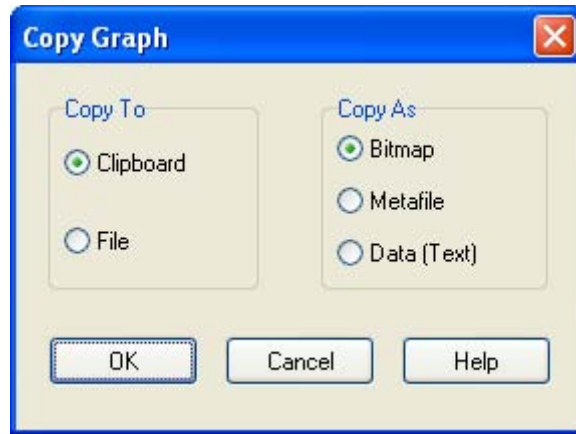
### 10.5 Copying to the Clipboard or to a File

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
- information about the 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 either saved or closed, the report and output files are saved with the same name as the project file, but with extensions of *.rpt* and *.out*. This will happen automatically if the program preference *Prompt to Save Results* is turned off (see Section 4.9). Otherwise the user is asked if the current results should be saved or not. If results are saved then the next time the project is opened, the results from these files will automatically be available for viewing.

### 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](http://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](http://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 and the name of the recording station referenced in the file. 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.

### 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](http://www.ncdc.noaa.gov/oa/ncdc.html).
- Canadian climate files available from Environment Canada at [www.climate.weatheroffice.ec.gc.ca](http://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.

## 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 Groundwater Flow
- Subcatchment Groundwater Elevation
- Subcatchment Snow Pack Depth
- Subcatchment Pollutant Washoff
- Node Depth
- Node Lateral Inflow
- Node Flooding
- Node Water Quality
- Link Flow

Calibration files are registered to a project by selecting **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. 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. The file must be created and edited outside of SWMM, using a text editor or spreadsheet program. A time series file can be linked to a specific time series object using SWMM's Time Series Editor.

The format of a time series file consists of one time series value per line. Comment lines can be inserted anywhere in the file as long as they begin with a semicolon. 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:

```

;Rainfall Data for Gage G1
07/01/2003  00:00  0.00000
              00:15  0.03200
              00:30  0.04800
              00:45  0.02400
              01:00  0.0100
07/06/2003  14:30  0.05100
              14:45  0.04800
              15:00  0.03000
              18:15  0.01000

```



In earlier releases of SWMM 5, a time series file was required to have two header lines of descriptive text at the start of the file that did not have to begin with the semicolon comment character. These files can still be used as long as they are modified by inserting a semicolon at the front of the first two lines.



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
- hot start 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 is an external text file that provides rainfall time series data for a single rain gage. The former is a binary file created internally by SWMM that processes all of 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.

### Hot Start Files

Hot start 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 hot start file saved after a run can be used to define the initial conditions for a subsequent run.

Hot start 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 hot start file from this run is then used to initialize a subsequent run where the inflows of real interest are imposed.

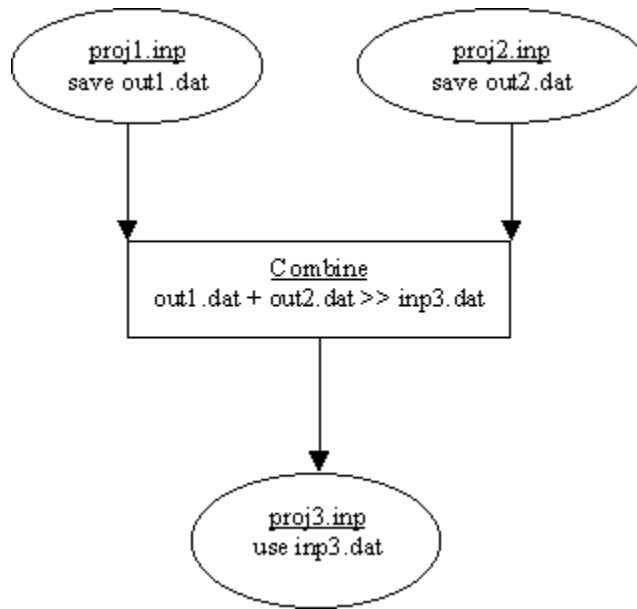
It is also possible to both use and save a hot start file in a single run, starting off the run with one file and saving the ending results either to the same or to another file. The resulting file can then serve as the initial conditions for a subsequent run if need be. This technique can be used to 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-dependent 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.



