Chapter

Welcome to Bentley SewerGEMS V8*i* Help

1

DAA038120-1/0001

Theory 719

Chapter

Getting Started 1

Thank you for purchasing Bentley SewerGEMS V8*i*. At Bentley Systems, we pride ourselves in providing the very best engineering software available. Our goal is to make software that is easy to install and use, yet so powerful and intuitive that it anticipates your needs without getting in your way.

When you first use Bentley SewerGEMS V8*i*, use the intuitive interface and interactive dialog boxes to guide you. If you need more information, use the online help by pressing the **F1** key or selecting Bentley SewerGEMS V8*i* **Help** from the Help menu. A help topic describing the area of the program in which you are working appears.

- "What is Bentley SewerGEMS V8i?" on page 1-1

What is Bentley SewerGEMS V8*i*?

Bentley SewerGEMS V8*i* is the first and only fully-dynamic, multi-platform (GIS, CAD, and Stand-Alone) sanitary and combined sewer modeling solution. With Bentley SewerGEMS V8*i*, you will analyze all sanitary and combined sewer system elements in one package and have the option of performing the analyses with the SWMM algorithm or our own implicit solution of the full Saint Venant equations.

Simply put, Bentley SewerGEMS V8*i* offers the most comprehensive solution available for optimizing Best Management Practice (BMP) designs and meeting sanitary sewer overflow (SSO) and combined sewer overflow (CSO) regulations.

With Bentley SewerGEMS V8*i*, you can:

- Develop system master plans
- Assess the impact of inflow and infiltration on SSOs
- Develop SSO and CSO remediation programs
- Perform system evaluations associated with US EPA CMOM and NPDES
- Optimize lift station and system storage capacities
- Determine developer connection fees

- Implement real-time control strategies
- Model relief sewers, overflow diversions, and inverted siphons
- Accurately simulate operations with variable-speed pumping and logical controls
- Simulate out-of-service or proposed sewers within the same model

Related Topics

- "Documentation" on page 1-4
- "Quick Start Lessons" on page 1-5
- "Contacting Us" on page 1-6
- "About Haestad Methods" on page A-919

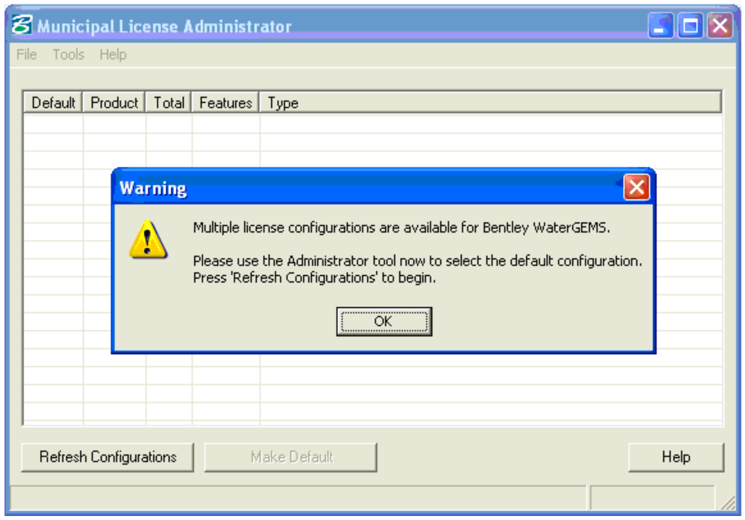
Installation, Upgrades, and Updates

For instructions on installing, registering, activating, and updating the software please refer to the Readme.pdf in the Program Files/Bentley/SewerGEMS V8*i* directory.

Municipal License Administrator Auto-Configuration

At the conclusion of the installation process, the Municipal License Administrator will be executed, to automatically detect and set the default configuration for your product, if possible. However, if multiple license configurations are detected on the license server, you will need to select which one to use by default, each time the product starts. If this is the case, you will see the screen below. Simply press OK to clear the

Warning dialog, then press Refresh Configurations to display the list of available configurations. Select one and press Make Default, then exit the License Administrator. (You only need to repeat this step if you decide to make a different configuration the default in the future.)



Software Updates via the Web and Bentley SELECT

Note: Your PC must be connected to the Internet to use the Check for Updates button.

Bentley SELECT is the comprehensive delivery and support subscription program that features product updates and upgrades via Web downloads, around-the-clock technical support, exclusive licensing options, discounts on training and consulting services, as well as technical information and support channels. It's easy to stay up-to-date with the latest advances in our software. Software updates can be downloaded from our Web site, and your version of Bentley SewerGEMS V8*i* can then be upgraded to the current version quickly and easily. Just click the **Check for Updates** button on the toolbar to launch your preferred Web browser and open our Web site. The Web site automatically checks to see if your installed version is the latest available, and if not, it provides you with the opportunity to download the correct upgrade to bring it up-to-date. You can also access our Knowledgebase for answers to your Frequently Asked Questions (FAQs).

For more information, see "Technical Support" on page 1-6.

Troubleshooting

Because of the multitasking capabilities of Windows, you may have applications running in the background that make it difficult for software setup and installations to determine the configuration of your current system. If you have difficulties during the installation or uninstallation process, please try these steps before contacting our technical support staff:

1. Shut down and restart your computer.
2. Verify that there are no other programs running. You can see applications currently in use by pressing Ctrl+Shift+Esc in Windows 2000 and Windows XP. Exit any applications that are running.
3. Disable any antivirus software that you are running.

   Caution: After you install Bentley SewerGEMS V8*i*, make certain that you restart any antivirus software you have disabled. Failure to restart your antivirus software leaves you exposed to potentially destructive computer viruses.

4. Try running the installation or uninstallation again (without running any other program first).

If these three steps fail to successfully install or uninstall the product, contact our Technical Support staff. For more information, see "Contacting Us" on page 1-6.

Documentation

Bentley SewerGEMS V8*i* documentation comes in three parts:

Online help: The online help is accessible from the **Help** menu or by pressing **F1**. Additionally, when you are using Bentley SewerGEMS V8*i*, you can call the online help at any time by clicking a help button in any dialog box or window.

The context-sensitive online help is designed to make it easy for you to quickly find specific information about a feature you are using in Bentley SewerGEMS V8*i*. The online help makes extensive use of hyperlinks and provides a table of contents, index, and keyword search to help you locate the information you need.

Online PDF Book: The content in the online help is also available in .pdf format and is available at docs.bentley.com. This pdf contains the same content as the online help, but includes hypertext and is designed to be printed by you from a local printing device. As well as being more easily printable than the online help, the online book also uses hypertext and is searchable.

Note: On-screen display of graphics in .pdf files is dependent on the zoom level you use. For more optimal viewing of graphics in Adobe® Acrobat® Reader, try using 167% and 208% zoom.

• "What is Bentley SewerGEMS V8i?" on page 1-1

Quick Start Lessons

The lessons quickly introduce you to specific features of Bentley SewerGEMS V8*i*. To access the lessons, select **Quick Start Lessons** from the **Help** menu. Run a lesson by selecting one of the entries in the list.

• "What is Bentley SewerGEMS V8i?" on page 1-1

Contacting Us

Contact Bentley Systems if you want product information, to upgrade your software, or need technical support.

- "What is Bentley SewerGEMS V8i?" on page 1-1

Sales

Bentley Systems, Inc.' professional staff is ready to answer your questions. Please contact your sales representative for any questions regarding Bentley Systems, Inc.' latest products and prices.

Toll-free U.S. Phone:	800-727-6555
Worldwide Phone:	+1-203-755-1666
Fax:	+1-203-597-1488
Email:	sales@bentley.com

Technical Support

We hope that everything runs smoothly and you never have a need for our technical support staff. However, if you do need support, our highly-skilled staff offers their services seven days a week, and may be contacted by phone, fax, email, and the Internet.

When calling for support, in order to assist our technicians in troubleshooting your problem, please be in front of your computer and have the following information available:

- Your computer's operating system.
- Name and build number of the Bentley Systems, Inc. software you are calling about. The build number can be determined by clicking **Help > About** Bentley SewerGEMS V8*i*. The build number is the number in brackets located in the lower-left corner of the dialog box that opens.
- A note of exactly what you were doing when you encountered the problem.
- Any error messages or other information displayed on your screen.

When emailing or faxing for support, please provide the following details, in addition to the above, to enable us to provide a more timely and accurate response:

- Company name, address, and phone number
- A detailed explanation of your concerns
- If you are emailing us, the Bentley SewerGEMS V8*i*.log files located in the product directory (e.g., C:\Documents and Settings\<User Name>\Local Settings\Application Data\Bentley\SewerGEMS V8*i*\8)

Support Hours

:Technical Support is available 24 hours a day, seven days a week.

You can contact our technical support team at:

Phone:	+1-203-755-1666
Fax:	+1-203-597-1488
Email:	support@bentley.com

Addresses

Use this address information to contact us:

Internet:	http://www.bentley.com
Email:	support@bentley.com
	sales@bentley.com
Toll-free U.S. Phone:	800-727-6555
Worldwide Phone:	+1-203-755-1666
Fax:	+1-203-597-1488
Mail:	Bentley Systems, Inc., Incorporated
	Haestad Methods Solutions Center
	Suite 200W
	37 Brookside Road
	Watertown, CT 06795

Your Suggestions Count

Bentley Systems, Inc. strives to continually provide you with sophisticated software and documentation. We are very interested in hearing your suggestions for improving the Bentley SewerGEMS V8*i* software, online help, and printed manual. Your feedback guides us in developing products that make your work easier.

Please let us hear from you!

Chapter

Introducing the Workspace

2

The Workspace

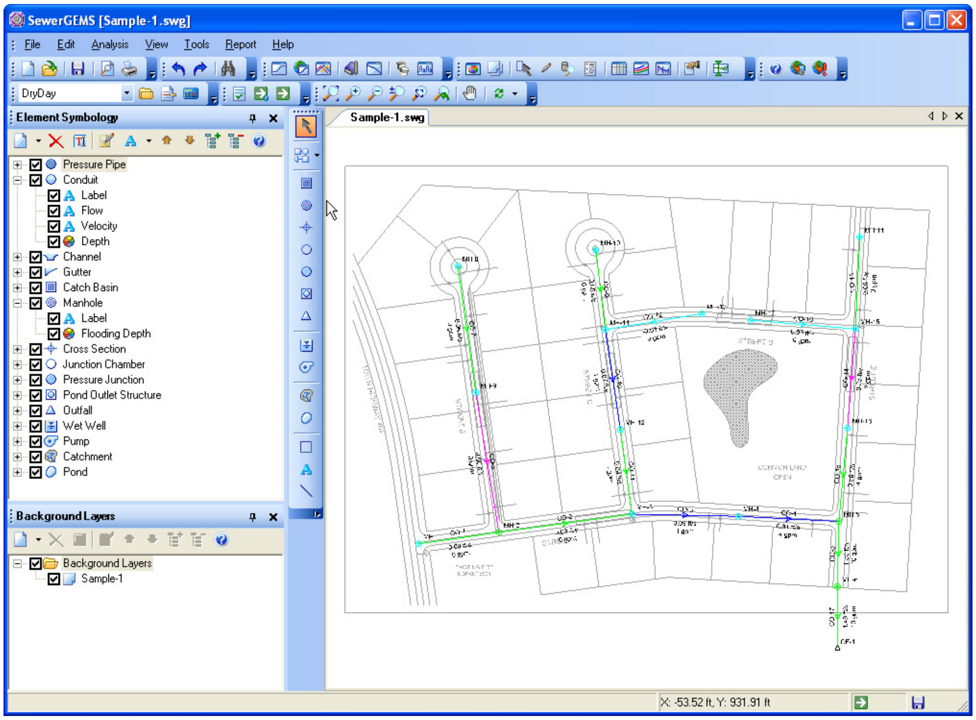
You use Bentley SewerGEMS V8*i* in one of these modes:

- "Stand-Alone Editor" on page 2-9
- "MicroStation Mode" on page 2-10
- "ArcGIS Mode" on page 2-11
- "AutoCAD Mode" on page 2-11

Stand-Alone Editor

The Stand-Alone Editor is the workspace that contains the various managers, toolbars, and menus, along with the drawing pane, that make up the Bentley SewerGEMS V8*i* interface. The Bentley SewerGEMS V8*i* interface uses dockable windows and toolbars, so the position of the various interface elements can be manually adjusted to suit your preference.

By default, the Bentley SewerGEMS V8*i* environment looks like this:



MicroStation Mode

MicroStation mode lets you create and model your network directly within your primary drafting environment. This gives you access to all of MicroStation's drafting and presentation tools, while still enabling you to perform Bentley SewerGEMS V8*i* modeling tasks like editing, solving, and data management. This relationship between Bentley SewerGEMS V8*i* and MicroStation enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in MicroStation. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

Note: **For more information about runnning Bentley SewerGEMS V8*i* in MicroStation mode, see** “MicroStation Environment” on page 12-700**.**

ArcGIS Mode

ArcGIS mode lets you create and model your network directly in ArcMap. Each mode provides access to differing functionality—certain capabilities that are available within ArcGIS mode may not be available when working in the Bentley SewerGEMS V8*i* Stand-alone Editor. All the functionality available in the Stand-alone Editor are, however, available in ArcGIS mode.

Note: For more information about running SewerGEMS V8*i* in ArcGIS mode, see “Working in ArcGIS Mode” on page 11-665**.**

AutoCAD Mode

AutoCAD mode lets you create and model your network directly within your primary drafting environment. This gives you access to all of AutoCAD’s drafting and presentation tools, while still enabling you to perform Bentley SewerGEMS V8*i* modeling tasks like editing, solving, and data management. This relationship between Bentley SewerGEMS V8*i* and AutoCAD enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in AutoCAD. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

Note: For more information about runnning Bentley SewerGEMS V8*i* in AutoCAD mode, see “Working in AutoCAD Mode” on page 13-709**.**

Menus

Menus are located at the top of Bentley SewerGEMS V8*i* stand-alone editor window and provide access to program commands, which are broken down by type of functionality.

The following menus are available:

- “File Menu” on page 2-12
- “Edit Menu” on page 2-15
- “Analysis Menu” on page 2-16
- “View Menu” on page 2-19
- “Tools Menu” on page 2-22
- “Report Menu” on page 2-24
- “Help Menu” on page 2-24

File Menu

The File menu contains the following commands:

New	Creates a new project. When you select this command, a new untitled project is created.
Open	Opens an existing project. When you select this command, the Open dialog box appears, allowing you to browse to the project to be opened.
Close	Closes the current project without exiting the program.
Close All	Closes all currently open projects.
Save	Saves the current project.
Save As	Saves the current project under a new project name and/or to a different directory location.
Save All	Saves all currently open projects.

ProjectWise

Opens a submenu containing the following commands:

- **Open**—Open an existing SewerGEMS V8*i* project from ProjectWise. You are prompted to log into a ProjectWise datasource if you are not already logged in.

  Note: Only projects that were originally saved into ProjectWise from the SewerGEMS V8*i* application can be opened

- **Save As**—Saves the current project to a ProjectWise datasource. You are prompted to log into a ProjectWise datasource if you are not already logged in.
- **Change Datasource**—Lets you connect to a different ProjectWise datasource for future Open and Save As operations.
- Import—Lets you import the following file types into the ProjectWise project:
  - **SWMM v5**—Opens a Windows Browse dialog box, allowing you to choose the SWMM v5 file to import.
  - Bentley SewerGEMS V8*i* **Database**—Lets you import a Bentley SewerGEMS V8*i* project database file.
  - **SewerCAD Exchange Database**—Lets you import a SewerCAD Exchange Database file (.swr.mdb file).
  - **StormCAD Exchange Database**—Lets you import a StormCAD Exchange Database file (.stm.mdb file).

  Note: For more information about using SewerGEMS V8*i* with ProjectWise, see “Considerations for ProjectWise Users” on page 4-158

Import	Opens a submenu containing the following commands: • **SWMM v5**—Opens a Windows Browse dialog box, allowing you to choose the SWMM v5 file to import. • Bentley SewerGEMS V8*i* **Database**—Lets you import a Bentley SewerGEMS V8*i* project database file. • **SewerCAD Exchange Database**—Lets you import a SewerCAD Exchange Database file (.swr.mdb file). • **StormCAD Exchange Database**—Lets you import a StormCAD Exchange Database file (.stm.mdb file). • **Bentley Wastewater Import**—Lets you import a an model .mdb and geometry data file (.dat) that was previously exported from Bentley Wastewater application.
Export	Opens a submenu containing the following commands: • **DXF**—Lets you export the current network layout as a DXF drawing. • **SWMM v5**—Lets you export the current project to SWMM format.
Print Setup	Defines the print settings that will be used when the current view is printed.
Print Preview	Opens the Print Preview window, displaying the current view exactly as it will be printed.
Print	Prints the current view.
Recent Files	When the Recent Files Visible option is selected in the Options dialog box, the most recently opened files will appear in the File menu. See "Options Dialog Box - Global Tab" on page 4-147 for more information.
Exit	Closes the program.

Edit Menu

The Edit menu contains the following commands:

Undo	Cancels the last data input action on the currently active dialog box. Clicking Undo again cancels the second-to-last data input action, and so on.
Redo	Cancels the last undo command.
Delete	Deletes the currently highlighted element.
Select By Polygon	Lets you select elements in your model by drawing a polygon in the drawing pane. Click in the drawing pane to draw each side of the polygon. After the polygon has been drawn, right-click to select from the following options: • **As Selected** - All elements contained within the polygon will be selected. Elements that were selected before the Select By Polygon opperation will be de-selected. • **Add to Selection** - All elements contained within the polygon will be selected in addition to any elements that were selected before the Select By Polygon operation were performed. • **Invert Selection** - All elements contained within the polygon that were selected before the operation will be de-selected; all elements contained within the polygon that were not selected before the operation will be selected • **Remove From Selection** - All elements contained within the polygon that were selected before the operation will be de-selected.
Select All	Selects all of the elements in the network.
Invert Selection	Selects all currently unselected elements and deselects all currently selected ones.
Select by Element	Opens a submenu listing all available element types. Select one of the element types from the submenu to select all elements of that type in the model.

Select by Attribute	Opens a menu listing all available attribute types. Select one of the attribute types from the menu and the Query Builder dialog box opens.
Clear Selection	Deselects the currently selected element(s).
Clear Highlight	Removes Network Navigator highlighting for all elements.
Find Element	Lets you find a specific element by entering the element's label.

Analysis Menu

The Analysis menu contains the following commands:

Scenarios	Opens the Scenario Manager, which lets you create, view, and manage project scenarios.
Alternatives	Opens the Alternative Manager, which lets you create, view, and manage alternatives.
Calculation Options	Opens the Calculation Options Manager, which lets you create, view, and manage calculation settings for the project.
EPS Results Browser	Opens the EPS Results Browser dialog box, which lets you manipulate the currently displayed time step and to animate the drawing pane.
Calculation Summary	Opens the calculation summary report, which reports the details of the calculations performed on your model.
User Notifications	Opens the User Notifications Manager, allowing you to view warnings and errors uncovered by the validation process.

Validate	Runs a diagnostic check on the network data to alert you to possible problems that may be encountered during calculation. This is the manual validation command, and it checks for input data errors. It differs in this respect from the automatic validation that SewerGEMS V8*i* runs when the compute command is initiated, which checks for network connectivity errors as well as many other things beyond what the manual validation checks. Pressing CTRL+F7 also selects this command.
Compute Hydrology	Lets you perform the hydrologic calculations for the current scenario. Pressing CTRL+F8 also selects this command.
Compute	Calculates the network. Before calculating, an automatic validation routine is triggered, which checks the model for network connectivity errors and performs other validation. For more information, see "Calculating Your Model" on page 8-431. Pressing F9 also selects this command.
Always Compute Hydrology	Lets you turn hydrology calculations on and off whenever the model is calculated. Turning hydrology computation off improves performance and is recommended when the hydrology input will not change.

Components Menu

The Components menu contains the following commands:

Storm Events	Opens the Storm Events dialog box, which lets you create, edit, and delete storm events. These storms are available for you to select for a catchment. For more information, see "Adding Storm Events" on page 7-395.
Global Storm Events	Opens the Global Storm Event Settings dialog box, which lets you define project-wide global storm event data. For more information, see "Adding Global Storm Events" on page 7-405.

Dimensionless Unit Hydrographs	Opens the Dimensionless Unit Hydrographs dialog box, Which lets you create, edit, and delete dimensionless unit hydrographs.
RTK Tables	Opens the RTK Tables dialog box, which lets you create wet weather flow hydrographs using the RTK method. For more information, see "Adding Hydrographs Based On the RTK Method" on page 7-415.
Pipe Catalog	Opens the Catalog Pipe dialog box, which lets you create, edit, and view catalog pipes. Catalog pipes are an efficient way to reuse common physical pipe definitions. For more information, see "Pipe Catalog Dialog Box" on page 6-291.
Pump Curve Definitions	Opens the Pump Curve Definitions dialog box, which lets you view, edit, and create pump curve definitions. For more information, see "Pump Curve Definitions Dialog Box" on page 6-233.
SWMM Extensions	Opens a submenu containing the following SWMM-specific commands: • **Evaporation**—Opens the Evaporation dialog box, allowing you to view and edit evaporation data for use in SWMM calculations. • **Aquifers**—Opens the Aquifers dialog box, allowing you to view and edit aquifer data for use in SWMM calculations. • **Control Sets**—Opens the Control Sets dialog box, allowing you to view, edit, and create control sets for use in SWMM calculations. • **Pollutants**—Opens the Pollutants dialog box, allowing you to view and edit pollutant data for use in SWMM calculations. • **Pollutographs**—Opens the Pollutograph dialog box, allowing you to view and edit pollutograph data for use in SWMM calculations. • **Land Uses**—Opens the Land Use dialog box, allowing you to view and edit land use data for use in SWMM calculations. For more information, see "Using the SWMM Water Quality Solver" on page 6-294.

Time Series Field Data	Opens the Time Series Field Data dialog, which allows you to define time series field data for the elements in the model.
Unit Sanitary (Dry Weather) Loads	Opens the Unit Sanitary (Dry Weather) Loads dialog box, which lets you create, edit, and delete unit sanitary loads. For more information, see “Adding Unit Sanitary (Dry Weather) Loads” on page 7-370.
Patterns	Opens the Pattern Manager where you can create and edit diurnal loading patterns for use with extended period simulations. For more information, see “Defining Patterns” on page 7-362.
Pattern Setups	Opens the Pattern Setup Manager where you can associate diurnal patterns with the appropriate unit sanitary loads for a given scenario. For more information, see “Defining Pattern Setups” on page 7-365.
Engineering Libraries	Opens the Engineering Libraries Manager.

View Menu

The View menu contains the following commands:

Element Symbology	Opens the Element Symbology Manager, which lets you create, view, and manage annotation and color-coding in your project.
Background Layers	Opens the Background Layers Manager, which lets you create, view, and manage the background layers associated with the project.
Network Navigator	Opens the Network Navigator.
Selection Sets	Opens the Selection Sets Manager, which lets you create, view, and manage selection sets associated with the project.

Queries	Opens the Query Manager, which lets you create SQL expressions for use with selection sets and FlexTables. For more information, see
Prototypes	Opens the Prototypes Manager, which lets you enter default values for elements in your model. Prototypes can reduce data entry requirements dramatically if a group of network elements share common data. For more information, see
FlexTables	Opens the FlexTables Manager, which lets you create, view, and manage the tabular reports for the project.
Graphs	Opens the Graph Manager, which lets you create, view, and manage graphs for the project.
Profiles	Opens the Profile Manager, which lets you create, view, and manage the profiles for the project.
Contours	Opens the Contours manager where you can create and edit contour definitions.
Named Views	Opens the Named Views manager where you can create, edit, and use Named Views.
Aerial View	Opens the Aerial View navigation window.
Properties	Turns the Properties Editor display on or off.
Auto-Refresh	Turns automatic updates to the main window view on or off whenever changes are made to the Bentley SewerGEMS V8*i* datastore. When selected, a check mark appears next to this menu command, indicating that automatic updates are turned on.
Refresh Drawing	Updates the main window view according to the latest information contained in the Bentley SewerGEMS V8*i* datastore.

Zoom	Opens a submenu containing the following commands: • **Zoom Extents**—Sets the view so that the entire network is visible in the drawing pane. • **Zoom Window**—Activates the manual zoom tool, which lets you specify a portion of the drawing to enlarge. • **Zoom In**—Enlarges the size of the model in the drawing pane. • **Zoom Out**—Reduces the size of the model in the drawing pane. • **Zoom Realtime**—Enables the realtime zoom tool, which lets you zoom in and out by moving the mouse while holding down the left mouse button. • **Zoom Center**—Opens the Zoom Center dialog box, which lets you enter drawing coordinates that will be centered in the drawing pane. • **Zoom Previous**—Resets the zoom level to the last setting. • **Zoom Next**—Resets the zoom level to the setting that was active before a Zoom Previous command was executed.
Pan	Activates the Pan tool, which lets you move the model within the drawing pane. When you select this command, the cursor changes to a hand, indicating that you can click and hold the left mouse button and move the mouse to move the drawing.
Toolbars	Opens a submenu that lists each of the available toolbars. Select one of the toolbars in the submenu to turn that toolbar on or off. For more information, see “Toolbars” on page 2-25.
Reset Workspace	Resets the Bentley SewerGEMS V8*i* workspace so that the dockable managers appear in their default factory-set positions.

Tools Menu

The Tools menu contains the following commands:

Active Topology Selection	Opens a Select dialog to select elements in the drawing to make them Inactive or Active.
LoadBuilder	Opens the LoadBuilder manager where you can assign demands to model nodes using data from outside sources.
Hyperlinks	Lets you associate external files, such as pictures or movie files, with elements. For more information, see "Adding Hyperlinks to Elements" on page 6-324.
ModelBuilder	Opens the ModelBuilder Connections Manager, which lets you create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process. For more information, see "ModelBuilder Connections Manager" on page 5-173.
User Data Extensions	**User Data Extensions**—Opens the User Data Extension dialog box, which lets you add and define custom data fields. For example, you can add new fields such as the pipe installation date. For more information, see "User Data Extensions" on page 6-335.
Inflow Control Center	Opens the Inflow Control Center, allowing you to create, edit, and delete sanitary inflow definitions.
Sanitary Load Control Center	Opens the Sanitary Load Control Center, allowing you to create, edit, and delete sanitary load definitions.

Database Utilities	Opens a submenu containing the following commands:

- **Compact Database**—When you delete data from a Bentley SewerGEMS V8*i* project, such as elements or alternatives, the database store that Bentley SewerGEMS V8*i* uses can become fragmented, causing unnecessarily large data files, which impact performance substantially. Compacting the database eliminates the empty data records, thereby defragmenting the datastore and improving the performance of the file.

  Note: Every tenth time a file is saved, Bentley SewerGEMS V8*i* will automatically prompt you to compact the database. If you open a file without saving it, the count does not go up. If you open and save a file multiple times in the same session, the count only goes up on the first save. If you open, save, and close the file, the count goes up. Click Yes to compact the database, or no to close the prompt dialog box without compacting. Since compacting the database can take time, especially for larger models, you may want to postpone the compact procedure until a later time. You can modify how Bentley SewerGEMS V8*i* compacts the database in the Options dialog box. For more information, see “Options Dialog Box - Global Tab” on page 4-147**.**

- **Synchronize Drawing**—Synchronizes the current model drawing with the project database.

Layout	Opens a submenu that lists each of the available element types. Select one of the element types in the submenu to place that element in your model.
External Tools	Run an existing external tool or create a new one by opening up the External Tools manager.

Options	Opens the Options dialog box, which lets you change global settings such as display pane settings, drawing scale, units, display precision and format used, and element labeling.

Report Menu

The Report menu contains the following commands:

Element Tables	Opens a submenu that lets you display FlexTables for any link or node element. These predefined FlexTables contain most of the input data and results for each instance of the selected element in the model.
Scenario Summary	Opens the Scenario Summary Report.
Calculation Executive Summary	Opens the calculation executive summary report, which reports a summary of the calculations performed on your model. For more information, see "Calculation Executive Summary Dialog Box" on page 8-439.
Project Inventory	Opens the Project Inventory Report, which contains the number of each of the various element types that are in the network.
Report Options	Opens the Report Options box where you can set Headers and Footers for the predefined reports.

Help Menu

The Help menu contains the following commands:

Bentley SewerGEMS V8*i* Help	Opens the online help Table of Contents.
Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Welcome Dialog	Opens the Welcome dialog box.

Check for Updates	Opens your Web browser to the our Web site, allowing you to check for Bentley SewerGEMS V8*i* updates.
Bentley Institute Training	Opens your browser to the Bentley Institute Training web site.
Bentley Professional Services	Opens your browser to the Bentley Professional Services web site.
Online Support	Opens your browser to SELECTservices area of the Bentley web site.
Discussion Groups	Opens your browser to Bentley's Haestad Discussion Groups.
Haestad.com	Opens to the Haestad page on the Bentley web site.
Bentley.com	Opens the home page on the Bentley web site.
About Bentley SewerGEMS V8*i*	Opens the About Bentley SewerGEMS V8*i* dialog box, which displays copyright information about the product, registration information, and the current version number of this release.

Toolbars

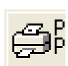
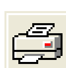
Toolbars provide access to frequently used menu commands and are organized by the type of functionality offered. Many of the toolbars have additional buttons available that are not displayed by default. You can display these additional buttons by following the procedure in "Adding and Removing Toolbar Buttons" on page 2-38.

The following toolbars are available:

- “Layout Toolbar” on page 2-33
- “Zoom Toolbar” on page 2-36

File Toolbar

The File toolbar contains the following buttons:

	**New**	Creates a new Bentley SewerGEMS V8*i* project. When you select this command, the **Select File to Create** dialog box appears, allowing you to define a name and directory location for the new project.
	**Open**	Opens an existing Bentley SewerGEMS V8*i* project. When this command is initialized, the **Select** Bentley SewerGEMS V8*i* **Project to Open** dialog box appears, allowing you to browse to the project to be opened.
	**Save**	Saves the current project.
	**Print Preview**	Opens the Print Preview window, displaying the current view exactly as it will be printed.
	**Print**	Prints the current view of the network as displayed in the drawing pane.

Edit Toolbar

The Edit toolbar contains the following buttons:

	**Undo**	Cancels your most recent action.
	**Redo**	Lets you redo the last cancelled action.
	**Find Element**	Lets you find a specific element by choosing it from a menu containing all elements in the current model.

Analysis Toolbar

The Analysis toolbar contains the following buttons:

	**Storm Events**	Opens the Storm Events dialog box, which lets you create, edit, and delete storm events.
	**Global Storm Event**	Opens the Global Storm Event Settings dialog box, which lets you define project-wide global storm event data.
	**RTK Tables**	Opens the RTK Tables dialog box, which lets you create wet weather flow hydrographs using the RTK method.
	**Pipe Catalog**	Opens the Catalog Pipe dialog box, which lets you create, edit, and view catalog pipes. Catalog pipes are an efficient way to reuse common physical pipe definitions.

	**Pump Curve Definitions**	Opens the Pump Curve Definitions dialog box, which lets you view, edit, and create pump curve definitions.
	**Patterns**	Opens the Pattern Manager where you can create and edit diurnal loading patterns for use with extended period simulations.
	**Unit Sanitary (Dry Weather) Loads**	Opens the Unit Sanitary (Dry Weather) Loads dialog box, which lets you create, edit, and delete unit sanitary loads.
	**Pattern Setups**	Opens the Pattern Setup Manager where you can associate diurnal patterns with the appropriate unit sanitary loads for a given scenario. This button does not appear in the toolbar by default, but can be added. For more information, see "Adding and Removing Toolbar Buttons" on page 2-38.

Scenarios Toolbar

The Scenario toolbar contains the following buttons:

	Scenario List Box	Lets you quickly change the current scenario.
	**Scenarios**	Opens the Scenario manager, which lets you create, view, and manage project scenarios.
	**Alternatives**	Opens the Alternative manager, which lets you create, view, and manage project alternatives.
	**Calculation Options**	Opens the Calculation Options manager, which lets you create different profiles for different calculation settings.

Compute Toolbar

The Compute toolbar contains the following buttons:

	**Validate**	Runs a diagnostic check on the network data to alert you to possible problems that may be encountered during calculation. This is the manual validation command, and it checks for input data errors. It differs in this respect from the automatic validation that SewerGEMS V8*i* runs when the compute command is initiated, which checks for network connectivity errors as well as many other things beyond what the manual validation checks.
	**Compute Hydrology**	Performs hydrology calculations.
	**Compute**	Calculates the network. Before calculating, an automatic validation routine is triggered, which checks the model for network connectivity errors and performs other validation. For more information, see “Calculating Your Model” on page 8-431.
	**User Notifications**	Opens the User Notifications Manager, allowing you to view warnings and errors uncovered by the validation process. This button does not appear in the toolbar by default, but can be added. For more information, see “Adding and Removing Toolbar Buttons” on page 2-38.

View Toolbar

The View toolbar contains the following buttons, which give you easy access to many of the managers in Bentley SewerGEMS V8*i*.:

	**Element Symbology**	Opens the Element Symbology manager, allowing you to create, view, and manage the element symbol settings for the project.
	**Background Layers**	Opens the Background Layers manager, allowing you to create, view, and manage the background layers associated with the project.
	**Selection Sets**	Opens the Selection Sets Manager, allowing you to create, view, and modify the selection sets associated with the project.
	**Network Navigator**	Opens the Network Navigator dialog box.
	**Queries**	Opens and closes the Query Manager.
	**Prototypes**	Opens and closes the Prototypes Manager.
	**FlexTables**	Opens the FlexTables manager, allowing you to create, view, and manage the tabular reports for the project.

	**Graphs**	Opens the Graph manager, allowing you to create, view, and manage the graphs for the project.
	**Profiles**	Opens the Profile manager, allowing you to create, view, and manage the profiles for the project.
	**Properties**	Opens and closes the Property Editor.
	**EPS Results Browser**	Opens the EPS Results Browser manager, allowing you to manipulate the currently displayed time step and to animate the drawing pane.

Help Toolbar

The Help toolbar provides quick access to the same commands that are available in the Help menu. The Help toolbar contains the following buttons.

	**Help**	Opens the Bentley SewerGEMS V8*i* online help.
	**Check for Updates**	Opens your Web browser to our Web site, allowing you to check for Bentley SewerGEMS V8*i* updates.
	**Support**	Opens your Web browser to the Support Center of the Haestad Methods' Web site.

Layout Toolbar

You use the Layout toolbar to lay out your model in the drawing pane. The Layout toolbar contains the following buttons:

	**Select**	Changes your mouse cursor into a selection tool. The selection tool behavior varies depending on the direction in which the mouse is dragged after defining the first corner of the selection box, as follows: If the selection is made from left-to-right, all elements that fall completely within the selection box that is defined will be selected. If the selection is made from right-to-left, all elements that fall completely within the selection box and that cross one or more of the lines of the selection box will be selected.

Layout

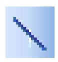
Changes your mouse cursor into a network layout tool. Right-click to change the type of element and the type of link.

- **Pressure Pipe—**Lets you place an element through which water moves under pressure. Pressure pipes typically discharge from a pumping station located upstream in the sewer collection system.
- **Conduit**—Lets you place a closed section element through which water moves. A conduit has a constant roughness and cross section shape along its entire length. Available conduit shapes consist of both open and closed cross sections.
- **Channel**—Lets you place an open-section element through which water moves. The shape of the channel is defined by the cross-section at one or both ends, so a channel does not have a constant rough-ness or cross section shape, because these attributes are interpolated along the length of the channel according to the respective values defined at the adjacent cross section nodes.
- **Gutter**—Lets you place an open-section element that models overflow. A gutter accepts the inflows that are not being taken in by an inlet because of capacity constraints. In addition, a gutter also takes in the overflow from an inlet due to flooding. There can only be one gutter downstream of any element and the gutter cannot be the only way water can leave an element, there must also be a channel or a conduit. Gutters are only used for routing; no dynamic calculations are performed for gutter elements.

Catch Basin

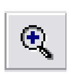
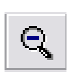
Changes your mouse cursor into a catch basin element symbol. Clicking the left mouse button while this tool is active causes a catch basin element to be placed at the location of the mouse cursor.

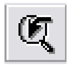
Manhole	Changes your mouse cursor into a manhole element symbol. Clicking the left mouse button while this tool is active causes a manhole element to be placed at the location of the mouse cursor.
Cross Section Node	Changes your mouse cursor into a cross section node element symbol. Clicking the left mouse button while this tool is active causes a cross section element to be placed at the location of the mouse cursor.
Junction Chamber	Changes your mouse cursor into a junction chamber element symbol. Clicking the left mouse button while this tool is active causes a junction chamber element to be placed at the location of the mouse cursor.
Pressure Junction	Changes your mouse cursor into a pressure junction element symbol. Clicking the left mouse button while this tool is active causes a pressure junction element to be placed at the location of the mouse cursor.
Pond Outlet Structure	Changes your mouse cursor into a pond outlet structure element symbol. Clicking the left mouse button while this tool is active causes a pond outlet structure element to be placed at the location of the mouse cursor.
Outfall	Changes your mouse cursor into an outfall element symbol. Clicking the left mouse button while this tool is active causes an outfall element to be placed at the location of the mouse cursor.
Wet Well	Changes your mouse cursor into a wet well element symbol. Clicking the left mouse button while this tool is active causes a wet well element to be placed at the location of the mouse cursor.
Pump	Changes your mouse cursor into a pump element symbol. Clicking the left mouse button while this tool is active causes a pump element to be placed at the location of the mouse cursor.
Catchment	Changes your mouse cursor into a catchment element symbol. When this tool is active, click in the drawing pane to begin drawing a polygon that represents the catchment.

	**Pond**	Changes your mouse cursor into a pond element symbol. When this tool is active, click in the drawing pane to begin drawing a polygon that represents the pond.
	**Border**	Changes your mouse cursor into a border symbol. When the border tool is active, you can draw a simple box in the drawing pane using the mouse. For example, you might want to draw a border around the entire model.
	**Text**	Changes your mouse cursor into a text symbol. When the text tool is active, you can add simple text to your model. Click anywhere in the drawing pane to display the Text Editor dialog box, which lets you enter text to be displayed in your model.
	**Line**	Changes your mouse cursor into a line symbol. When this tool is active, you can draw lines and polygons in your model using the mouse.

Zoom Toolbar

The Zoom toolbar provides access to the zooming and panning tools. It contains the following buttons:

	**Zoom Extents**	Sets the view so that the entire model is visible in the drawing pane.
	**Zoom In**	Magnifies the current view in the drawing pane.
	**Zoom Out**	Reduces the current view in the drawing pane.
	**Zoom Realtime**	Enables the realtime zoom tool, which lets you zoom in and out by moving the mouse while the left mouse button is depressed.

	**Zoom Center**	Opens the Zoom Center dialog box, which lets you enter drawing coordinates that are centered in the drawing pane.
	**Zoom Previous**	Returns the zoom level to the most recent previous setting.
	**Pan**	Activates the Pan tool, which lets you move the model within the drawing pane. When you select this command, the cursor changes to a hand, indicating that you can click and hold the left mouse button and move the mouse to move the drawing.
	**Refresh Drawing**	Updates the main window view according to the latest information contained in the Bentley SewerGEMS V8*i* datastore.

Customizing the Toolbars

You can customize Bentley SewerGEMS V8*i* toolbars in any of the following ways:

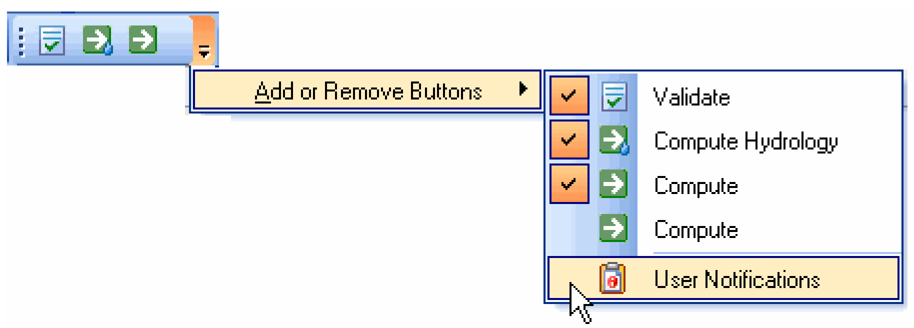
- "Adding and Removing Toolbar Buttons" on page 2-38
- "Controlling Toolbars" on page 2-38

Adding and Removing Toolbar Buttons

Toolbar buttons represent Bentley SewerGEMS V8*i* menu commands. You can remove buttons from any toolbar, and add commands to any toolbar on the Commands tab of the Customize dialog box.

To add or remove a button from a toolbar:

1. Click the down arrow on the end of the toolbar you want to customize. A series of submenus appear, allowing you to select or deselect any button in that toolbar.
2. Click **Add or Remove Buttons** then move the mouse cursor to the right until all of the submenus appear, as shown in the following figure:



3. Click the space to left of the toolbar button you want to add. A check mark appears in the submenu and the button appears in the toolbar.

   or

   Click the check mark next to the toolbar button you want to remove. The button will no longer appear in the toolbar.

Controlling Toolbars

You can control toolbars in Bentley SewerGEMS V8*i* on the Toolbars tab of the Customize dialog box. You can turn toolbars on and off, or move the toolbar to a different location in the workspace.

To turn toolbars off:

Click **View** > **Toolbars**, then click the check mark next to the toolbar you want to turn off.

To turn toolbars on:

Click **View** > **Toolbars**, then click in the space to the left of the toolbar you want to turn on.

To move a toolbar to a different location in the workspace:

Move your mouse to the vertical dotted line on the left side of any toolbar, then drag the toolbar to the desired location. If you move a toolbar away from the other toolbar, the toolbar becomes a floating dialog box.

Dynamic Manager Display

You access most of the features in Bentley SewerGEMS V8*i* through a system of dynamic windows called managers. For example, the look of the elements is controlled in the Element Symbology manager while animation is controlled in the EPS Results Browser manager.

When you first start Bentley SewerGEMS V8*i*, only two managers are displayed: the Element Symbology and Background Layers managers. This is the default workspace. You can display as many managers as you want and move them to any location in the Bentley SewerGEMS V8*i* workspace.

To return to the default workspace:

Click **View** > **Reset Workspace**.

- If you return to the default workspace, the next time you start Bentley Sewer-GEMS V8*i*, you will lose any customizations you might have made to the dynamic manager display.

Opening Managers

To open a manager:

1. Do one of the following:
   – Select the desired manager from the View menu.

– Click a manager's button on one of the toolbars.

– Press the keyboard shortcut for the desired manager.

2. If the manager is not already docked, you can drag it to the top, left- or right-side, or bottom of the SewerGEMS V8*i* window to dock it. For more information on docking managers, see "Customizing Managers" on page 2-42.

Bentley SewerGEMS V8*i* Managers

The following table lists all the Bentley SewerGEMS V8*i* managers, their toolbar buttons, and keyboard shortcuts.

Toolbar Button	Manager	Keyboard Shortcut
	**Scenarios**—lets you build a model run from alternatives. For more information, see "Scenario Manager" on page 9-469.	ALT+1
	**Alternatives**—lets you create and manage alternatives. For more information, see "Alternative Manager" on page 9-473.	ALT+2
	**Calculation Options**—lets you set parameters for the numerical engine. For more information, see "Calculation Options Manager" on page 8-431.	ALT+3
	**Element Symbology**—controls how elements look and what attributes are displayed. For more information, see "Element Symbology Manager" on page 10-538.	CTRL+1
	**Background Layers**—lets you control the display of background layers. For more information, see "Background Layer Manager" on page 10-529.	CTRL+2
	**Selection Sets**—lets you create and manage selection sets. For more information, see "Selection Sets Manager" on page 6-274.	CTRL+3
	**Network Navigator**—helps you find nodes in your model. For more information, see "Using the Network Navigator" on page 6-281.	CTRL+4

Toolbar Button	Manager	Keyboard Shortcut
	**Queries**—lets you create SQL expressions for use with selection sets and FlexTables For more information, see "Using Queries" on page 6-327.	CTRL+5
	**Prototypes**—lets you create and manage prototypes. For more information, see "Using Prototypes" on page 6-285.	CTRL+6
	**FlexTables**—lets you display and edit tables of elements. For more information, see "FlexTables Manager" on page 10-557.	CTRL+7
	**Graphs**—lets you create and manage graphs. For more information, see "Graph Manager" on page 10-577.	CTRL+8
	**Profiles** —lets you draw profiles of parts of your network. For more information, see "Profiles Manager" on page 10-549.	CTRL+9
	**Property Editor**—displays properties of individual elements or managers. For more information, see "Property Editor" on page 6-265.	F4
	**EPS Results Browser**—controls animated displays. For more information, see "Animating Profiles" on page 10-550.	F7
	**User Notifications**—presents error and warning messages resulting from a calculation.	F8

Note: Although the toolbar button for this manager does not appear by default, you can add it to the Compute toolbar.

Customizing Managers

When you first start Bentley SewerGEMS V8*i*, you will see the default workspace, in which a limited set of dockable managers are visible. You can decide which managers will be displayed at any time and where they will be displayed. You can also return to the default workspace any time.

There are four states for each manager:

Floating—A floating manager sits above the Bentley SewerGEMS V8*i* workspace like a dialog box. You can drag a floating manager anywhere and continue to work.

You can also:

- Resize a floating manager by dragging its edges.
- Close a floating manager by clicking on the x in the top right-hand corner of the title bar.
- Change the properties of the manager by right-clicking on the title bar.
- Switch between multiple floating managers in the same location by clicking the manager’s tab.
- Dock the manager by double-clicking the title bar.

Docked static—A docked static manager attaches to any of the four sides of the Bentley SewerGEMS V8*i* window. If you drag a floating manager to any of the four sides of the Bentley SewerGEMS V8*i* window, the manager will attach or dock itself to that side of the window. The manager will stay in that location unless you close it or make it dynamic. A vertical pushpin in the manager’s title bar indicates its static state; click the pushpin to change the manager’s state to dynamic. When the push pin is pointing downward (vertical push pin), the manager is docked.

You can also:

- Close a docked manager by left clicking on the x in the upper right corner of the title bar.
- Change a docked manager into a floating manager by double-clicking the title bar, or by dragging the manager to the desired location (for example, away from the side of the Bentley SewerGEMS V8*i* window).
- Change a static docked manager into a dynamically docked manager by clicking the push pin in the title bar.
- Switch between multiple docked managers in the same location by clicking the manager’s tab.

Docked dynamic—A docked dynamic manager also docks to any of the four sides of the Bentley SewerGEMS V8*i* window, but remains hidden except for a single tab. Show a docked dynamic manager by moving the mouse over the tab, or by clicking the tab. When the manager is showing (not hidden), a horizontal pushpin in its title bar indicates its dynamic state.

You can also:

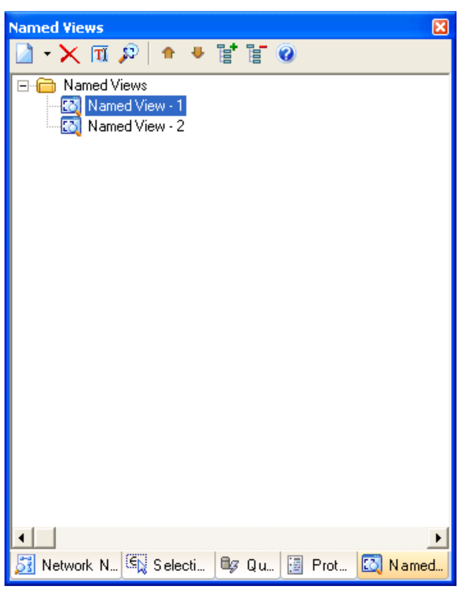
- Close a docked manager by left clicking on the x in the upper right corner of the title bar.
- Change a docked dynamic manager into a docked static manager by clicking the push pin (converting it from vertical to horizontal).
- Switch between multiple docked managers in the same location by moving the mouse over the manager’s tab or by clicking the manager’s tab.

Closed—When a manager is closed, you cannot view it. Close a manager by clicking the x in the right corner of the manager’s title bar. Open a manager by selecting the manager from the View menu (for example, View > Element Symbology), or by selecting the button for that manager on the appropriate toolbar.

Using Named Views

The Named View dialog box is where you can store the current views X and Y coordinates. When you set a view in the drawing pane and add a named view, the current view is saved as the named view. You can then center the drawing pane on the named view with the **Go To View** command.

Choose View > Named Views to open the Named View dialog box.



The toolbar contains the following controls:

New	Contains the following commands: • **Named View**—Opens a Named View Properties box to create a new named view. • **Folder**—Opens a Named Views Folder Properties box to enter a label for the new folder.
Delete	Deletes the named view or folder that is currently selected.
Rename	Rename the currently selected named view or folder.
Go to View	Centers the drawing pane on the named view.
Shift Up and Shift Down	Moves the selected named view or folder up or down.
Expand All or Collapse All	Expands or collapses the named views and folders.
Help	Displays online help for Named Views.

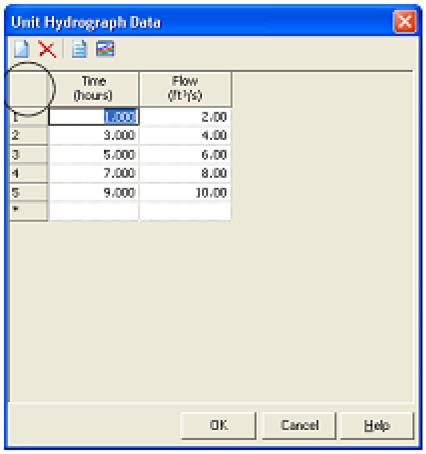
Copying and Pasting Data To and From Tables

This topic describes the best practices used to copy and paste data from and to the various tables.

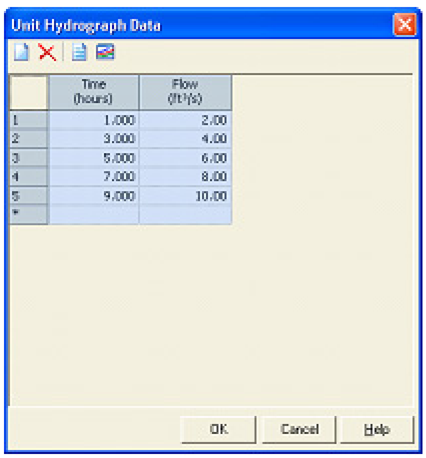
Copying data from a table

There are generally 3 ways to copy data from tables.