**Figure 11-1. Example of using the Combine utility to merge Routing files together.**

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.

#### RDII / Routing File Format

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

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

## CHAPTER 12 - USING ADD-IN TOOLS

---

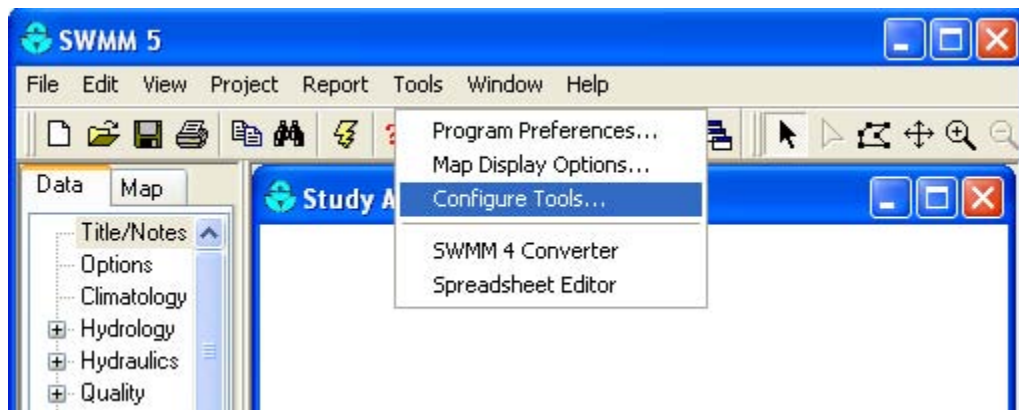
*SWMM 5 has the ability to launch external applications from its graphical user interface that can extend its capabilities. This section describes how such tools can be registered and share data with SWMM 5.*

### 12.1 What Are Add-In Tools

Add-in tools are third party applications that users can add to the **Tools** menu of the main SWMM menu bar and be launched while SWMM is still running. SWMM can interact with these applications to a limited degree by exchanging data through its pre-defined files (see Chapter 11) or through the Windows clipboard. Add-in tools can provide additional modeling capabilities to what SWMM already offers. Some examples of useful add-ins might include:

- a tool that performs a statistical analysis of long-term rainfall data prior to adding it to a SWMM rain gage,
- an external spreadsheet program that would facilitate the editing of a SWMM data set,
- a unit hydrograph estimator program that would derive the R-T-K parameters for a set of RDII unit hydrographs which could then be copied and pasted directly into SWMM's Unit Hydrograph Editor,
- a post-processor program that uses SWMM's hydraulic results to compute suspended solids removal through a storage unit,
- a third-party dynamic flow routing program used as a substitute for SWMM's own internal procedure.