1. The first is to highlight (or select) the data by clicking in the top-left corner of the table.



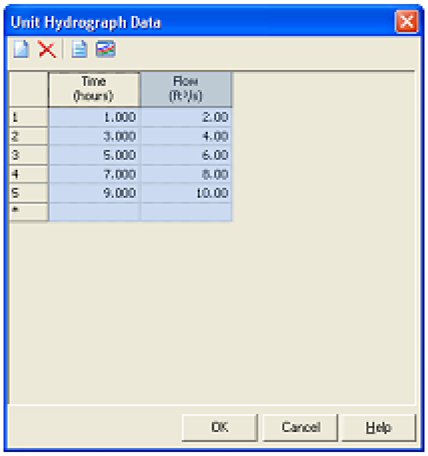
This will highlight all the data in the table including the column headers:



When you use the windows short combination, CTRL-C, it will copy the highlighted data to the windows clipboard. The data copied will include the column headers (in this case Time (hours) and Flow (ft^3/s)) and the rows below it. This also includes the last row which is blank.

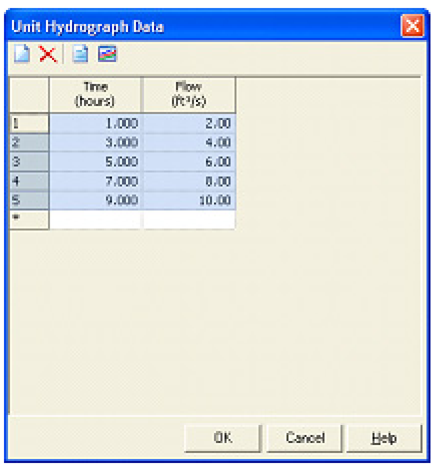
It will not copy the row headers (numbered 1 - 5 in this case).

2. The second approach is to highlight the two columns (in this case).

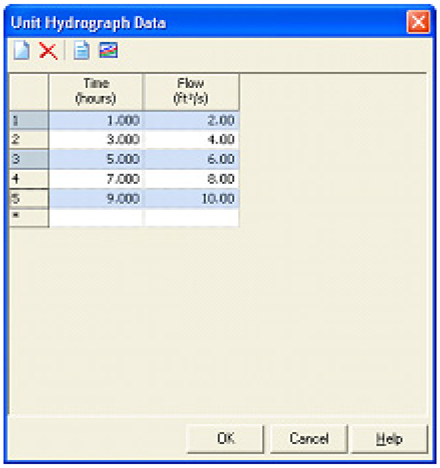


This is similar to the first approach except it does not highlight the row headers. When you use the CTRL-C combination it will again copy the header data along with the row data including the last blank row.

3. The best approach for copying the data is to highlight just the rows you want to copy without highlighting the column headers. To do this you can just click on the first row (#1 here) and drag your mouse down while holding the left mouse button. This will highlight the table as follows:



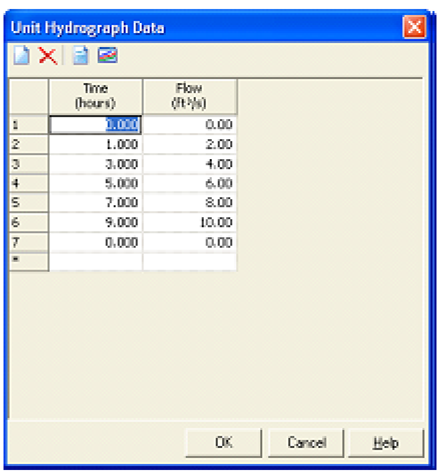
When you use the CTRL-C combination in this case it will copy only the data that you want. The column and row headers will not be copied. You can also use the CTRL-<left click> approach to copy non-consecutive rows.



Pasting data into a table

When you paste data into one of our tables you want to make sure you do not include any header data. If you include any non-numerical data a row will be inserted and the default values for the columns will be used in place of any text or non-numerical data that was pasted.

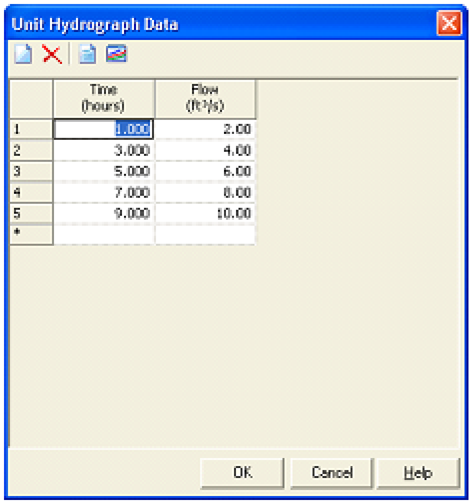
For example, if you copy using the first approach described above, you will get the following results:



The first row is the row that represents the column headers that were pasted into the table. The last row of 0's is the blank row at the end of the table.

Note: We advise that you delete any rows containing "0" values that may be inadvertently created during a copy-paste operation.

If you used the 3rd approach described above by just highlighting the rows you want to copy and then paste, you will get a more desirable result as follows:



As a result, the first row is exactly what you wanted along with the last row. There are no unexpected values pasted into the grid.

Chapter

Quick Start Lessons 3

Overview

The Quick Start lessons give you hands-on experience with many of the features and capabilities of SewerGEMS V8*i*. These detailed lessons will help you get started exploring and using the software.

Another way to become acquainted with Bentley SewerGEMS V8*i* is to run and experiment with the included sample files, located in the **C:/Program Files/Bentley/SewerGEMS/Sample** directory. To open one of these existing examples:

From the Welcome to Bentley SewerGEMS V8*i* dialog, click the Open Existing project button and browse to the **C:/Program Files/Bentley/SewerGEMS/Sample** directory. Highlight a sample file and click Open.

If the Welcome to Bentley SewerGEMS V8*i* dialog is not open, click the File pull-down menu and select the Open command. Then browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Sample** directory. Highlight a sample file and click Open.

Remember, you can right-click or press the F1 key to access the context-sensitive online help at any time.

Lesson 1: Overview of the SewerGEMS V8*i* Workspace

SewerGEMS has an extensively customizable user interface. In this lesson you will learn how to personalize SewerGEMS's toolbars, managers, and other interface elements. The following lessons guide you through the process of setting up the SewerGEMS interface in an alternate configuration, starting from the default configuration that is in use when the software is first installed. The advantage of the alternate configuration is that it increases the size of the drawing pane.

To return the interface to the default component placement settings, click **View** > **Reset Workspace**.

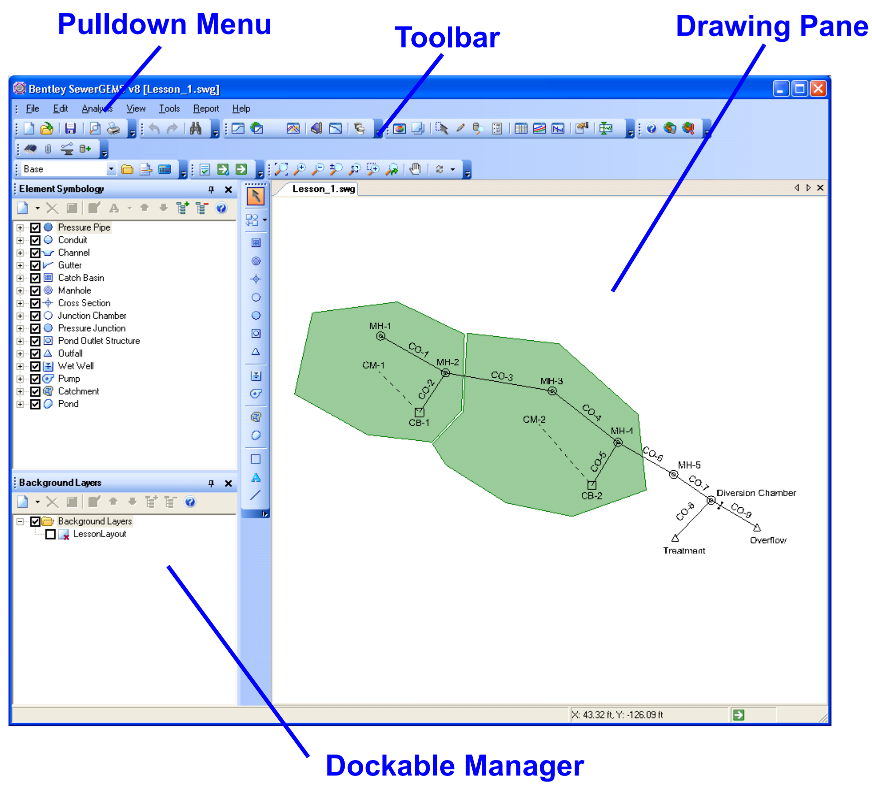
Part 1: Workspace Components Overview

The SewerGEMS workspace contains three different type of components: the drawing pane, toolbars, and dockable managers. The behavior and use of these components is discussed in the following parts of this lesson. When starting Bentley SewerGEMS V8*i* for the first time, the default interface settings will be used.

Let's begin by starting up Bentley SewerGEMS V8*i*. If SewerGEMS V8*i* is already open, skip step one and click the **File** pulldown menu and select the **Open** command instead, then proceed to step two.

1. In the Welcome to Bentley SewerGEMS V8*i* dialog that appears, click the **Open Existing Project** button.
2. Browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_1.swg**, then click **Open**.

The example network will appear in the drawing pane, and the interface will look like this:



Part 2: Working With the Drawing Pane

The drawing pane is where you create and view your model. You can pan and zoom your model in this pane.

If you have already completed Part 1 of this lesson, proceed to the Panning section below. Otherwise, complete the following steps before proceeding to the Panning section:

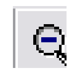
1. Start Bentley SewerGEMS V8*i*.
2. In the Welcome to Bentley SewerGEMS V8*i* dialog that appears, click the **Open Existing Project** button.
3. Browse to the **C:/Program Files/Bentley/SewerGEMS/Lessons** folder, highlight **Lesson_1.swg**, then click **Open**.

Panning

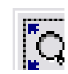
You can change the position of the drawing view in the network by using the Pan tool. Alternatively, if your mouse is equipped with a mousewheel, you can pan by simply holding down the mousewheel and moving the mouse to reposition the current view. This changes the mouse cursor to the Pan icon, allowing you to reposition the current view in the same manner—click on the drawing, hold down the mouse button, and move the mouse to reposition the current view.

Zooming

You can perform a number of zooming operations in the drawing pane to change the current view of your model.

The simple Zoom In and Zoom Out commands let you increase or decrease, respectively, the zoom level of the current view by one step per mouse click.



The Zoom Extents command automatically changes the zoom level so that the entire network is displayed in the drawing pane.

The Zoom Realtime command lets you dynamically scale up and down the zoom level. The zoom level is defined by the magnitude of mouse movement while the tool is active.

The Zoom Center command opens the Zoom Center dialog, allowing you to center the drawing pane view on the coordinates you enter, at the zoom level you specify in the Zoom menu.

Click the **Zoom Center** button, and enter **75** in the **X** field, **30** in the **Y** field, and choose **200** in the **Zoom** menu, then click **OK**. The drawing pane will center on the manhole element MH-5 at a magnification of 200%.

Part 3: Working With Toolbars

Toolbars provide efficient access to frequently used commands. All of the toolbars in SewerGEMS are customizable both in terms of the buttons that they display and in the position that they occupy within the interface.

If you have already completed Part 2 of this lesson, proceed to the Adding and Removing Toolbar Buttons section below. Otherwise, complete the following steps before proceeding to the following section:

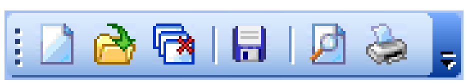
1. Start Bentley SewerGEMS V8*i*.
2. In the Welcome to Bentley SewerGEMS V8*i* dialog that appears, click the **Open Existing Project** button.
3. Browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_1.swg**, then click **Open**.

Adding and Removing Toolbar buttons

You can add and remove buttons to/from toolbars to suit your preference.

Let's begin by adding a button to the File toolbar:

1. Click the arrow button that is located on the far right of the File toolbar (the one with New, Open, Save, etc.) and move the mouse cursor over the Add or Remove Buttons command that appears.
2. In the shortcut menu that appears, move the mouse cursor over the Close All command.
3. Your toolbar should now look like this:



   Now let's remove a button from the Zoom toolbar.

4. Click the arrow button that is located on the far right of the Zoom toolbar (the Zoom toolbar is the one with the zoom and pan buttons) and move the mouse cursor over the Add or Remove Buttons command that appears.
5. Note that in the submenu that appears, there is a menu item for each of the buttons available to the Zoom toolbar. Most of the items have a checked box next to them, indicating that they are all visible on the toolbar. Click the Zoom Realtime menu item.
6. The Zoom Realtime button disappears from the toolbar, and the corresponding menu item loses its check.

Your toolbar should now look like this:

Repositioning Toolbars

You can reposition any toolbar so that the toolbar is floating (not docked, or attached, to the toolbar areas of the interface), and you can move any toolbar to a new docked location.

Reposition the Layout toolbar (the Layout toolbar is the one along the left side of the interface that contains all of the element layout tools) so that it is situated horizontally underneath the other toolbars.

1. Hover the mouse cursor over the top edge of the toolbar, in the “textured” area indicated by a line of dots.
2. When the mouse cursor changes to a four-directional arrow icon, click and hold down the mouse button.
3. Drag the mouse to the area beneath the set of horizontal toolbars along the top of the interface and release the mouse button.

Part 4: Working With Dockable Manager Components

By default, the dockable managers occupy the majority of the interface. They are called dockable managers because they can be docked, or attached, to the edges of the Bentley SewerGEMS V8*i* window. You can position any of the managers along the top, bottom, left, or right of the Bentley SewerGEMS V8*i* window, and when more than one manager is situated in the same spot, the docked pane becomes tabbed, allowing you to switch between all of the managers that are docked in the same area.

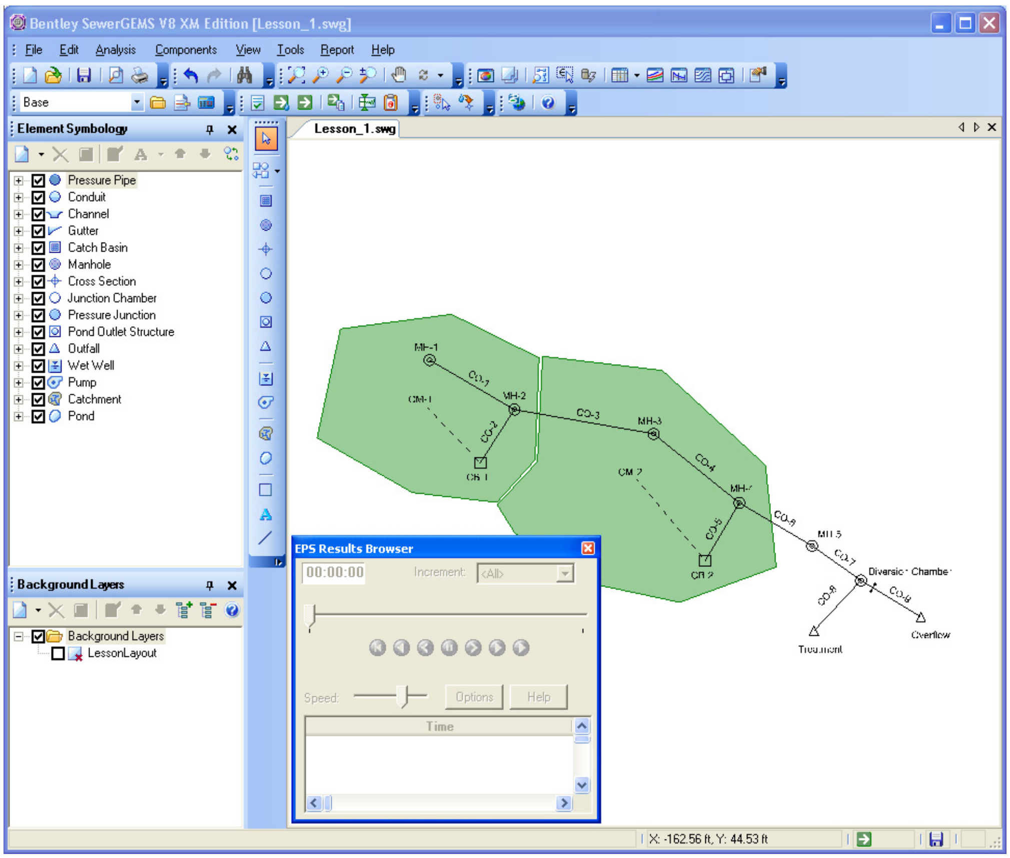
If you have already completed Part 3 of this lesson, proceed to the paragraph starting “In this part of the lesson...” below. Otherwise, complete the following steps before proceeding to the following section:

1. Start Bentley SewerGEMS V8*i*.
2. In the Welcome to Bentley SewerGEMS V8*i* dialog that appears, click the **Open Existing Project** button.

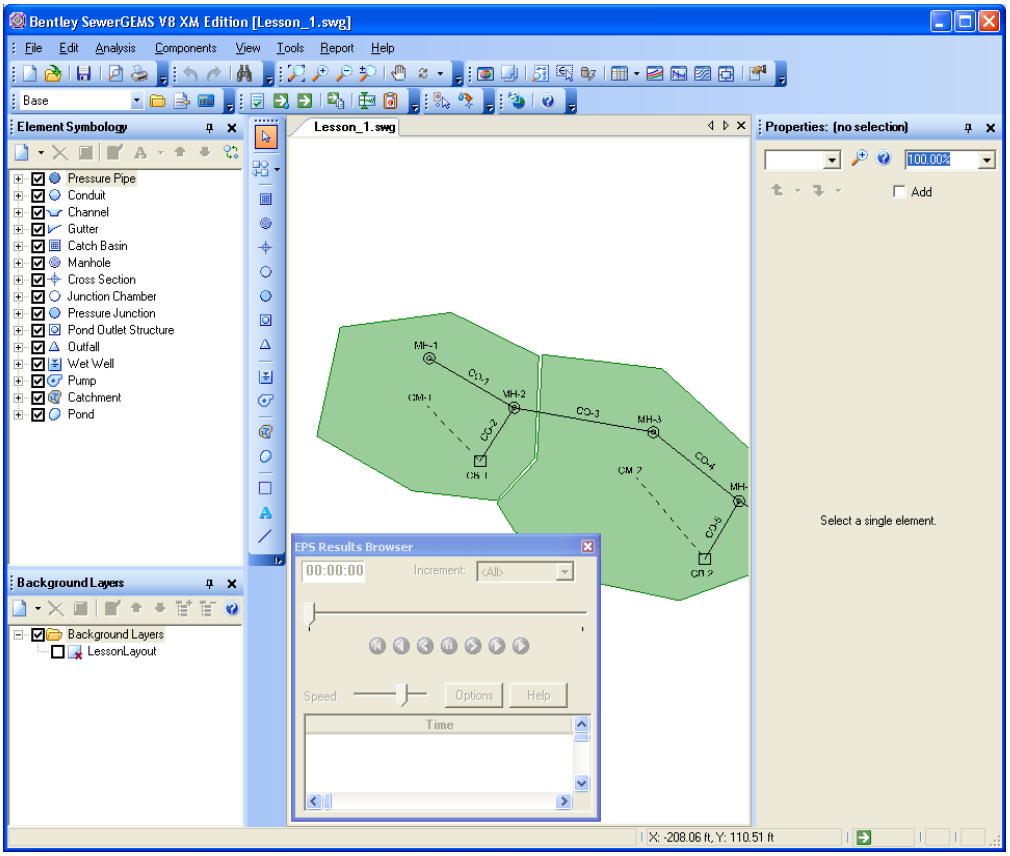
Browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_1.swg**, then click **Open**.

In this part of the lesson, we will create an alternate interface setup that makes certain frequently used interface components more accessible. Specifically, we will add the EPS Results Browser dialog as a floating window, and add the Property Editor to the interface as a dockable manager.

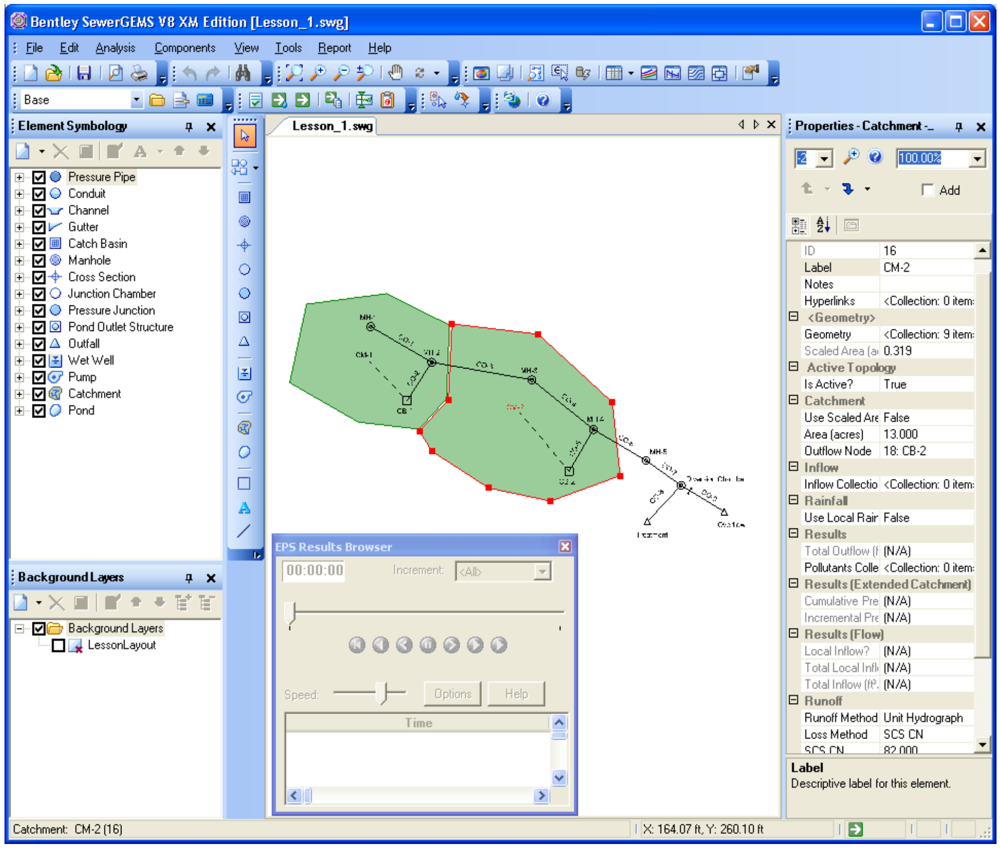
1. Open the EPS Results Browser dialog box. Click **Analysis > EPS Results Browser**.
2. Click the EPS Results Browser tab and hold the mouse, and drag the window, represented by a grey outline, to the bottom left corner of the drawing pane.
3. Hover the mouse cursor over the bottom right corner of the EPS Results Browser dialog box, until the mouse cursor turns into diagonal two-headed arrow and resize the dialog box until it is as small as possible while still accommodating all of the controls, like this:



4. Open the Property Editor. Click **View > Properties**.
5. Click the heading of the Property Editor, and, while holding down the mouse button, drag the mouse to the upper-right corner of the interface, just below the bottom toolbar. The dialog box is correctly positioned when the grey outline representing the dialog box occupies the length of the right side of the interface, like this:



6. Resize the width of the dockable managers on the left and the Property Editor on the right so that the drawing pane is larger, displaying more of the network at once. Hover the mouse cursor over the right and left edges, respectively, until the mouse cursor changes to the double-headed horizontal arrow cursor, then click, hold, and drag the mouse until the desired size is obtained.
7. Click the Zoom Extents button to view the entire network in the drawing pane.
8. The Property Editor will be gray, with the message Select a single element. Click on an element to see the attributes associated with it in the Property Editor.
9. Your interface should now look like this:



When the managers are docked, they can be in static or dynamic mode. When in static mode, the managers are always visible. When a manager is in dynamic mode, it isn't visible unless the mouse cursor is hovered over the associated tab. When the mouse cursor is moved away from the tab, the manager retracts. To switch between the two, click the pushpin toggle button in the top-right corner of the manager. A horizontal pushpin indicates that the manager is in dynamic mode. When the pushpin is vertical, the manager is in static mode.

Lesson 2: Laying Out a Network

SewerGEMS is an extremely efficient tool for laying out a storm or sanitary sewer network. It is easy to prepare a schematic or scaled model and let SewerGEMS take care of the link-node connectivity.

In constructing the network for this lesson, you do not need to be concerned with assigning labels to pipes and nodes, because the software assigns labels automatically. A schematic drawing is one in which pipe lengths are entered manually, in the user-defined length field. In a scaled drawing, pipe lengths are automatically calculated from the position of the pipes' bends and start and stop nodes in the drawing pane. For the purposes of this lesson, we will build a schematic model.

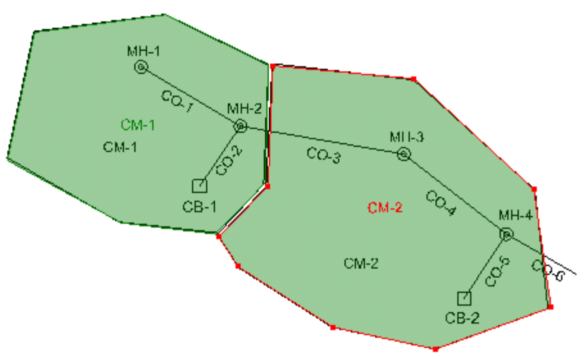
Part 1: Laying Out Catchments and Ponds

Let's begin by starting up Bentley SewerGEMS V8*i*. If you already have SewerGEMS V8*i* open from the previous lesson, click the Open button and skip ahead to step 2.

1. In the Welcome to Bentley SewerGEMS V8*i* dialog box that appears, click the **Open Existing Project** button.
2. Browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_ 2_1.swg**, then click **Open**.

   Catchments and ponds are polygon elements that graphically depict the area represented by the element. Begin by laying out the two catchments by tracing the catchment outlines shown in the .dxf background, as follows:
3. Select the **Catchment** layout tool. 
4. Click on one of the corners of the catchment outline for **CM-1**. Drag the mouse to the next corner, and click again.
5. Continue laying out the catchment boundaries by clicking each corner until you click the last one. Right-click and select **Done** from the submenu that appears.

   Repeat steps 4 and 5 for the remaining catchment. Your model should now look like this:



6. In a schematic model, where the shape isn't necessarily important, SewerGEMS V8*i* provides a shortcut for laying out catchments and ponds. Rather than drawing a polygon line-by-line, you can also use a default shape by holding down the **Ctrl**

key while clicking to define the center point of the element, then dragging the mouse to define the size and orientation of the pentagon. Clicking again places the element.

This concludes this part of the lesson. The next part of the lesson continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next lesson at a later time, you can either save the current project, or use the **Lesson_2_2.swg** project located in the **C:/ Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder.

Part 2: Laying Out Nodes and Links

The term Nodes refers to any of the available point element types:

- catch basins
- Manholes
- Cross Section Nodes
- Junction Chamber
- Pressure Junction
- Outlet Structures
- Outfalls
- Wet Wells
- Pumps

The term links refers to any of the available line elements:

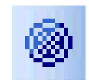
- Pressure Pipe
- Conduits
- Channels
- Gutters

In this part of the lesson, we'll begin by laying out all of the nodes, and then connect them using links.

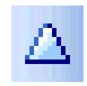
If you've already completed Part 1 of this lesson, you can continue using the same model. Otherwise, begin by clicking the Open button; then, browse to the **C:/ Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_2_2.swg**, then click **Open.**

1. Select the **Catchbasin** layout tool and click in the lower-right corner of **CM-1** to place a catch basin node there, at the spot indicated on the dxf background. Place another as indicated in the background in **CM-2**.

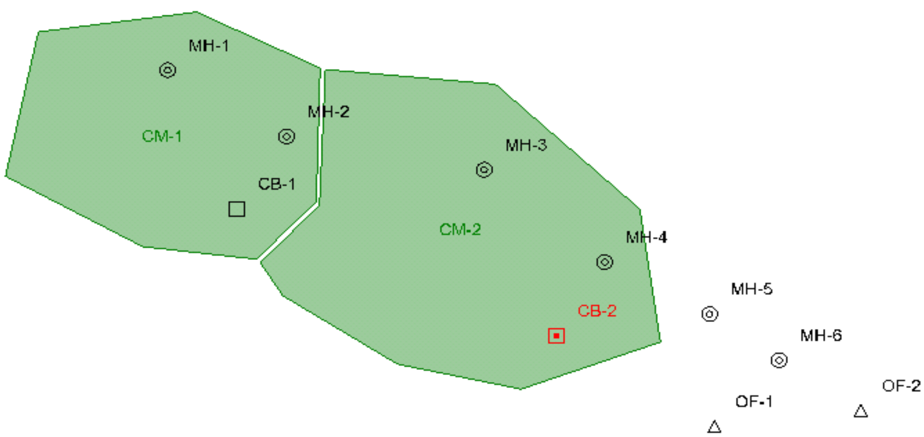
2. Select the **Manhole** layout tool and click each of the locations indicated by the dxf background, 6 in all. Note that the element labeled **Diversion Chamber** in the background is also a manhole.



3. Select the **Outfall** layout tool and click each of the locations indicated by the dxf background, 2 in all. In the dxf background, the outlets are labeled Treatment and Overflow.



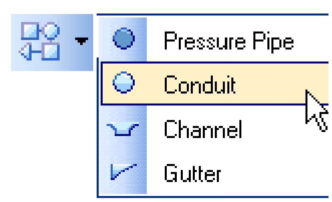
Your model should now look like this (when the dxf background is turned off - click the checkbox next to LessonLayout in the Background Layers manager to turn off the background):



Now lay out link elements to connect the nodes, as follows:

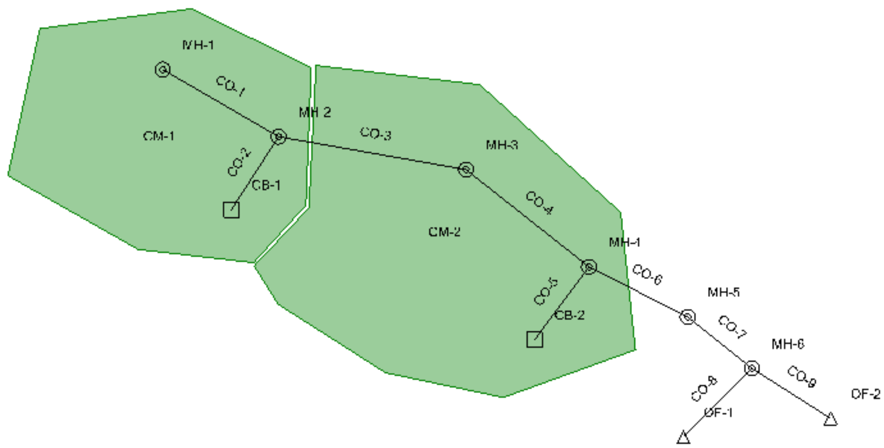
4. Click the **Layout** tool and select the **Conduit** tool from the submenu that appears.

5. Click on **MH-1** and drag the mouse to **MH-2**, and click again to lay out the conduit. Right-click and select Done from the submenu to complete the conduit.



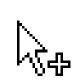
6. Click **CB-1**, drag the mouse to **MH-2,** and click to lay out a second conduit. The conduit tool is still active, so drag the mouse to **MH-3** and click again to lay out a third conduit. Drag and click on **MH-4** to lay out a fourth. Right-click and select **Done** from the submenu.
7. Click **CB-2** drag the mouse to **MH-4,** and click. Drag the mouse to **MH-5** and click again. Drag to and click on **MH-6**. Right-click and select **Done** from the submenu.
8. Click **OF-1**, drag the mouse to **MH-6**, and click again. Drag the Mouse to **OF-2** and click to lay out the final conduit. Right-click and select **Done** from the submenu.

   Your model should now look like this:



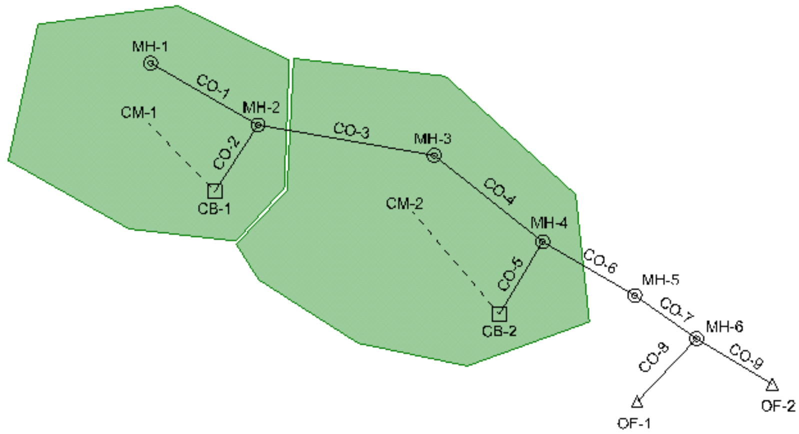
   The last step in laying out the network is establishing the connectivity between the polygon elements and the node elements.

9. Click the **View** pulldown menu and select the **Properties** command.
10. Dock the **Property Editor** dialog that appears to the right side of the Sewer-GEMS V8*i* window (For a description of how to do this, see Lesson 1, Part 4, Step 5).
11. Highlight **CM-1**. The attributes associated with the catchment will appear in the Property Editor.
12. Click the pulldown menu in the **Outflow Node** field, and choose the **<Select...>** command.
13. Your mouse cursor changes into a Pick Element tool. Click on **CB-1**.



14. Highlight **CM-2**, click the pulldown menu in the **Outflow Node** field, and choose the **<Select...>** command. Click on **CB-2**.

Your model should now look like this:



Note the dashed lines indicating the connectivity between the polygon and node elements.

This concludes this part of the lesson. The next part of the lesson continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part of the lesson at a later time, you can either save the current project, or use the **Lesson_2_3.swg** project located in the **C:/Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder.

Part 3: Moving Element Labels

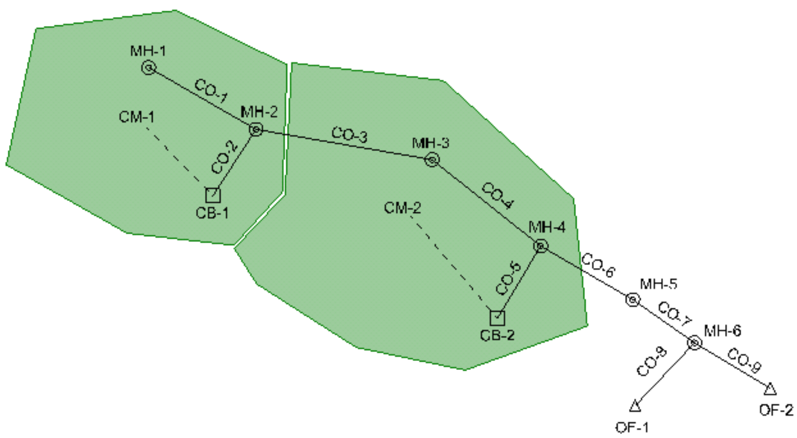
If you've already completed Part 2 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson _2_3.swg**, then click **Open.**

As you can see, the default placement of the element labels can sometimes interfere with the visibility of other elements and labels. You can manually move the labels by clicking on them until just the label is highlighted. Note that when an element is highlighted, the label is highlighted as well. To move the label, make sure that only the label is highlighted.

When the label is highlighted, you will see a small square, or “grip”, near the label, as shown below:

Click on this grip and move the label to the new position, such that it doesn’t interfere with the visibility of other labels and elements. Repeat this process with the other element labels in the model.

When you have done so, the model should look like this:



By default, labels for link elements are positioned so that they are placed at an angle parallel to the link. You can change the angle of orientation by right-clicking a label and selecting the **Rotate** command from the submenu that appears. By moving your mouse up and down while this command is active, you can rotate the label angle around a pivot point. When you are finished rotating the label, right-click and select **Done** from the submenu that appears.

This concludes Lesson 2. The next lesson continues where this one leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next lesson at a later time, you can either save the current project or use the **Lesson_3_1.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

Lesson 3: Entering Data

Input data in SewerGEMS V8*i* can be divided into two categories: element input data and global project data. This lesson explains the difference between the two and describes how to define both types of data.

Part 1: Entering Element Input Data

Element input data refers to the data associated with the elements in the model. It can be entered using the Property Editor, as described in Lesson 2, through FlexTables, or through the Alternatives Editor, as described in the “ Editing Alternatives” topic.

Each method offers different advantages:

- The Property Editor is easily accessible and can be positioned next to the drawing pane so you can see the visual context of the element whose data is being modified.
- The FlexTables are categorized according to element type, so they are best suited for entering data for large groups of elements at once. They also provide global editing and filtering functionality to allow you to enter data common to a large number of elements quickly and easily.
- The Alternatives Editor allows you to use data inheritance functionality (discussed in “ Editing Alternatives”) and the categorized nature of the data (alternatives are grouped according to the type of data they contain, such as physical data, inflow data, hydrologic data, etc.), can be useful.

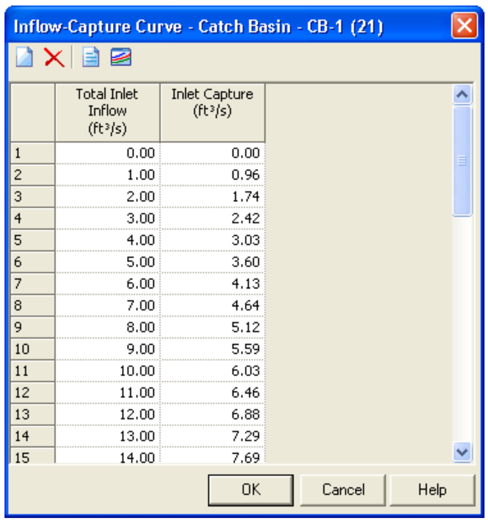
Let’s begin by using the Property Editor to define the attributes for the individual elements in the model.

If you’ve already completed Lesson 2, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button, then browse to the **C:/Program Files/ Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_3_1.swg**, then click **Open.**

1. Highlight **CB-1**.
2. The fields that are available for a given element type vary depending on the settings in other fields. Change the **Structure Shape Type** to **Rectangular Structure**. Note that the **Diameter** field disappears, and a **Length** field and a **Width** field appear in its place.
3. Change the **Inlet Type** to **Inflow-Capture Curve**.
4. Click the ellipsis button in the **Inflow-Capture Curve** field to open the **Inflow Capture Curve** dialog.

5. Using Windows Explorer, browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder and open the text document entitled **CB Inlet Data.txt**. Highlight all of the data in the text file, and press **Ctrl+C** on your keyboard to copy the data.

6. Back in the **Inflow-Capture Curve** dialog, press **Ctrl+V** to copy the data into the dialog. (If the copy command doesn't work immediately, click the first column heading, hold down the mouse button, and drag the cursor over to the second column to highlight both columns, then press **Ctrl+V**).



7. Click **OK** in the **Inflow-Capture Curve** dialog to close it.

8. Enter the following data in the specified fields:

Table 3-1: CB-1 Attribute Values

Field Name	Value
Ground Elevation	106.00 ft.
Invert Elevation	102.00 ft.

9. Highlight **CB-2**.

10. Change the **Structure Shape Type** to **Rectangular Structure**, leaving the default values for **Length** and **Width** at **3.00 ft.**

11. Change the **Inlet Type** to **Inflow-Capture Curve**.

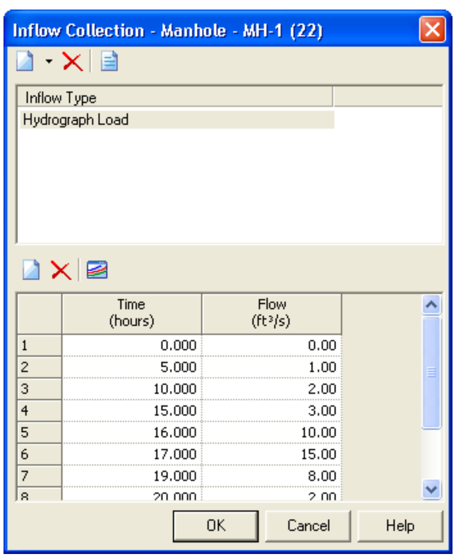
12. Click the ellipsis button in the **Inflow-Capture Curve** field to open the **Inflow Capture Curve** dialog. Press **Ctrl+V** to copy the data you copied during step 5 into the dialog.

13. Enter the following data in the specified fields:

Table 3-2: CB-2 Attribute Values

Field Name	Value
Ground Elevation	101.00 ft.
Invert Elevation	97.00 ft.

14. Highlight **MH-1**.
15. Enter a local inflow (local inflow is inflow that occurs at a specific element, as opposed to the inflow from global storm events, which is applied over the whole model). Click the ellipsis button in the **Inflow Collection** field.
16. In the **Inflow Collection** dialog that appears, click the **New** button, then click **Hydrograph Inflow** in the submenu that appears.
17. Using Windows Explorer, browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder and open the text document entitled **MH1 Inflow Collection.txt**. Highlight all of the data in the text file, and press **Ctrl+C** on your keyboard to copy the data.
18. Back in the **Inflow-Collection** dialog, press **Ctrl+V** to copy the data into the dialog. (If the copy command doesn't work immediately, click the first column heading, hold down the mouse button, and drag the cursor over to the second column to highlight both columns, then press **Ctrl+V**).



19. Click **OK** in the **Inflow Collection** dialog to close it.

20. Enter the following data in the specified fields:

Table 3-3: MH-1 Attribute Values

Field Name	Value
Ground Elevation	107.00 ft.
Invert Elevation	103.00 ft.
Diameter	3.00 ft.

For the other elements in the model, of which there are more than one of each type, let's enter the data using FlexTables.

21. Click the **View** pulldown menu and select the **FlexTables** command.
22. Under the **Tables - Predefined** node, double-click the **Manhole Table**.
23. Enter the following data in the **Manhole Table** dialog that appears:

Table 3-4: Manhole Attributes

Element	Ground Elevation	Invert Elevation	Diameter)
MH-2	105.00 ft	101.00 ft	3.00 ft.
MH-3	103.00 ft	99.00 ft	3.00 ft.
MH-4	100.00 ft	96.00 ft	3.00 ft.
MH-5	99.00 ft	95.00 ft	3.00 ft.
MH-6	97.00 ft	93.00 ft	3.00 ft.

24. Change the **Label** for **MH-6** to **Diversion Chamber**. When you have entered the data, close the **Manhole Table**. In the **FlexTables** dialog, double-click the **Catchment Table** node.

25. Enter the following data in the **Catchment Table** dialog that appears (if one or more attributes does not appear in the FlexTable by default, you will have to add them to the FlexTable. For instructions on adding attributes to a FlexTable, see Lesson 1 Part 2):

Table 3-5: Catchment Attributes

Element	Area (acres)	SCS CN	Tc (hours)
CM-1	10.000	80.000	0.400
CM-2	13.000	82.000	0.500

26. When you have entered the data, close the **Catchment Table**. In the **FlexTables** dialog, double-click the **Conduit Table** node.

27. Enter the following data in the **Conduit Table** that appears (note that the stop and start inverts are defined by the values of the adjacent nodes, since the **Set Invert To Start/Stop Node** fields are set to **True** by default):

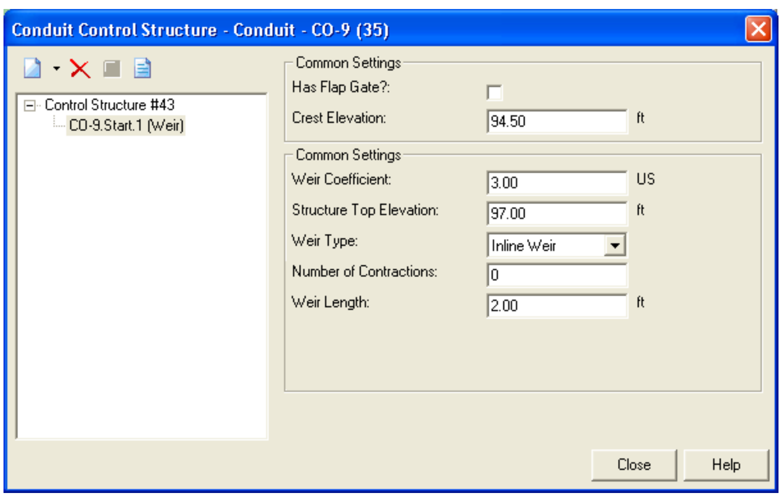
Table 3-6: Conduit Attributes

Element	Has User Defined Length	User Defined Length (ft.)	Diameter (in)	Start Invert (ft.)
CO-1	True	250.00	24.0	Automatically Assigned
CO-2	True	50.00	24.0	Automatically Assigned
CO-3	True	500.00	30.0	Automatically Assigned
CO-4	True	500.00	30.0	Automatically Assigned
CO-5	True	50.00	24.0	Automatically Assigned

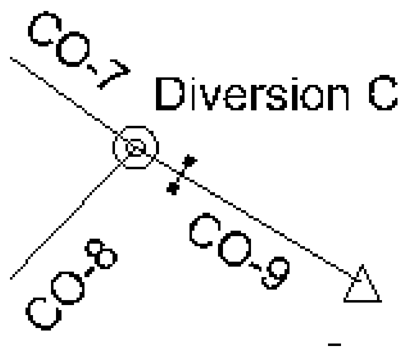
Table 3-6: Conduit Attributes

Element	Has User Defined Length	User Defined Length (ft.)	Diameter (in)	Start Invert (ft.)
CO-6	True	500.00	36.0	Automatically Assigned
CO-7	True	300.00	36.0	Automatically Assigned
CO-8	True	300.00	30.0	Automatically Assigned
CO-9	True	300.00	30.0	Automatically Assigned

28. When you have entered the data, close the **Conduit Table**.
29. Click on **CO-9** in the drawing pane.
30. In the **Properties Editor**, change the **Has Start Control Structure?** value to **True**. Click the ellipsis button in the **Start Control Structure** field.
31. In the **Conduit Control Structure** dialog that appears, click the **New** button and select **Weir** from the submenu.
32. Change the **Crest Elevation** to **94.50** ft. Change the **Structure Top Elevation** to **97.00** ft. Change the **Weir Length** to **2.00** ft. Click the **Close** button.



33. In the drawing pane, an icon appears on the upstream end of CO-9 to indicate that a control is present.



34. In the **FlexTables** dialog, double-click the **Outfall Table** node.
35. In the **Outfall Table** dialog that appears, enter the following data:

Table 3-7: Outfall Attributes

Element	Ground Elevation (ft.)	Set Invert Elevation to Ground Elevation	Invert Elevation (ft.)	Change Label To
OF-1	95.00	False	91.00	Treatment
OF-2	95.00	False	91.00	Overflow

This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_3_2.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

Part 2: Entering Global Project Data

Global project data refers to information that applies to the project as a whole. This includes project properties, storm events, and global storm events.

Defining Project Properties

Project Properties are purely informational data that store the Project Title, Engineer's Name, Company Name, and the Date, along with any notes associated with the project

This data, if entered, will be added to the footer of any of the preformatted reports that are generated by SewerGEMS V8*i*, with the exception of the data entered in the Notes field, which will not be displayed.

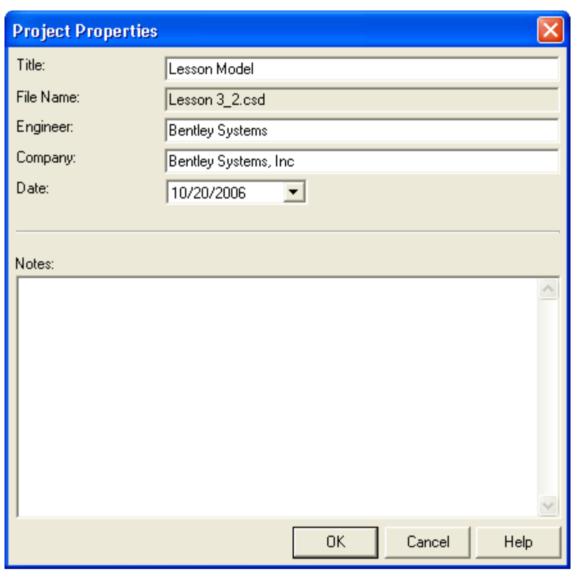
If you've already completed Part 1 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_3_2.swg**, then click **Open.**

1. Click the **File** menu and select the **Project Properties** command.
2. In the **Project Properties** dialog that appears, enter the following information in the specified fields:

Table 3-8: Project Properties

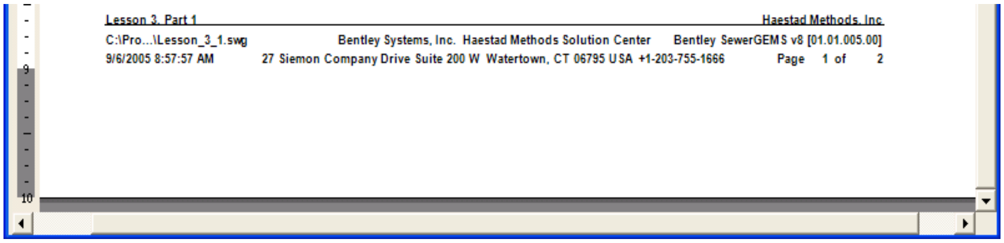
Field Name	Value
Title	Lesson Model
Engineer	<Your Name Here>
Company:	<Your Company Name Here>

3. The date is automatically entered using the information in your system calendar. The dialog should now look like this:



4. Click the **OK** button.

5. To see how the information will appear on the preformatted reports, click the **Report** pulldown menu and select the **Project Inventory** command.
6. In the **Project Inventory** report dialog, scroll to the bottom of the page to view the footer.



Defining Storm Events

A storm event is a single curve that represents one rainfall event for a given recurrence interval. Once the storm event is created it can either be used locally at a catchment, or it can be used globally, as discussed in the next section of this part of the lesson.

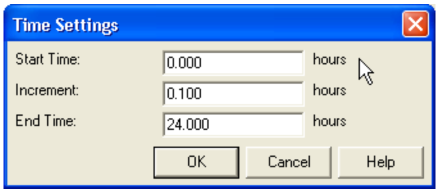
This example shows how to retrieve and paste an external rainfall data source file into SewerGEMS as a cumulative rainfall data table. See Lesson 7 for a detailed example on how to set up multiple return events from synthetic rainfall data (such as SCS Type I, IA, II, III distributions).