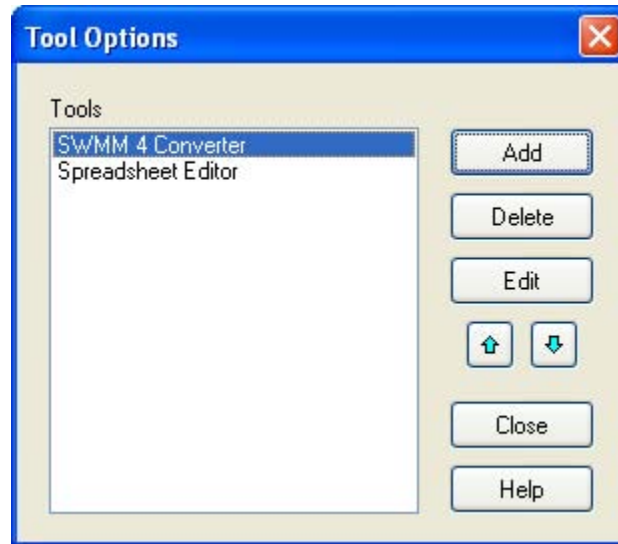
Figure 12-1 shows what the **Tools** menu might look like after several add-in tools have been registered with it. The **Configure Tools** option is used to add, delete, or modify add-in tools. The options below this are the individual tools that have been made available (by this particular user) and can be launched by selecting them from the menu.



**Figure 12-1. SWMM's Tools menu.**

## 12.2 Configuring Add-In Tools

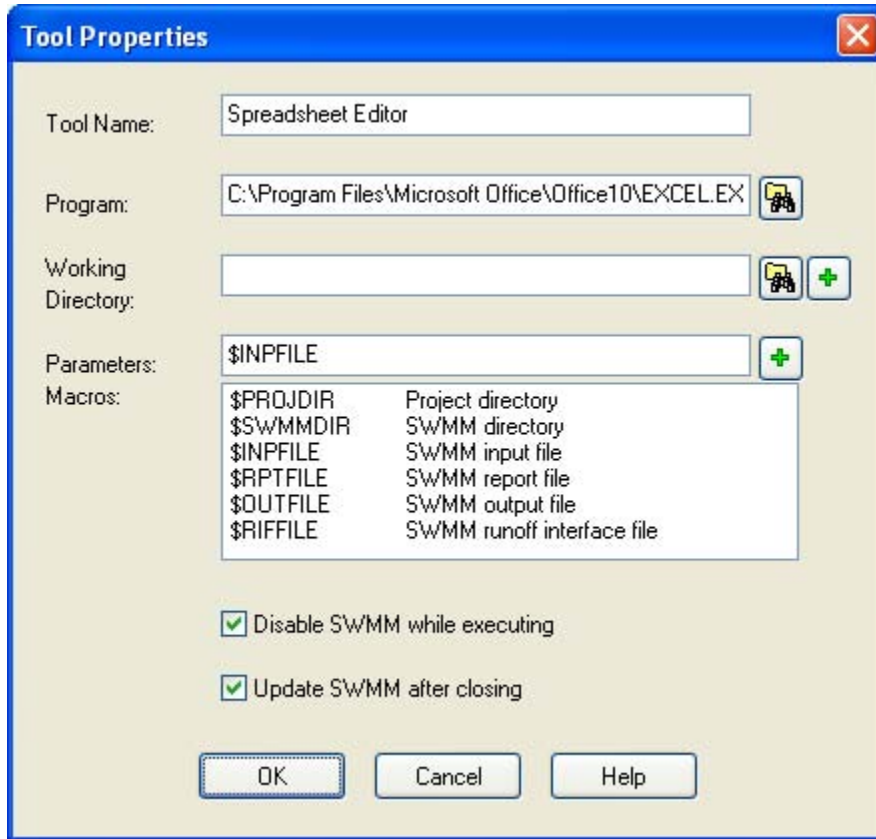
To configure one's personal collection of add-in tools, select **Configure Tools** from the **Tools** menu. This will bring up the Tool Options dialog as shown in Figure 12-2. The dialog lists the currently available tools and has command buttons for adding a new tool and for deleting or editing an existing tool. The up and down arrow buttons are used to change the order in which the registered tools are listed on the **Tools** menu.



**Figure 12-2. The Tools Options dialog.**

Whenever the **Add** or **Edit** button is clicked on this dialog a Tool Properties dialog will appear as shown in Figure 12-3. This dialog is used to describe the properties of the new tool being added or the existing tool being edited.






**Figure 12-3. The Tool Properties dialog.**

The data entry fields of the Tool Properties dialog consist of the following:



**Tool Name**

This is the name to be used for the tool when it is displayed in the Tools Menu.