1. Click the **Components** pulldown menu and select the **Storm Events** command.
2. In the **Storm Events** dialog that appears, click the **New** button and select **Cumulative** from the submenu that appears.
3. In the **Time Settings** dialog that appears, enter the following data:

Table 3-9: Time Settings

Field Name	Value (hrs)
Start Time	0.000
Increment	0.100
End Time	24.000

4. The start and end times define the duration of the storm event, while the increment defines the amount of time between each ordinate when the storm is calculated. Click the **OK** button to close the **Time Settings** dialog.

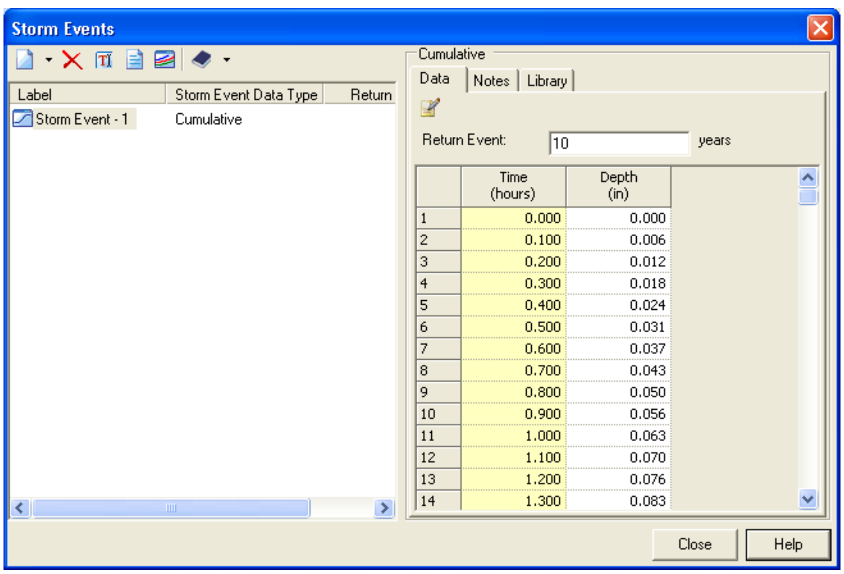


5. In the **Return Event** field, enter a value of **10** years.

6. Using Windows Explorer, browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder and open the text document entitled **Storm Event Data.txt**. Highlight all of the data in the text file, and press **Ctrl+C** on your keyboard to copy the data.

   This procedure demonstrated how you can copy storm data from a predefined storm event, created from external rainfall data sources. Lesson 6 describes a detailed example on how to set up multiple return events from synthetic rainfall data (such as SCS Type I, IA, II, III distributions)

7. Back in the **Storm Events** dialog, highlight the **Depth** column of the **Time vs. Depth** table on the right side of the dialog and press **Ctrl+V** on your keyboard to paste the data into the table.

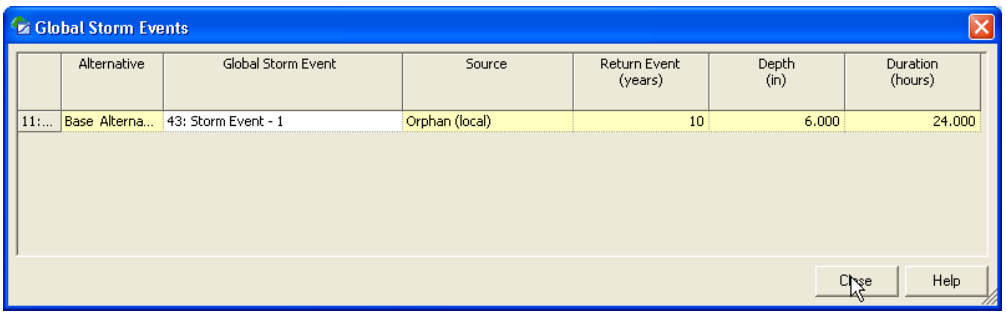


8. Click the **Close** button to close the **Storm Events** dialog.

Defining Global Storm Events

As mentioned in the last section, once a storm event is created, it can be assigned locally to one or more catchments, or assigned to the project globally. When a storm event is defined as a global event, it applies the storm event to every catchment in the scenario that does not have localized rainfall.

1. To apply the storm event we just created globally, click the **Components** pull-down menu and select the **Global Storm Events** command.
2. In the **Global Storm Event** dialog that appears, click the arrow button in the **Global Storm Event** field. Select the only event in the list, **Storm Event - 1**.
3. Note that the other fields in the dialog, colored yellow to denote their read-only status, are filled in automatically with the data associated with the selected storm event. Click the **Close** button to close the **Global Storm Events** dialog.

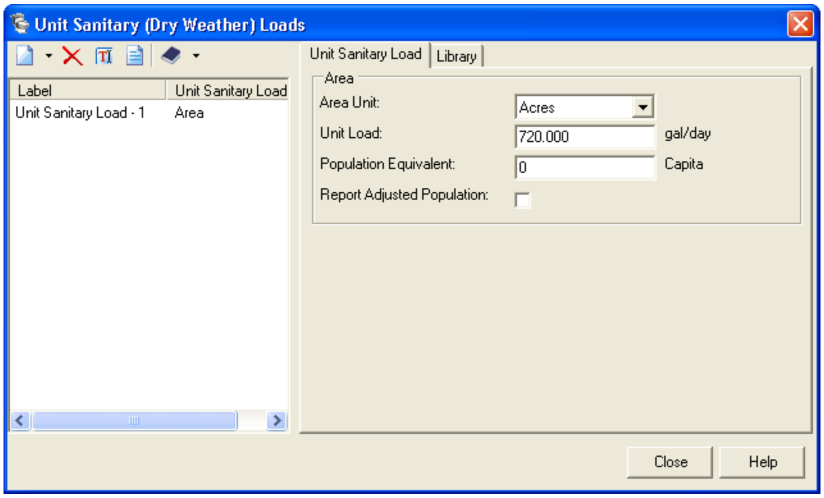


Adding Sanitary Loads

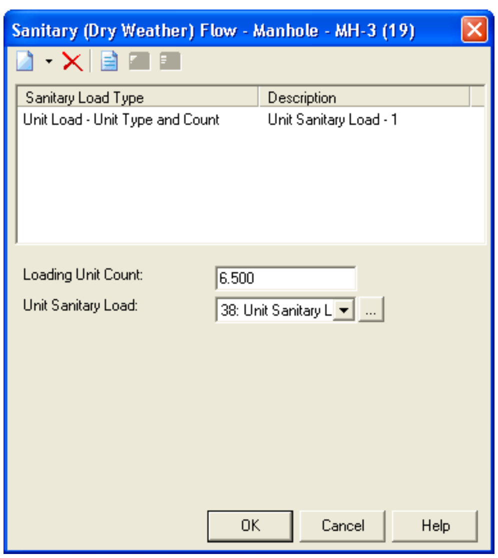
In this part of the lesson, we will set up a unit sanitary load and a pattern sanitary load. A unit sanitary load can be set up once and then used at multiple nodes, and can also be saved as an engineering library for use in other models (see "Adding Unit Sanitary (Dry Weather) Loads" on page 7-370). A pattern sanitary load is a load that varies over time.

1. To create a new unit sanitary load, click the **Components** menu and select the **Unit Sanitary (Dry Weather) Loads** command.
2. In the dialog that appears, click the **New** button and select **Area** from the submenu that appears.
3. Right-click the **gal/min** label to the right of the **Unit Load** field and select the **Units and Formatting** command from the submenu that appears.
4. In the **Set Field Options** dialog that appears, change the **Unit** value to **gal/day**. Click **OK**.

5. Back in the **Unit Sanitary Loads** dialog, enter **720** in the **Unit Load** field. Click the **Close** button.

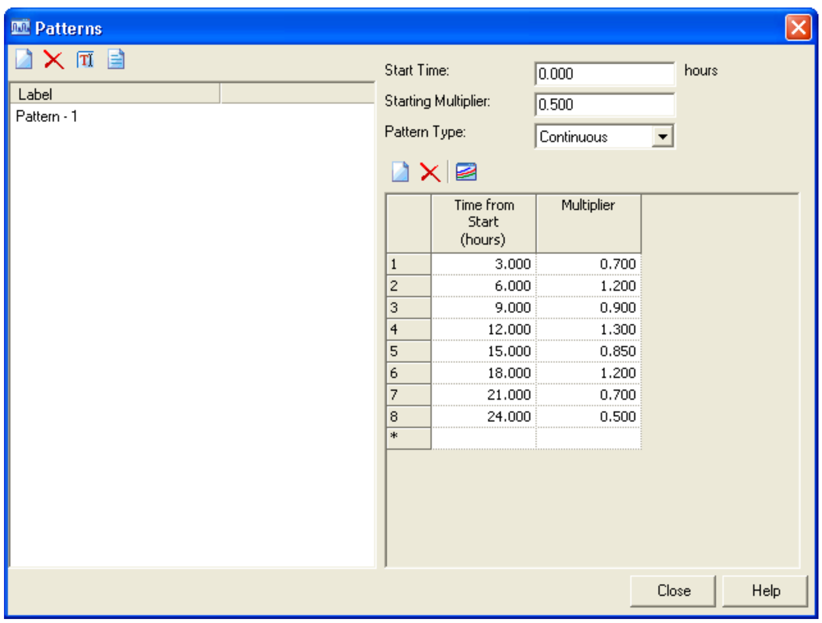


6. Highlight **MH-3** in the drawing pane.
7. In the **Properties Editor**, click the **ellipsis** button in the **Sanitary Loads** field.
8. In the **Sanitary (Dry Weather) Flow** dialog that appears, click the **New** button and select the **Unit Load - Unit Type and Count** command from the submenu that appears.
9. Enter **6.5** in the **Loading Unit Count** field.
10. Click the **Unit Sanitary Load** pulldown menu and select **Unit Sanitary Load - 1**. Click **OK**.

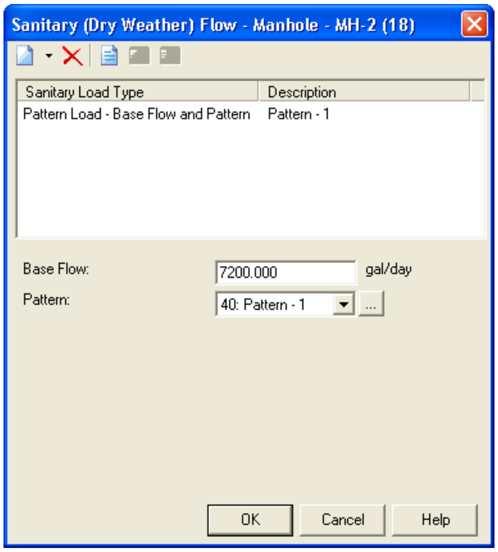


11. Highlight **MH-4** and repeat steps **7 - 10**.
12. Highlight **MH-2**.

13. In the **Properties Editor**, click the **ellipsis** button in the **Sanitary Loads** field.
14. In the **Sanitary (Dry Weather) Flow** dialog that appears, click the **New** button and select the **Pattern Load - Base Flow** and **Pattern** command from the submenu that appears.
15. Enter **7200** in the **Base Flow** field.
16. Click the **ellipsis** button next to the **Pattern** field. In the **Patterns** dialog that appears, click the **New** button. Enter **0.5** for the **Starting Multiplier**.
17. Using Windows Explorer, browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder and open the text document entitled **Sanitary Load Pattern.txt**. Highlight all of the data in the text file, and press **Ctrl+C** on your keyboard to copy the data.
18. Back in the **Patterns** dialog, click the **Time From Start** column heading, hold down the mouse button, and drag the cursor over to the **Multiplier** column to highlight both columns, then press **Ctrl+V**. Click the **Close** button.



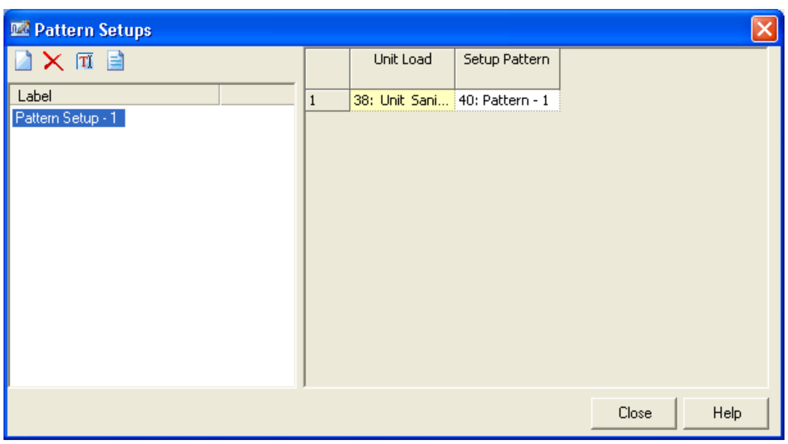
19. Back in the **Sanitary (Dry Weather) Flow** dialog, click the **Pattern** pulldown menu and select **Pattern - 1**. Click **OK**.



20. To apply the pattern to the unit load we created, you must create a Pattern Setup. Click the Components pulldown menu and select the Pattern Setups command.

21. In the Pattern Setups dialog, click the New button to create a new pattern setup.

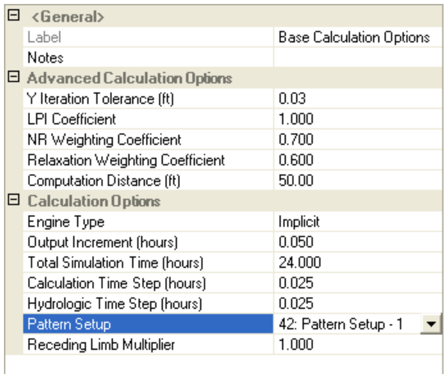
22. Click the pulldown menu in the Setup Pattern column and select Pattern - 1.



23. Click the Close button.

24. Finally, change the calculation option so that the Pattern Setup is applied. Click the Analysis pulldowm menu and select the Calculation Options command.

25. Change the Pattern Setup value to Pattern Setup - 1.



This concludes Lesson 3. The next lesson continues where this one leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next lesson at a later time, you can either save the current project or use the **Lesson_4.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

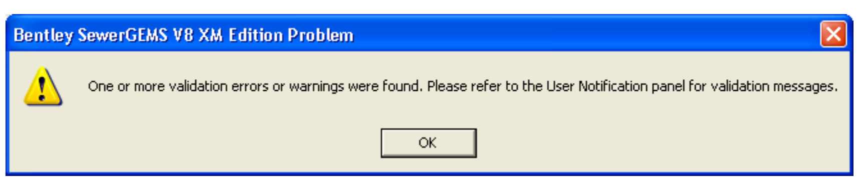
Lesson 4: Validating and Calculating a Model

In Lessons 2 and 3 we created the model and entered the required input data. The model is now ready to be calculated. When you execute the Compute command, SewerGEMS V8*i* performs a validation routine to detect any errors or input data that will interfere with the successful calculation of the model. Alternatively, you can also validate your model at any time, without calculating.

If you've already completed Lesson 3, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button, then browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_4.swg**, then click **Open.**

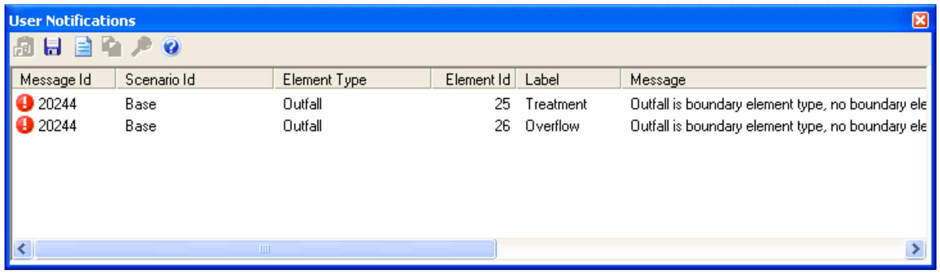
1. Click the **Validate** button in the toolbar, or click the **Analysis** menu and select the **Validate** command.

2. A Bentley SewerGEMS V8*i* **Problem** dialog appears, informing you that one or more validation errors were found. Click **OK** in this box.



3. The **User Notifications** dialog appears. This dialog lists data entry errors that prevent the model from calculating successfully. These types of errors are marked with a red icon.

4. Double-click the first message in the list, “Outfall is boundary element type, no boundary element selected.”

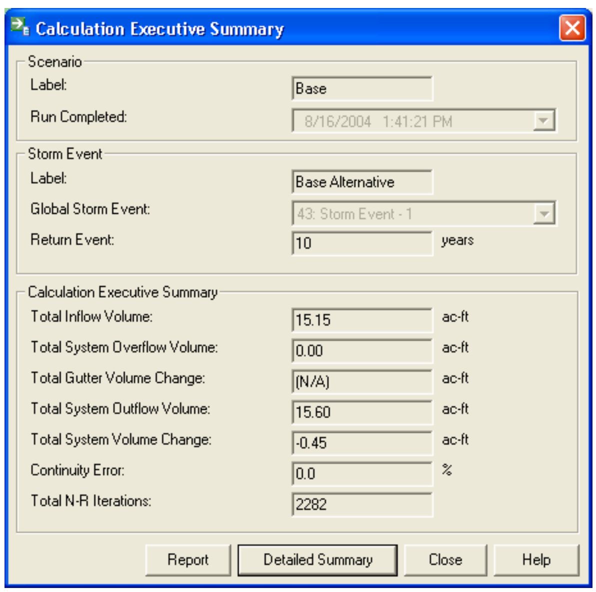


5. The drawing pane centers on the element referenced by the error message, and the element is highlighted. In the **Property Editor**, change the **Boundary Condition Type** to **Free Outfall**.

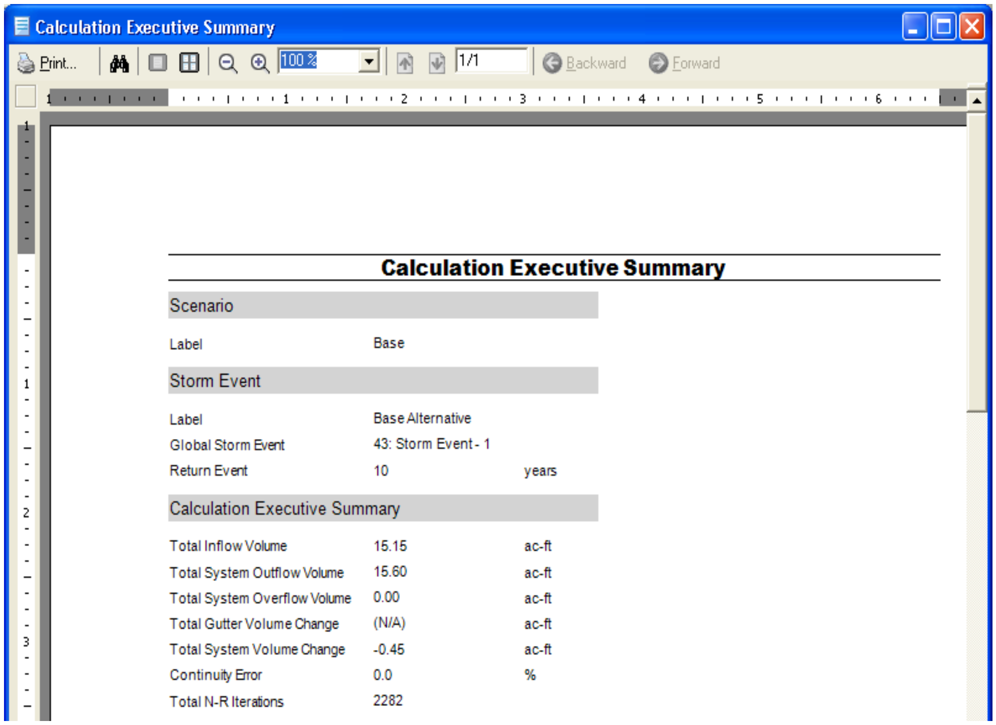
6. Repeat the above step with the second message in the list.

7. **Close** the **User Notifications** dialog.

8. Click the **Compute** button. When the model has been computed, the **Calculation Executive Summary** appears. This dialog displays some of the important calculated results.



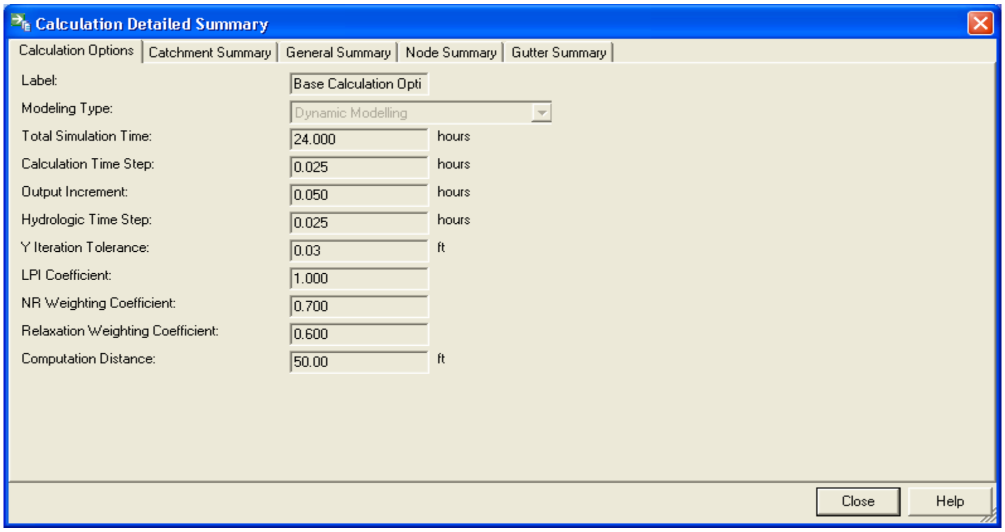
9. You can generate an executive summary report by clicking the **Report** button in this dialog.



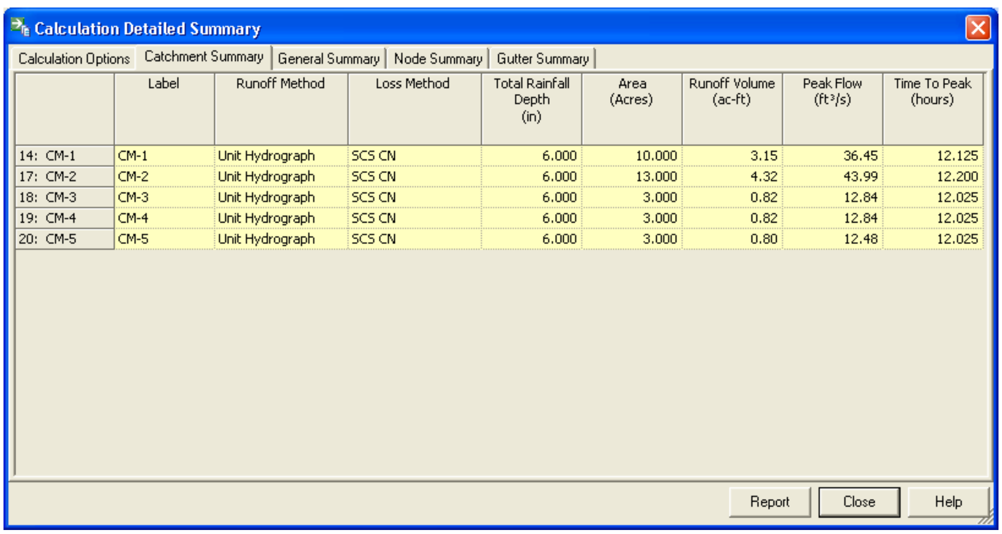
10. Close the **Calculation Executive Summary** report dialog.

11. Click the **Detailed Summary** button. The **Detailed Summary** dialog appears. The Detailed summary contains information divided by category:

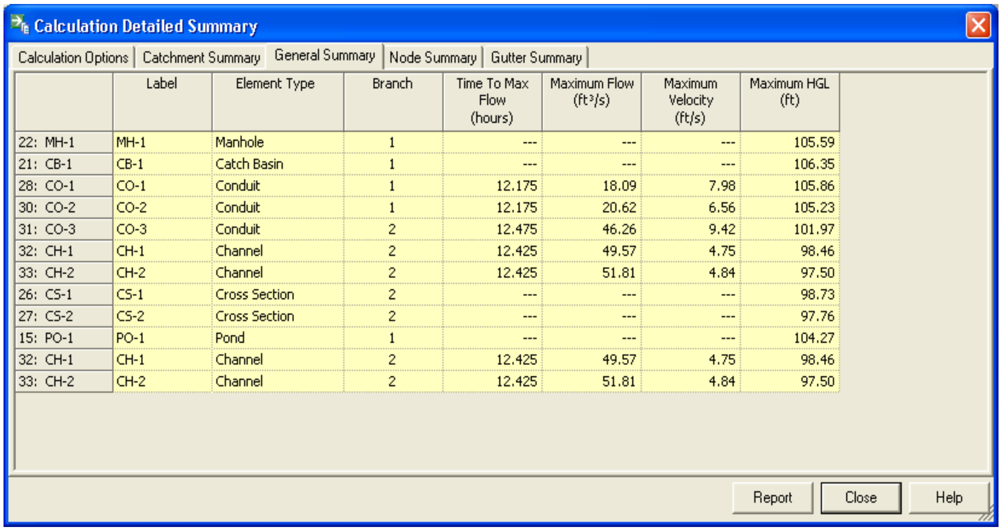
   – Calculation Detailed Summary - Calculation Options (see “Calculation Options Tab” on page 8-441 for more details).



– Calculation Detailed Summary - Catchment Summary (see “Catchment Summary Tab” on page 8-442 for more details).

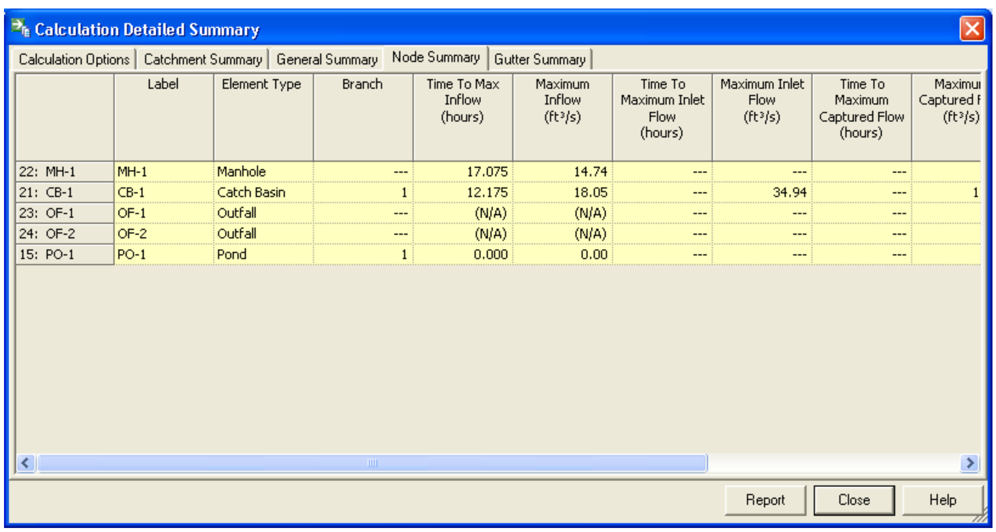


– Calculation Detailed Summary - General Summary (see “General Summary Tab” on page 8-443 for more details).



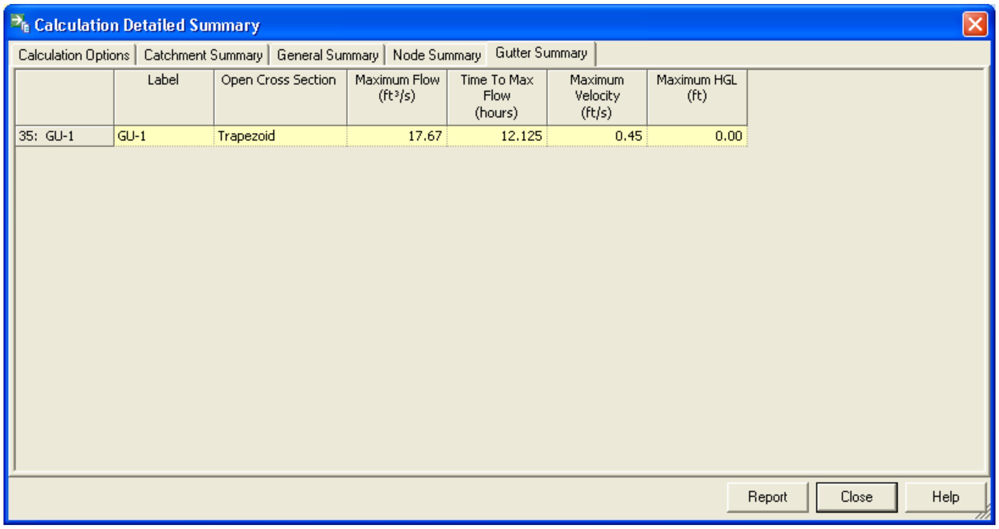
	Label	Element Type	Branch	Time To Max Flow (hours)	Maximum Flow (ft³/s)	Maximum Velocity (ft/s)	Maximum HGL (ft)
22: MH-1	MH-1	Manhole	1	---	---	---	105.59
21: CB-1	CB-1	Catch Basin	1	---	---	---	106.35
28: CO-1	CO-1	Conduit	1	12.175	18.09	7.98	105.86
30: CO-2	CO-2	Conduit	1	12.175	20.62	6.56	105.23
31: CO-3	CO-3	Conduit	2	12.475	46.26	9.42	101.97
32: CH-1	CH-1	Channel	2	12.425	49.57	4.75	98.46
33: CH-2	CH-2	Channel	2	12.425	51.81	4.84	97.50
26: CS-1	CS-1	Cross Section	2	---	---	---	98.73
27: CS-2	CS-2	Cross Section	2	---	---	---	97.76
15: PO-1	PO-1	Pond	1	---	---	---	104.27
32: CH-1	CH-1	Channel	2	12.425	49.57	4.75	98.46
33: CH-2	CH-2	Channel	2	12.425	51.81	4.84	97.50

– Calculation Detailed Summary - Node Summary (see “Node Summary Tab” on page 8-444 for more details).

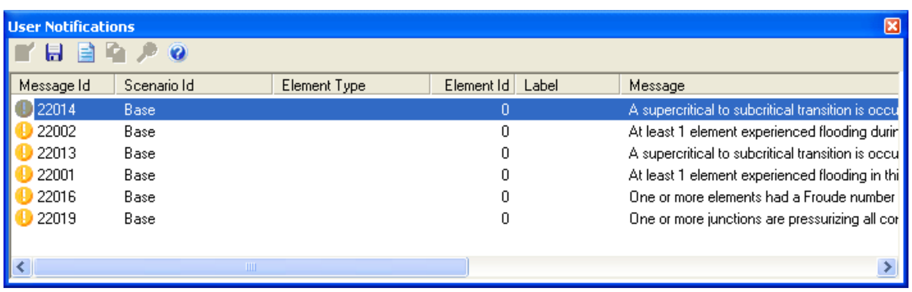


	Label	Element Type	Branch	Time To Max Inflow (hours)	Maximum Inflow (ft³/s)	Time To Maximum Inlet Flow (hours)	Maximum Inlet Flow (ft³/s)	Time To Maximum Captured Flow (hours)	Maximu Captured F (ft³/s)
22: MH-1	MH-1	Manhole	---	17.075	14.74	---	---	---	
21: CB-1	CB-1	Catch Basin	1	12.175	18.05	---	34.94	---	1
23: OF-1	OF-1	Outfall	---	(N/A)	(N/A)	---	---	---	
24: OF-2	OF-2	Outfall	---	(N/A)	(N/A)	---	---	---	
15: PO-1	PO-1	Pond	1	0.000	0.00	---	---	---	

- Calculation Detailed Summary - Gutter Summary (see “Gutter Summary Tab” on page 8-445 for more details).



12. As with the **Executive Summary**, you can generate a report using the **Report** button.
13. Click the **Analysis** menu and select **User Notifications**. Note that there are still messages listed here, although the model did compute successfully. These messages, marked with an orange icon, are warnings (in contrast with errors, marked in red). Warnings do not interfere with the calculations, and do not necessarily invalidate the results. They simply call attention to certain conditions within the network that may be a result of incorrect input data, or that the user might not be immediately aware of.



This concludes Lesson 4. The next lesson continues where this one leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next lesson at a later time, you can either save the current project or use the **Lesson_5_1.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/ **Lessons** folder.

Lesson 5: Presenting Calculated Results

An important feature in all software is the ability to present results clearly. This lesson outlines several of Bentley SewerGEMS V8*i*'s reporting features, including:

- **Reports**, which display and print information on any or all elements in the system.
- **Tabular Reports** (FlexTables), for viewing, editing, and presentation of selected data and elements in a tabular format.
- **Graphs**, to display calculated result attribute values over time for any element in the model.
- **Profiles**, to graphically show, in a profile view, how a selected attribute, such as hydraulic grade, varies along an interconnected series of pipes over time.
- **Element Annotation**, for dynamic presentation of the values of user-selected variables in the plan view.
- **Color Coding**, which assigns colors based on ranges of values to elements in the plan view. Color coding is useful in performing quick diagnostics on the network.

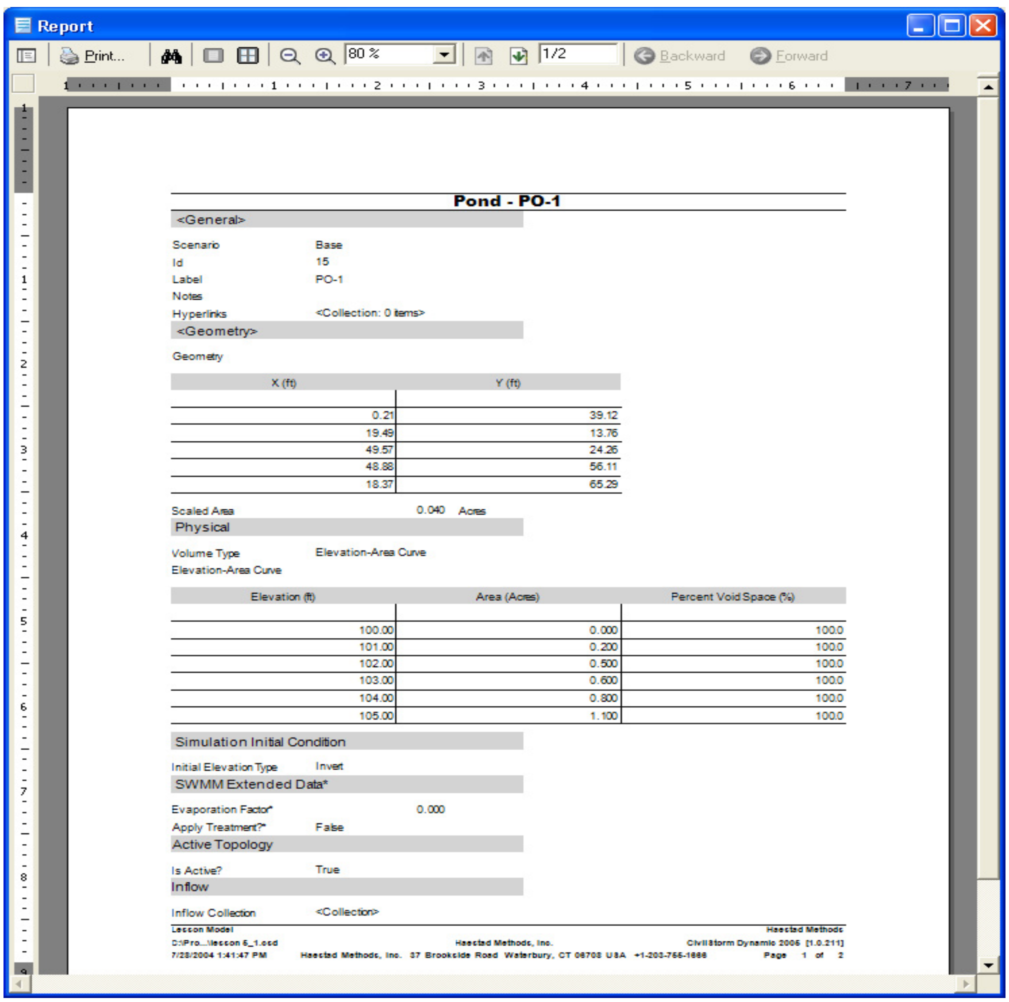
Part 1: Generating Preformatted Reports

Bentley SewerGEMS V8*i* has the ability to generate a number of different preformatted reports. Two of these reports were mentioned in the previous lesson, the Calculation Executive Summary and the Calculation Detailed Summary.

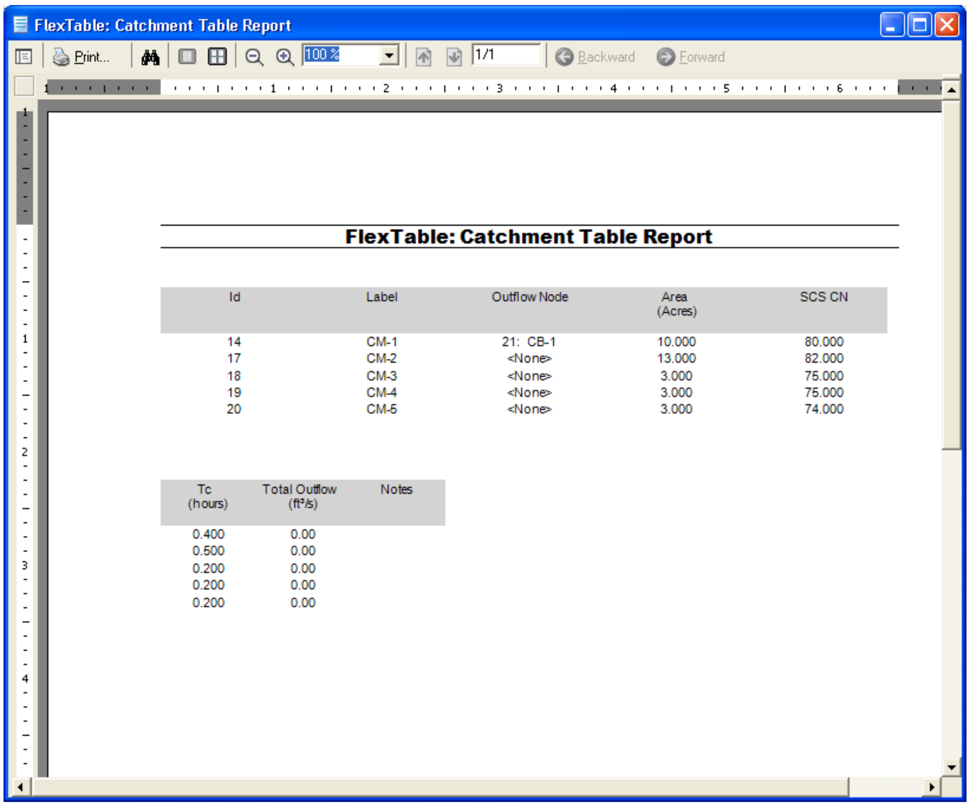
If you've already completed Lesson 4, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button, then browse to the **C:/Program Files/ Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_5_1.swg**, then click **Open.**

1. Click the **Compute** button, then close the **Calculation Executive Summary** dialog.

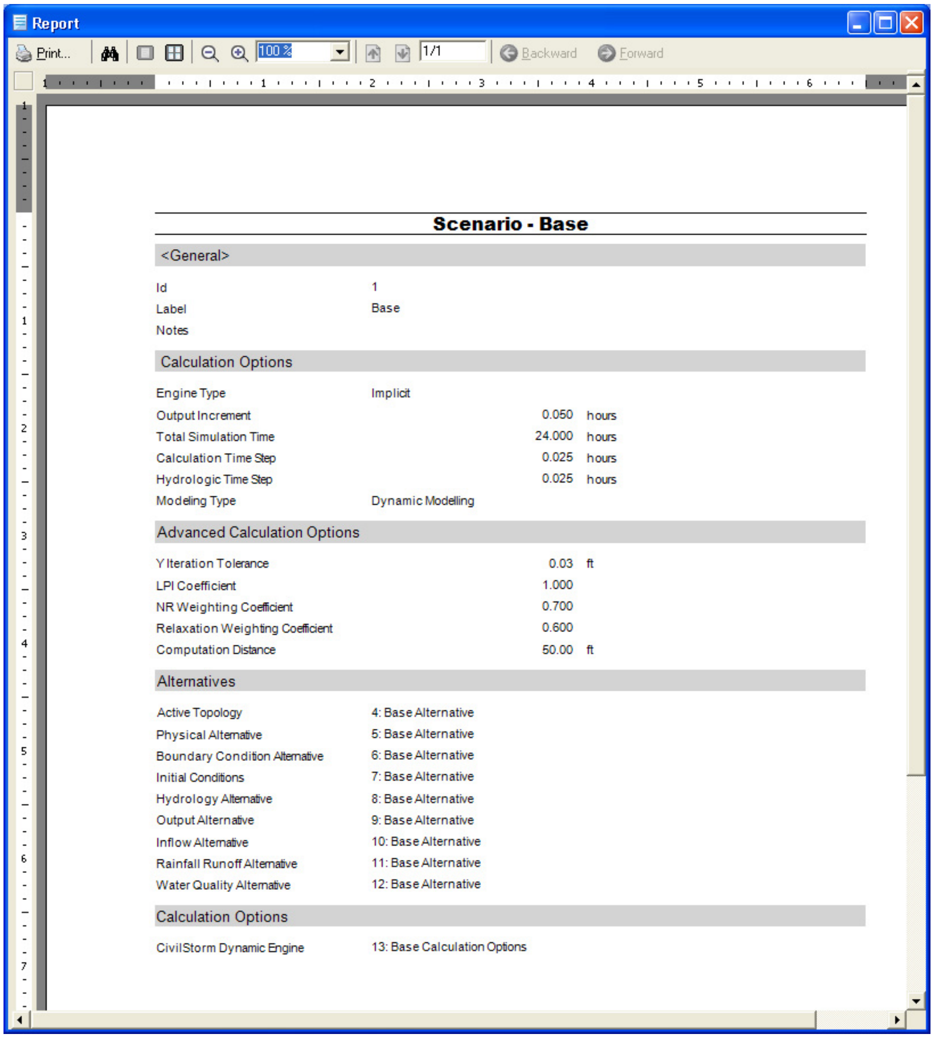
2. Right-click **Diversion Chamber** and select Report from the shortcut menu that appears. You can use this function for any element in the model, allowing you to quickly generate a report detailing a single element's input and output data, as shown below.



3. **Close** the **Print Preview** window.
4. You can also generate reports that detail input and output data for all elements of a single type. Click the **Report** menu, move the mouse cursor over **Element Tables**, and click the **Conduit** command from the submenu that appears.
5. In the **FlexTable** dialog (FlexTables will be discussed further in the next part of this lesson) that appears, click the **Report** button.

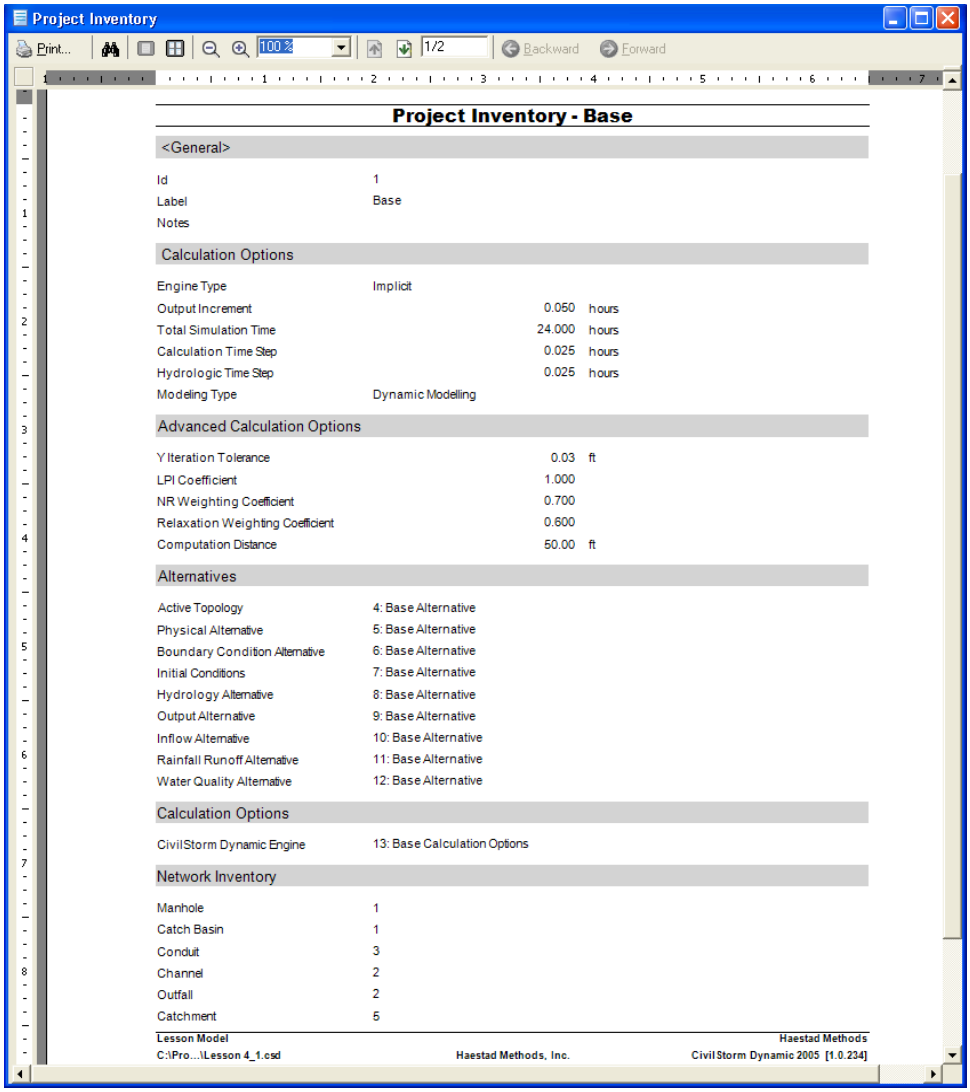


6. **Close** the **Print Preview** and **FlexTable** windows.
7. The Scenario Summary Report details the calculation options that are associated with the current scenario, along with the alternatives that it is comprised of (Scenarios are discussed in the next lesson). Click the **Report** menu and select the **Scenario Summary** command.



8. **Close** the **Print Preview** window.

9. The Project Inventory Report details all of the information contained in the Scenario Summary Report mentioned above, along with a count of the number of each element type in the model. Click the **Report** menu and select the **Project Inventory** command.



10. **Close** the **Project Inventory** report window.

This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_5_2.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

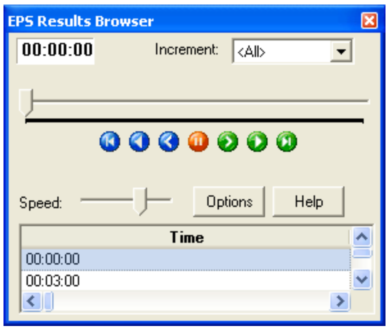
Part 2: Generating Custom Tabular Reports

In Lesson 3, we saw how FlexTables can be used as an efficient means of data entry. They are also useful for creating customized reports of calculated results, as demonstrated in this part of the lesson.

If you've already completed Part 1 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_5_2.swg**, then click **Open.**

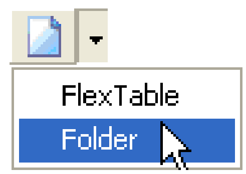
1. Click the **Compute** button, then close the **Calculation Executive Summary** dialog.

2. Click the **Analysis** menu, select the **EPS Results Browser** command. A FlexTable displays the calculated results for the current time step. The EPS Results Browser allows you to control the time step that is displayed.

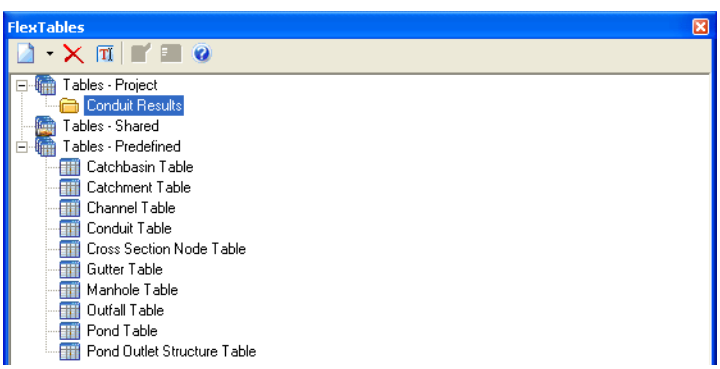


3. Click the **View** menu and select the **FlexTables** command.

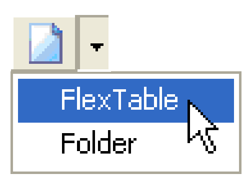
4. In the **FlexTables** dialog, highlight the Tables-Project node, then click the **New** button and select **Folder** from the submenu that appears.



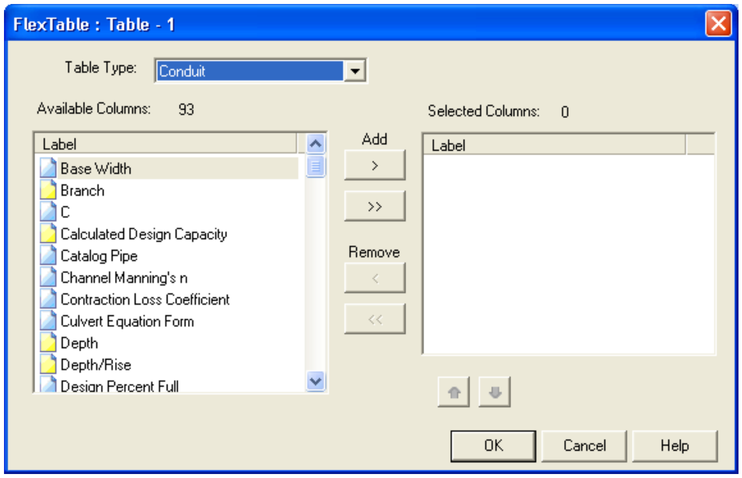
5. A folder is created in the tree view of the FlexTables dialog. This allows you to organize your custom tables. Highlight the newly created folder and click the **Rename** button. Rename the folder to **Conduit Results**.