**Program**

Enter the full path name to the program that will be launched when the tool is selected. You can click the  button to bring up a standard Windows file selection dialog from which you can search for the tool's executable file name.

**Working Directory**

This field contains the name of the directory that will be used as the working directory when the tool is launched. You can click the  button to bring up a standard directory selection dialog from which you can search for the desired directory. You can also enter the macro symbol **\$PROJDIR** to utilize the current SWMM project's directory or **\$SWMMDIR** to use the directory where the SWMM 5 executable resides. Either of these macros can also be inserted into the Working Directory field by selecting its name in the list of macros provided on the dialog and then clicking the  button. This field can be left blank, in which case the system's current directory will be used.

## Parameters

This field contains the list of command line arguments that the tool's executable program expects to see when it is launched. Multiple parameters can be entered as long as they are separated by spaces. A number of special macro symbols have been pre-defined, as listed in the Macros list box of the dialog, to simplify the process of listing the command line parameters. When one of these macro symbols is inserted into the list of parameters, it will be expanded to its true value when the tool is launched. A specific macro symbol can either be typed into the Parameters field or be selected from the Macros list (by clicking on it) and then added to the parameter list by clicking the  button. The available macro symbols and their meanings are defined in Table 12-1 below.

As an example of how the macro expansion works, consider the entries in the Tool Properties dialog shown in Figure 12-3. This Spreadsheet Editor tool wants to launch Microsoft Excel and pass it the name of the SWMM input data file to be opened by Excel. SWMM will issue the following command line to do this

```
c:\Program Files\Microsoft Office\Office10\EXCEL.EXE $INPFILE
```

where the string **\$INPFILE** is replaced by the name of a temporary file that SWMM creates internally which will contain the current project's data.

**Table 12-1. Macros Used as Command Line Parameters for External Tools**

<b>MACRO SYMBOL</b>	<b>EXPANDS TO</b>
<b>\$PROJDIR</b>	The directory where the current SWMM project file resides.
<b>\$SWMMDIR</b>	The directory where the SWMM 5 executable is installed.
<b>\$INPFILE</b>	The name of a temporary file containing the current project's data that is created just before the tool is launched.
<b>\$RPTFILE</b>	The name of a temporary file that is created just before the tool is launched and can be displayed after the tool closes by using the <b>Report &gt;&gt; Status</b> command from the main SWMM menu.
<b>\$OUTFILE</b>	The name of a temporary file to which the tool can write simulation results in the same format used by SWMM 5, which can then be displayed after the tool closes in the same fashion as if a SWMM run were made.
<b>\$RIFFILE</b>	The name of the Runoff Interface File, as specified in the Interface Files page of the Simulation Options dialog, to which runoff simulation results were saved from a previous SWMM run (see Sections 8.1 and 11.7).

### **Disable SWMM while executing**

Check this option if SWMM should be minimized and disabled while the tool is executing. Normally you will need to employ this option if the tool produces a modified input file or output file, such as when the **\$INPFILE** or **\$OUTFILE** macros are used as command line parameters. When this option is enabled, SWMM's main window will be minimized and will not respond to user input until the tool is terminated.

### **Update SWMM after closing**

Check this option if SWMM should be updated after the tool finishes executing. This option can only be selected if the option to disable SWMM while the tool is executing was first selected. Updating can occur in two ways. If the **\$INPFILE** macro was used as a command line parameter for the tool and the corresponding temporary input file produced by SWMM was updated by the tool, then the current project's data will be replaced with the data contained in the updated temporary input file. If the **\$OUTFILE** macro was used as a command line parameter, and its corresponding file is found to contain a valid set of output results after the tool closes, then the contents of this file will be used to display simulation results within the SWMM workspace.

Generally speaking, the suppliers of third-party tools will provide instructions on what settings should be used in the Tool Properties dialog to properly register their tool with SWMM.

(This page intentionally left blank.)