6. Click the **New** button and select **FlexTable** from the submenu that appears.



7. The **FlexTable** setup dialog appears. Change the **Table Type** to **Conduit**.

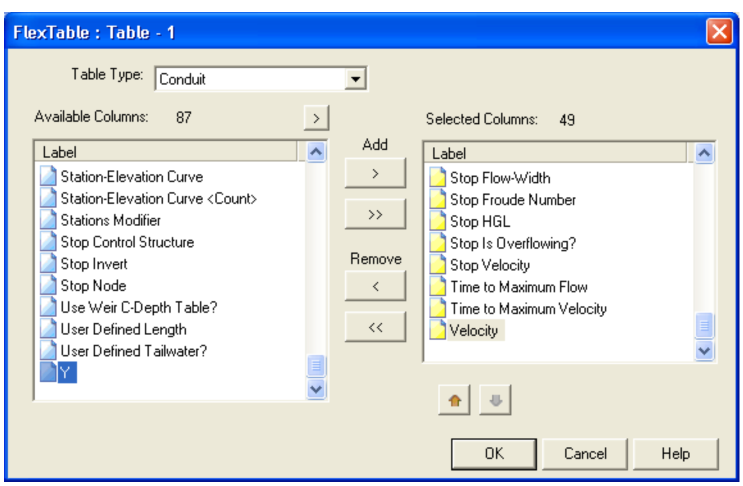


The Available Columns list pane on the left side of the dialog displays all of the attributes that are available for display in the custom table. The Selected Columns list pane, which is currently empty, displays all of the attributes that will be displayed in the custom table. You define a custom table by moving the attributes that you want the table to display from the Available Columns list to the Selected Columns list.

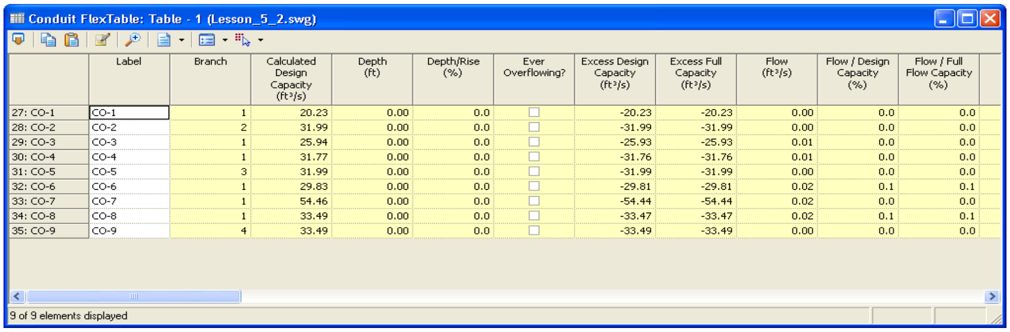
Input data attributes are denoted by a blue icon. 

Output data (result) attributes are denoted by a yellow icon. 

8. Double-click Label to move it to the Selected Columns list.
9. Double-click each of the yellow result attributes in the **Available Column** list to move them to the **Selected Columns** list. Alternatively, you can highlight the attributes and click the **Add** button. 
10. When you have finished moving all of the output attributes to the Selected Columns list, click the **OK** button.



11. The FlexTable display dialog appears. The table currently displays the results for each of the conduits in the model for the 00:00:00 hour timestep. Resize the FlexTable dialog so that more columns are visible, and position the FlexTable dialog so that the EPS Results Browser dialog is also visible.

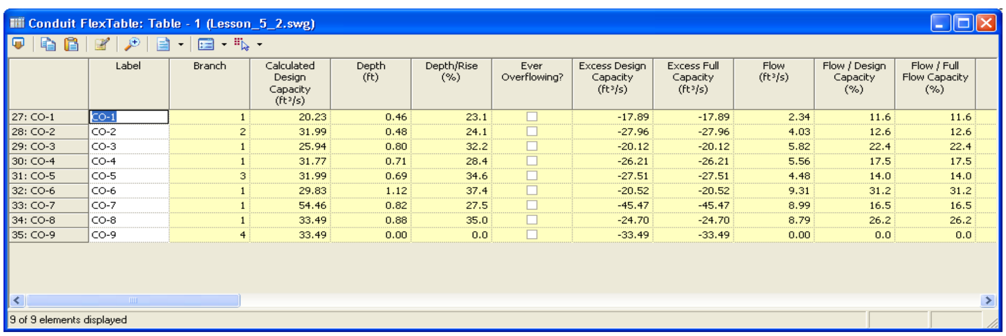


Conduit FlexTable: Table - 1 (Lesson_5_2.swg)

	Label	Branch	Calculated Design Capacity (ft³/s)	Depth (ft)	Depth/Rise (%)	Ever Overflowing?	Excess Design Capacity (ft³/s)	Excess Full Capacity (ft³/s)	Flow (ft³/s)	Flow / Design Capacity (%)	Flow / Full Flow Capacity (%)
27: CO-1	CO-1	1	20.23	0.00	0.0	☐	-20.23	-20.23	0.00	0.0	0.0
28: CO-2	CO-2	2	31.99	0.00	0.0	☐	-31.99	-31.99	0.00	0.0	0.0
29: CO-3	CO-3	1	25.94	0.00	0.0	☐	-25.93	-25.93	0.01	0.0	0.0
30: CO-4	CO-4	1	31.77	0.00	0.0	☐	-31.76	-31.76	0.01	0.0	0.0
31: CO-5	CO-5	3	31.99	0.00	0.0	☐	-31.99	-31.99	0.00	0.0	0.0
32: CO-6	CO-6	1	29.83	0.00	0.0	☐	-29.81	-29.81	0.02	0.1	0.1
33: CO-7	CO-7	1	54.46	0.00	0.0	☐	-54.44	-54.44	0.02	0.0	0.0
34: CO-8	CO-8	1	33.49	0.00	0.0	☐	-33.47	-33.47	0.02	0.1	0.1
35: CO-9	CO-9	4	33.49	0.00	0.0	☐	-33.49	-33.49	0.00	0.0	0.0

9 of 9 elements displayed

12. Click and hold the **Time Slider** control, then slowly drag the control to the right. Note that the values for most of the result attributes in the table vary over time. Move the slider to around the halfway point.

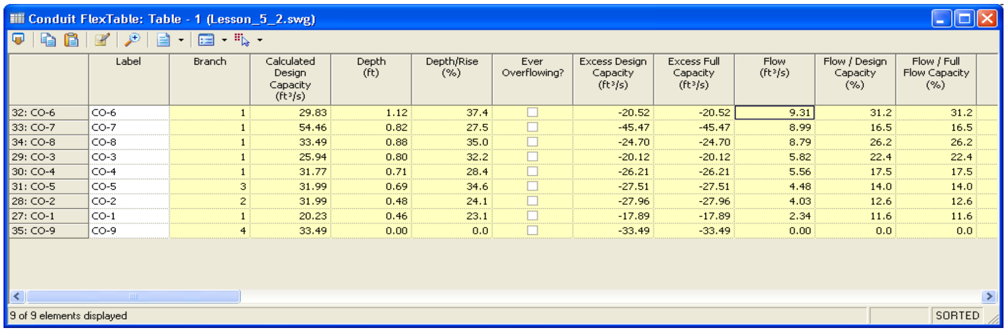


Conduit FlexTable: Table - 1 (Lesson_5_2.swg)

	Label	Branch	Calculated Design Capacity (ft³/s)	Depth (ft)	Depth/Rise (%)	Ever Overflowing?	Excess Design Capacity (ft³/s)	Excess Full Capacity (ft³/s)	Flow (ft³/s)	Flow / Design Capacity (%)	Flow / Full Flow Capacity (%)
27: CO-1	CO-1	1	20.23	0.46	23.1	☐	-17.89	-17.89	2.34	11.6	11.6
28: CO-2	CO-2	2	31.99	0.48	24.1	☐	-27.96	-27.96	4.03	12.6	12.6
29: CO-3	CO-3	1	25.94	0.80	32.2	☐	-20.12	-20.12	5.82	22.4	22.4
30: CO-4	CO-4	1	31.77	0.71	28.4	☐	-26.21	-26.21	5.56	17.5	17.5
31: CO-5	CO-5	3	31.99	0.69	34.6	☐	-27.51	-27.51	4.48	14.0	14.0
32: CO-6	CO-6	1	29.83	1.12	37.4	☐	-20.52	-20.52	9.31	31.2	31.2
33: CO-7	CO-7	1	54.46	0.82	27.5	☐	-45.47	-45.47	8.99	16.5	16.5
34: CO-8	CO-8	1	33.49	0.88	35.0	☐	-24.70	-24.70	8.79	26.2	26.2
35: CO-9	CO-9	4	33.49	0.00	0.0	☐	-33.49	-33.49	0.00	0.0	0.0

9 of 9 elements displayed

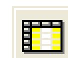
13. The rows in the FlexTable can be sorted according to any attribute. Right-click the **Flow** column heading and select the **Sort...Descending** command from the shortcut menu that appears. The table rows will be arranged so that the element with the highest flow during the current time step is at the top and the element with the lowest flow is at the bottom.


Conduit FlexTable: Table - 1 (Lesson_5_2.swg)

	Label	Branch	Calculated Design Capacity (ft³/s)	Depth (ft)	Depth/Rise (%)	Ever Overflowing?	Excess Design Capacity (ft³/s)	Excess Full Capacity (ft³/s)	Flow (ft³/s)	Flow / Design Capacity (%)	Flow / Full Flow Capacity (%)
32: CO-6	CO-6	1	29.83	1.12	37.4	☐	-20.52	-20.52	9.31	31.2	31.2
33: CO-7	CO-7	1	54.46	0.82	27.5	☐	-45.47	-45.47	8.99	16.5	16.5
34: CO-8	CO-8	1	33.49	0.88	35.0	☐	-24.70	-24.70	8.79	26.2	26.2
29: CO-3	CO-3	1	25.94	0.80	32.2	☐	-20.12	-20.12	5.82	22.4	22.4
30: CO-4	CO-4	1	31.77	0.71	28.4	☐	-26.21	-26.21	5.56	17.5	17.5
31: CO-5	CO-5	3	31.99	0.69	34.6	☐	-27.51	-27.51	4.48	14.0	14.0
28: CO-2	CO-2	2	31.99	0.48	24.1	☐	-27.96	-27.96	4.03	12.6	12.6
27: CO-1	CO-1	1	20.23	0.46	23.1	☐	-17.89	-17.89	2.34	11.6	11.6
35: CO-9	CO-9	4	33.49	0.00	0.0	☐	-33.49	-33.49	0.00	0.0	0.0

9 of 9 elements displayed SORTED

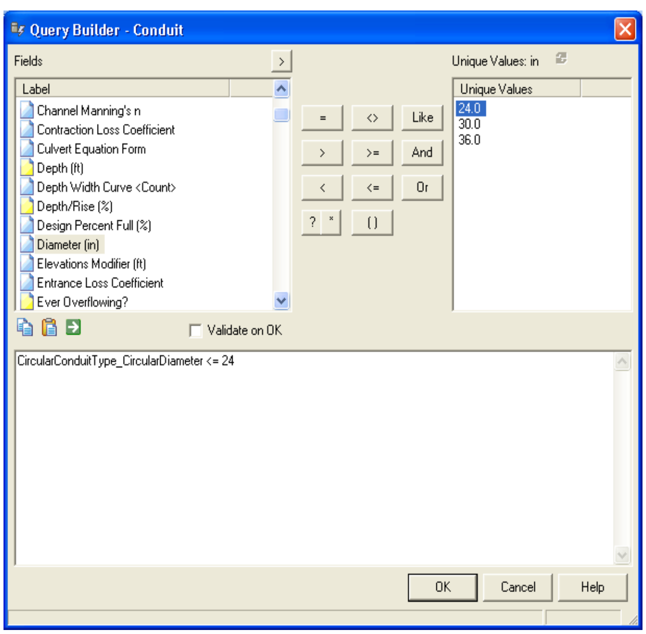
14. You can apply filters to any FlexTable. Filters let you change the table so that only rows that match the specified criteria will appear. Tables can be filtered according to any attribute. Click the **Edit** button.



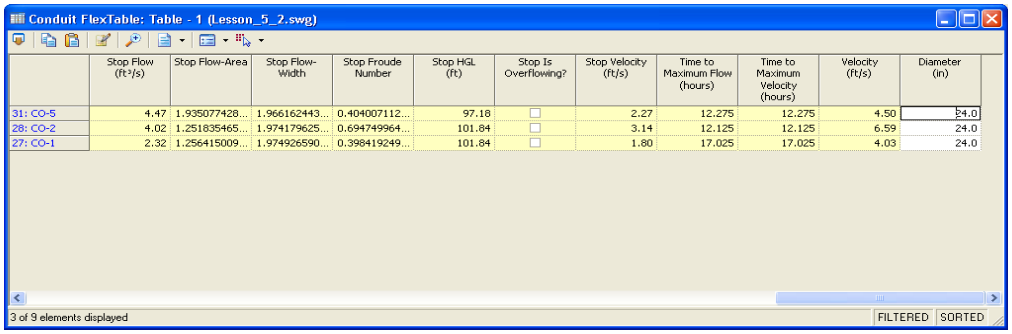
15. In the **FlexTable** setup dialog, double-click the **Diameter** attribute in the Available Columns list. Click **OK**.

16. In the **FlexTable** display dialog, scroll to the **Diameter** column. Right-click the column heading and select the **Filter...Filter** command from the shortcut menu that appears.

17. In the **Filter** dialog, scroll to the **Diameter** attribute in the **Fields** pane and double-click it to add it to the query pane at the bottom of the dialog. Click the <= operator button. Click the refresh button above the Unique Values pane and double-click **24.0** in the list that appears. The dialog should now look like this:



18. Click the **Apply** button, click **OK** in the Query Successful prompt, then click **OK** to close the Query Builder dialog.
19. In the FlexTable display dialog, note that there are only three elements displayed, and a message has appeared along the bottom of the dialog: “3 of 9 elements displayed” and a “FILTERED” notification appears to the right of the message. Only the elements with a diameter of 24 inches or less are shown in the table because of the filter we created.


Conduit FlexTable: Table - 1 (Lesson_5_2.swg)

	Stop Flow (ft^3/s)	Stop Flow-Area	Stop Flow-Width	Stop Froude Number	Stop HGL (ft)	Stop Is Overflowing?	Stop Velocity (ft/s)	Time to Maximum Flow (hours)	Time to Maximum Velocity (hours)	Velocity (ft/s)	Diameter (in)
31: CO-5	4.47	1.935077428...	1.966162443...	0.404007112...	97.18	☐	2.27	12.275	12.275	4.50	24.0
28: CO-2	4.02	1.251835465...	1.974179625...	0.694749964...	101.84	☐	3.14	12.125	12.125	6.59	24.0
27: CO-1	2.32	1.256415009...	1.974926590...	0.398419249...	101.84	☐	1.80	17.025	17.025	4.03	24.0

3 of 9 elements displayed FILTERED SORTED

20. **Close** the **FlexTable** display dialog, then **close** the **FlexTables** manager dialog.

This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_5_3.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

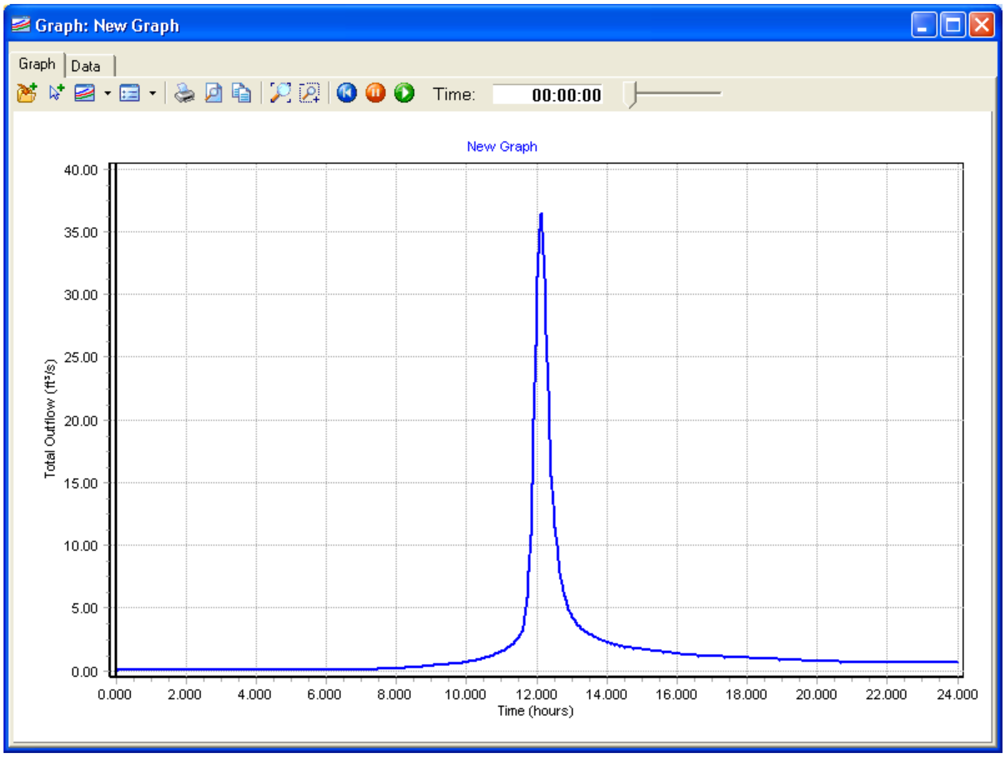
Part 3: Using Graphs

Graphs display calculated result attribute values over time for any element in the model. For elements with more than one attribute available for graphing, you can display all of the attributes on the same graph.

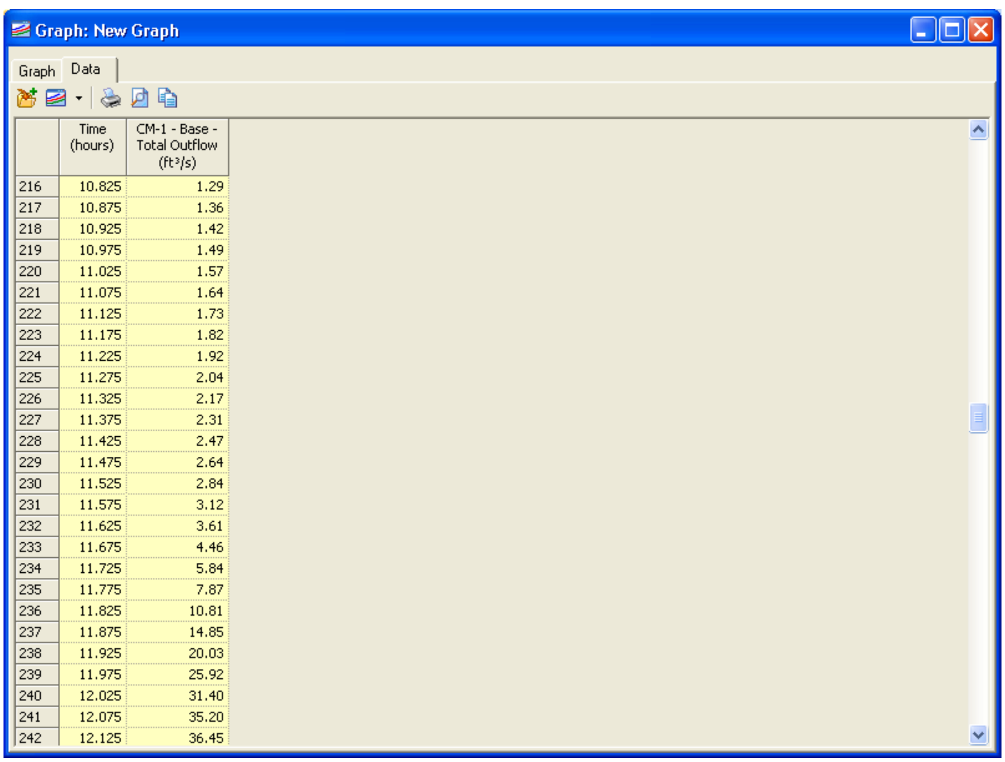
If you’ve already completed Part 2 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_5_3.swg**, then click **Open.**

1. Click the **Compute** button, then close the **Calculation Executive Summary** dialog.

2. Right-click on **CM-1** and select **Graph** from the shortcut menu that appears.
3. Close the **Graph Series Options** dialog that appears. The **Graph** dialog opens, displaying the plot of total outflow over time (the default attribute for catchment graphs) for CM-1 for the duration of the simulation.

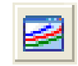


4. You can view the data on which the graph is based by clicking the **Data** tab.

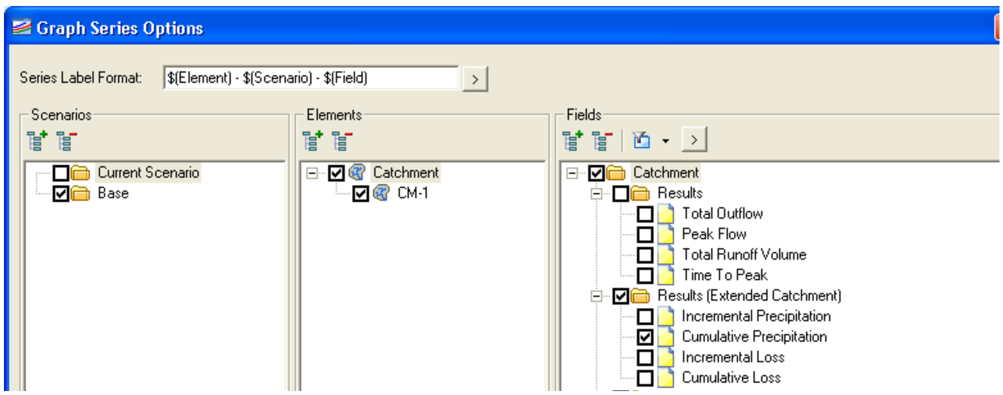


	Time (hours)	CM-1 - Base - Total Outflow (ft³/s)
216	10.825	1.29
217	10.875	1.36
218	10.925	1.42
219	10.975	1.49
220	11.025	1.57
221	11.075	1.64
222	11.125	1.73
223	11.175	1.82
224	11.225	1.92
225	11.275	2.04
226	11.325	2.17
227	11.375	2.31
228	11.425	2.47
229	11.475	2.64
230	11.525	2.84
231	11.575	3.12
232	11.625	3.61
233	11.675	4.46
234	11.725	5.84
235	11.775	7.87
236	11.825	10.81
237	11.875	14.85
238	11.925	20.03
239	11.975	25.92
240	12.025	31.40
241	12.075	35.20
242	12.125	36.45

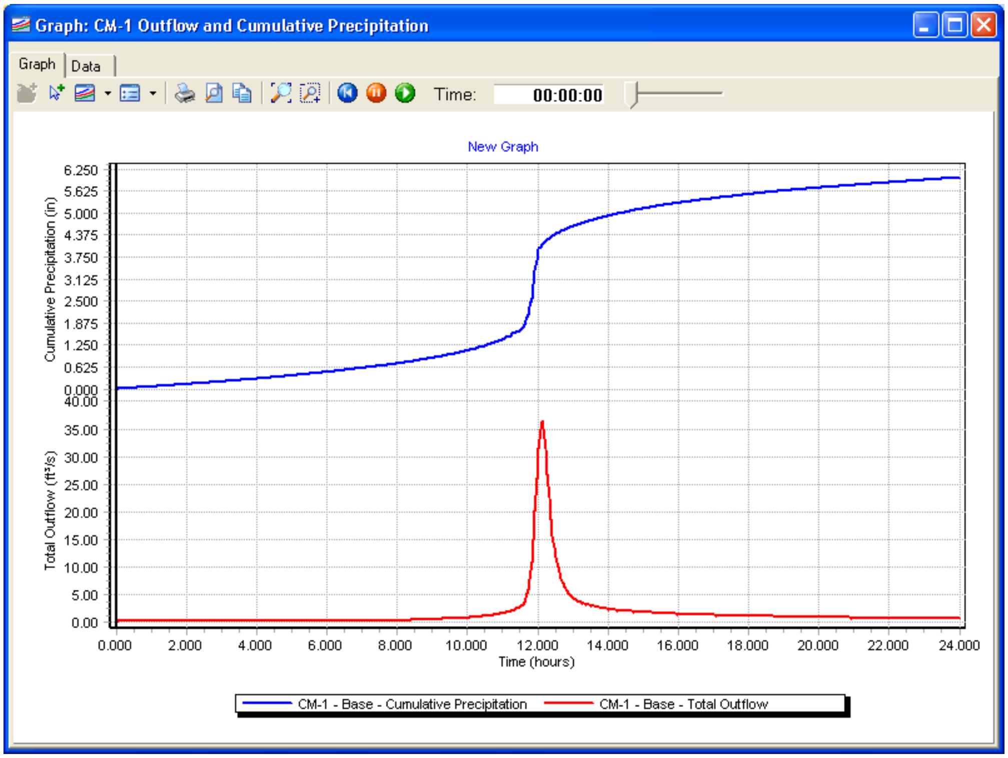
5. Switch back to the Graph tab and click the **Graph Series Options** button.



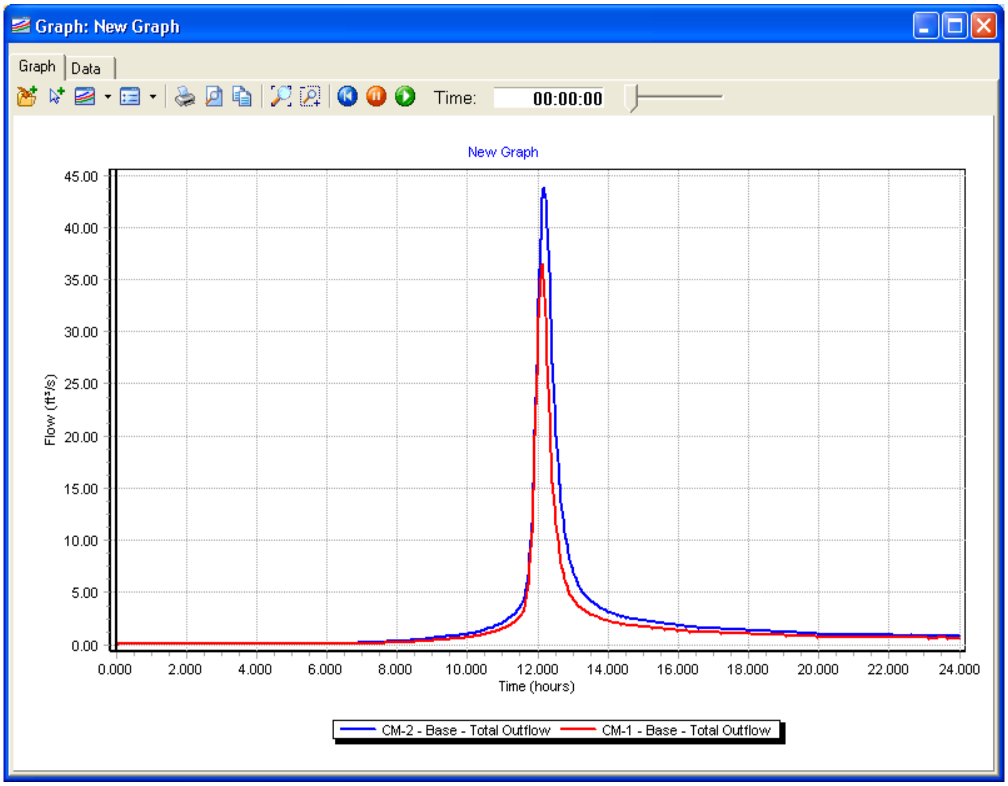
6. The **Graph Series Options** dialog that appears allows you to control what is displayed by the graph, including scenarios, elements, and attributes. Scenarios will be discussed in the next lesson, and we will revisit graphs at that time.
7. In the **Fields** list, clear the **Total Outflow** checkbox (under the **Results** folder) and click the **Cumulative Precipitation** checkbox (under the **Results (Extended Catchment)** folder), then click the **OK** button.



8. You can also display more than one attribute simultaneously on the same graph. Click the **Graph Series Options** button.
9. Click the **Total Outflow** checkbox in the **Fields** pane, then click the **OK** button.
10. The graph now displays both outflow and cumulative precipitation for CM-1. **Click the Add to Graph Manager** button to save the Graph and enter the name **CM-1 Outflow and Cumulative Precipitation** in the **Create Graph** dialog that appears**. Close** the Graph view dialog.

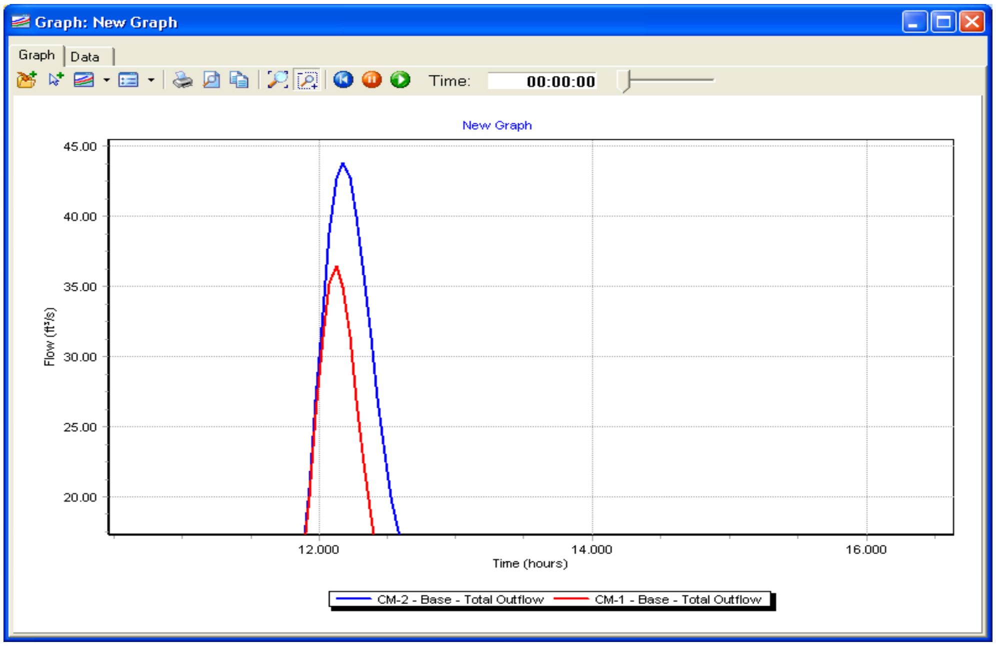


11. You can graph multiple elements on the same graph. Hold down the **Ctrl** key on your keyboard and click both of the catchment elements in turn.
12. Right-click on one of the catchments and select the **Graph** command from the shortcut menu. Close the **Graph Series Options** dialog.



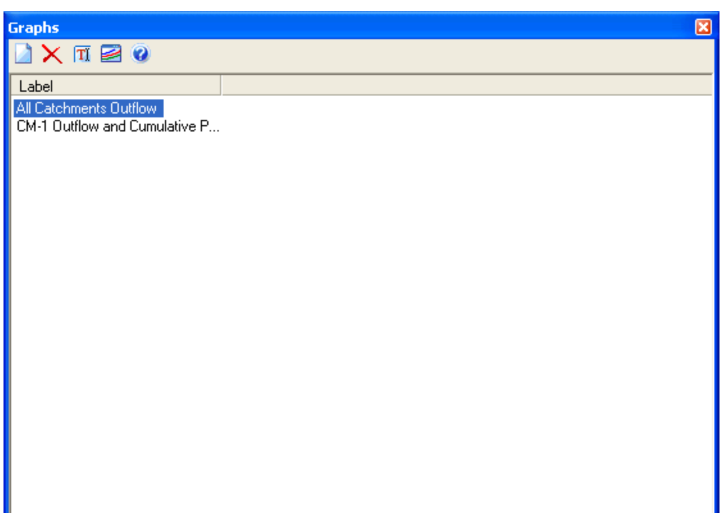
13. You can zoom in on any area of the graph using the Zoom tool. Click the **Zoom** tool to activate it. The zoom tool behaves like the Zoom Window tool for the drawing pane; you define the area to be zoomed by clicking on

the graph to define the top-left corner of the zoom area, then hold down the mouse button and drag down and to the right, releasing the mouse button when the cursor is positioned at the bottom-right corner of the zoom area.



14. To zoom back out, click, hold, and drag to the left, then release the mouse button. This returns the view to the full extent zoom level, displaying the entire graph. Alternatively, you can click the **Zoom Extents** button.

15. Click the **Save** button and enter the name **All Catchments Outflow** in the **Create Graph** dialog**,** then click **OK**. Close the **Graph** dialog.
16. Click the **View** menu and select the **Graphs** command.
17. In the **Graphs** manager dialog that appears, note the saved graphs.



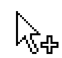
18. Close the **Graphs** manager dialog.

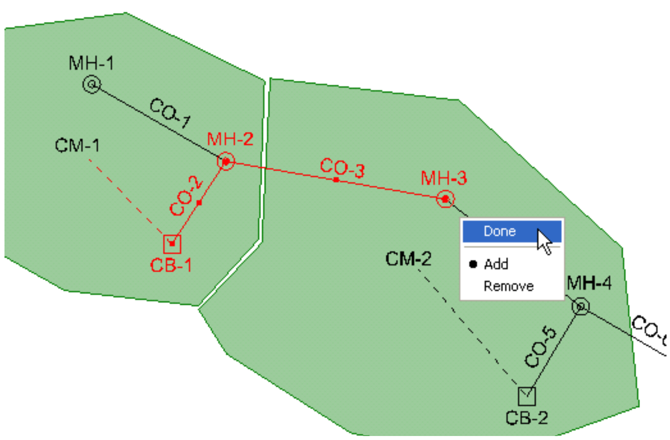
This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_5_4.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

Part 4: Generating Profiles

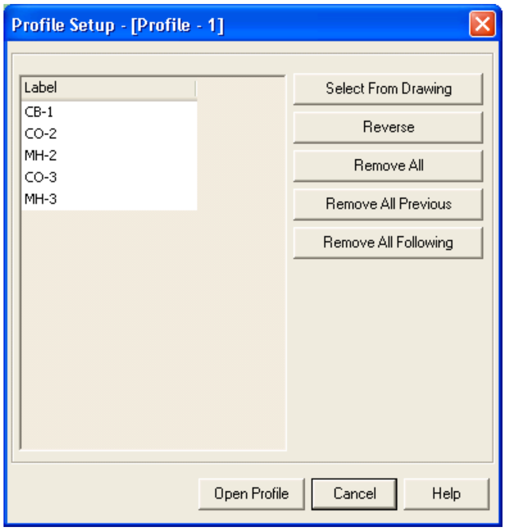
Profiles graphically show how a selected attribute, such as hydraulic grade, varies along an interconnected series of network elements over time.

If you've already completed Part 3 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_5_4.swg**, then click **Open.**

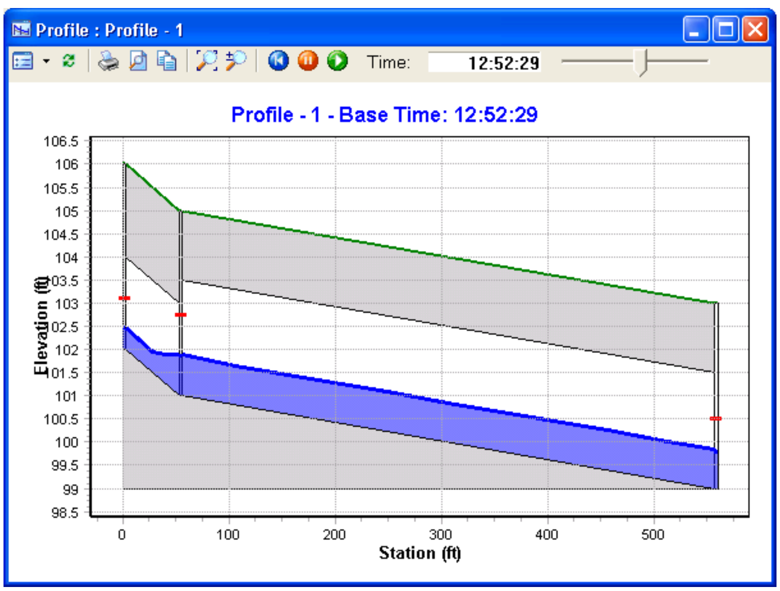
1. Click the **Compute** button, then close the **Calculation Executive Summary** dialog. 
2. Click the **View** menu and select the **Profiles** command.
3. In the **Profiles** manager that appears, click the **New** button. 
4. In the **Profile Setup** dialog that appears, click the **Select From Drawing** button.
5. The mouse cursor changes to an Element Selection tool. Click on **CB-1** and **MH-3**. **MH-2**, which is the intermediate node between them, is also highlighted, along with the link elements that join these elements. Right-click and select **Done** from the shortcut menu that appears. 



6. This returns you to the **Profile Setup** dialog. Note that the list pane now contains the elements that were highlighted in the drawing pane. These are the elements that will be displayed in the profile, in the order they will appear from left to right in the profile view. Click the **Open Profile** button.



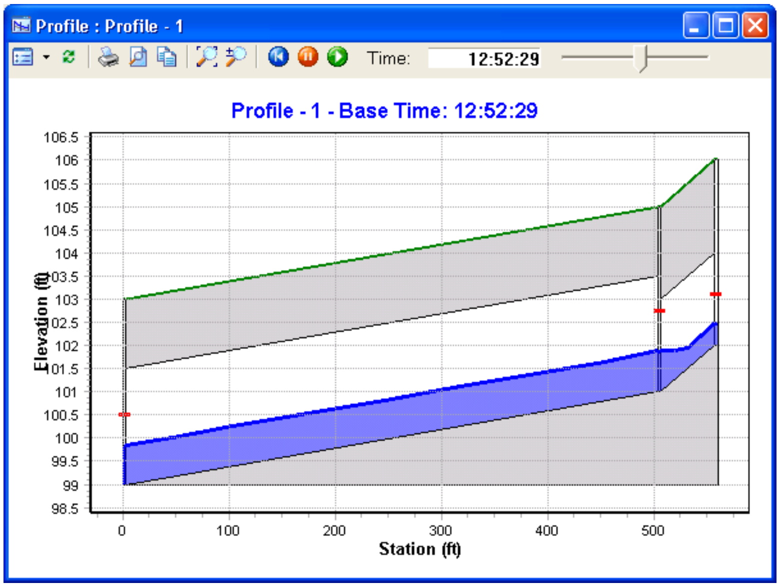
7. The **Profile** view dialog appears, displaying the three nodes and two links that were highlighted. The blue line represents the calculated HGL. The green line represents the ground elevation. The parallel vertical lines represent the node elements. The red marks superimposed over the node elements represent the maximum HGL calculated for that node. Click the **Play** button to see how the HGL changes over the course of the simulation.



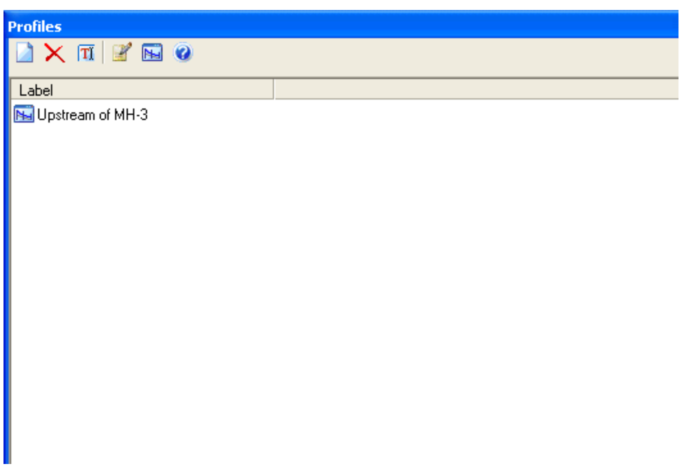
8. **Close** the **Profile** display dialog. In the **Profiles** manager dialog, highlight **Profile-1** and click the **Edit** button.

9. This opens the **Profile Setup** dialog for the highlighted profile. Click the **Reverse** button, then click the **Open Profile** button.

10. Note that the direction of the profile has been reversed. **Close** the **Profile** view dialog.

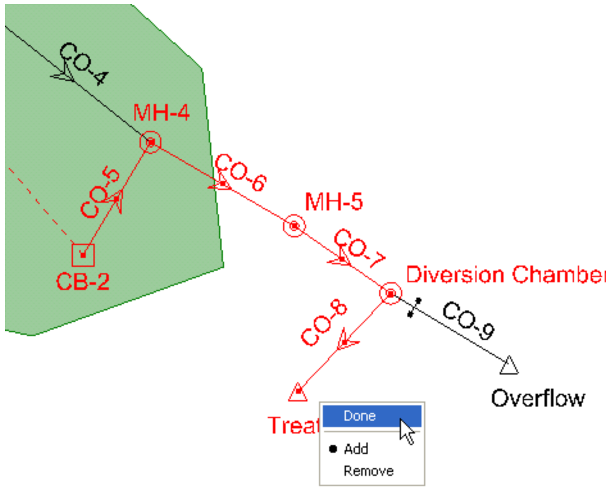


11. In the **Profiles** manager, highlight **Profile-1** and click the **Rename** button. Enter the name **Upstream Of MH-3**.

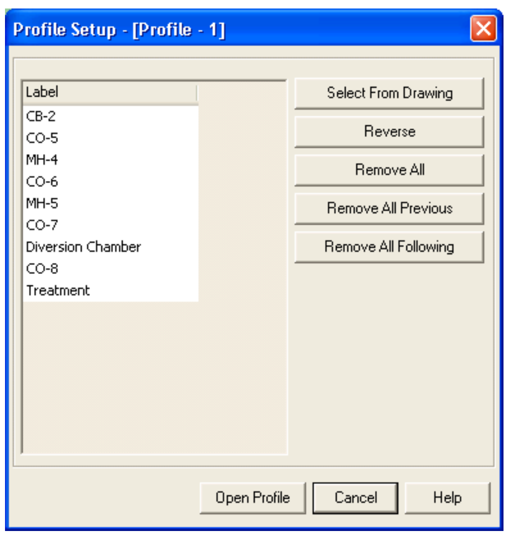


12. Click the **New** button. In the **Profile Setup** dialog, click the **Select From Drawing** button.

13. In the drawing pane, click on **CB-2**, **MH-5**, **Diversion Chamber**, and **Treatment**. Right-click and select **Done** from the shortcut menu.

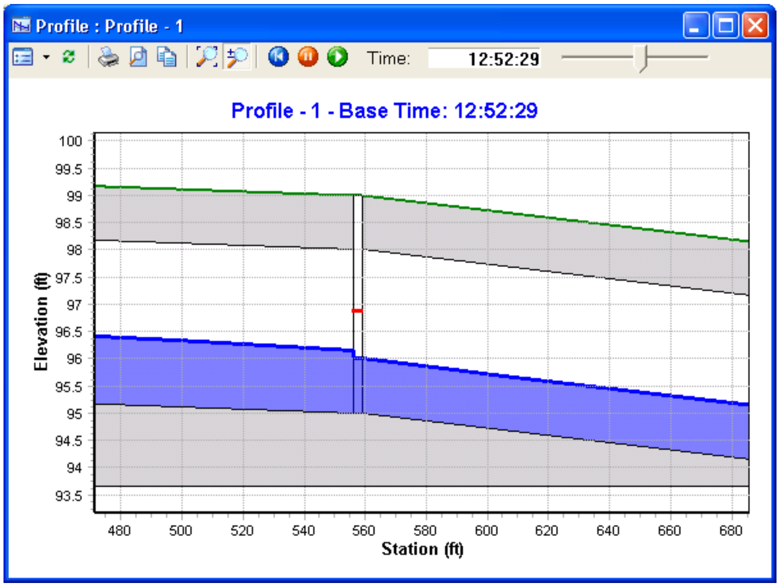


14. In the **Profile Setup** dialog, click the **Open Profile** button.

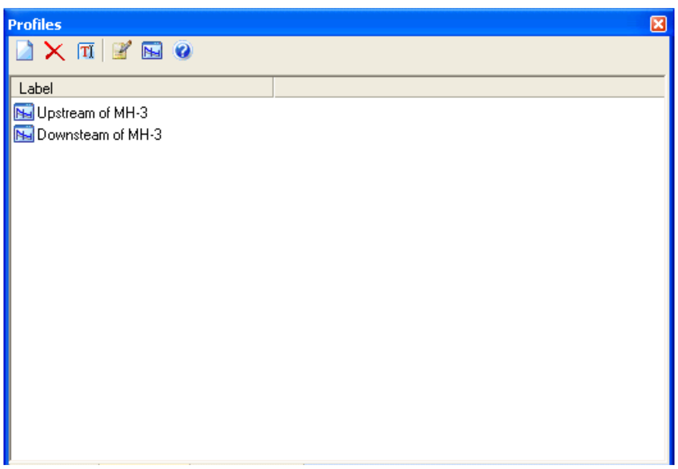


15. You can zoom in on any area of the profile using the Zoom tool. Click the **Zoom** tool to activate it. The zoom tool behaves like the Zoom Window tool for the drawing pane; you define the area to be zoomed by clicking on

the graph to define the top-left corner of the zoom area, then holding down the mouse button and dragging down and to the right, releasing the mouse button when the cursor is positioned at the bottom-right corner of the zoom area.



16. To zoom back out, click, hold, and drag to the left, then release the mouse button. This returns the view to the full extent zoom level, displaying the entire profile. Alternatively, you can click the **Zoom Extents** button. 
17. Close the **Profile** view dialog. In the **Profiles** manager, highlight **Profile-1** and click the **Rename** button. Enter the name **Downstream of MH-3**.



18. Close the **Profiles** manager dialog.

This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_5_5.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

Part 5: Applying Element Annotation

Element annotation functionality allows the display of values for user-selected attributes in the drawing pane. These values are dynamically updated when the current time step is changed.

If you've already completed Part 4 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_5_5.swg**, then click **Open.**

1. Click the **Compute** button, then close the **Calculation Executive Summary** dialog.

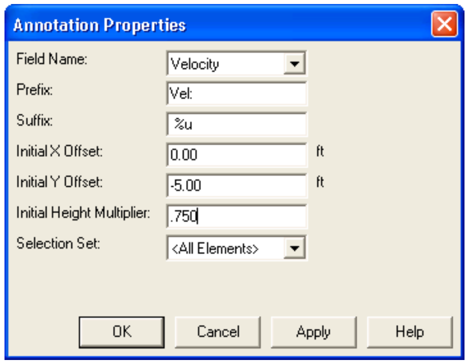
2. Annotation is assigned through the **Element Symbology** manager. If you are using the default workspace configuration, the Element Symbology manager is located directly below the toolbars on the left side of the dialog. If not, click the **View** menu and select the **Element Symbology** command. Highlight **Conduit** and click the **New** button, then select **New Annotation** from the shortcut menu that appears.

3. In the **Annotation Properties** dialog that appears, change the **Field Name** to **Velocity**. In the **Prefix** field, type in **Vel:** (with a space after the colon).

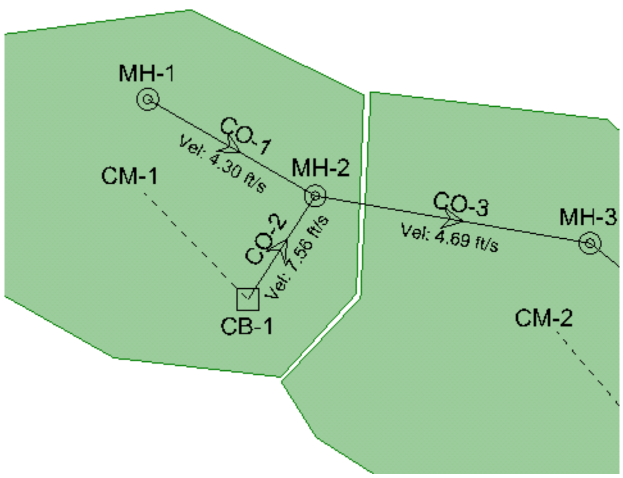
4. The X and Y Initial offset fields allow you to define, respectively, the horizontal and vertical distance between the element and the annotation. A positive value for Initial X Offset will cause the annotation to be placed to the right of the element at the distance specified; a negative value will cause the annotation to be placed to

the left of the element. A positive value for Initial Y Offset will cause the annotation to be placed above the element; a negative value will cause the annotation to be placed below it. Enter a value of **-5.00** feet for the **Initial Y Offset**.

5. The **Initial Height Multiplier** allows you to increase the size of text used for the annotation. Change this value to **0.750**. The **Selection Set** control allows you to apply the current annotation to only those elements contained within a previously defined selection set. Leave this value at **<All Elements>**. Click the **OK** button.



6. In the **EPS Results Browser**, click and slowly drag the time slider to the right. Note that the velocity annotation values display the updated value for the current time step as it changes.



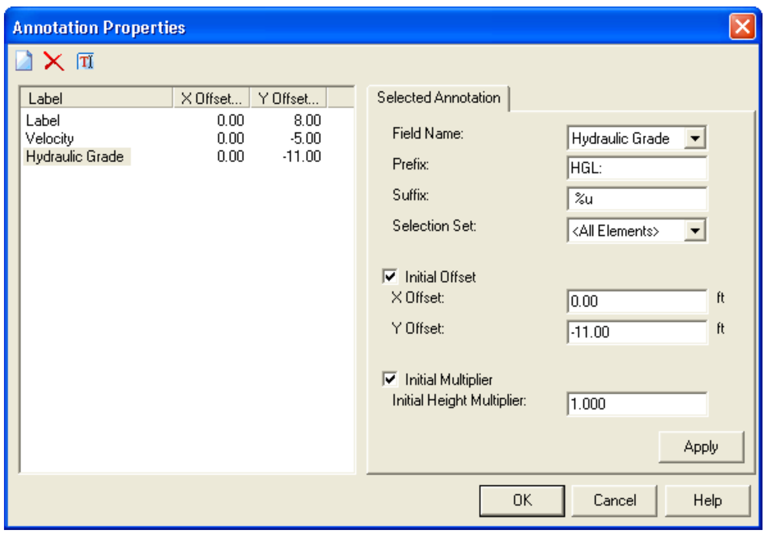
7. In the **Element Symbology** manager, click the **New** button and select **New Annotation** from the shortcut menu that appears.

8. Change the **Field Name** to **Hydraulic Grade**. Enter **HGL:** (with a space after the colon) in the **Prefix** field. Change the **Y Offset** to **-8.00**. Change the **Initial Height Multiplier** value to **0.750**. Click the **OK** button.

9. Note that the hydraulic grade line value is now displayed below the velocity annotation. However, the two annotations slightly overlap. Highlight the **Hydraulic Grade** annotation node in the **Element Symbology** manager and click the **Edit** button.

10. In the **Annotation Properties** dialog that appears, highlight **Hydraulic Grade** in the list pane on the left side of the dialog. Change the **Y Offset** to **-11.00** and click the **Apply** button.

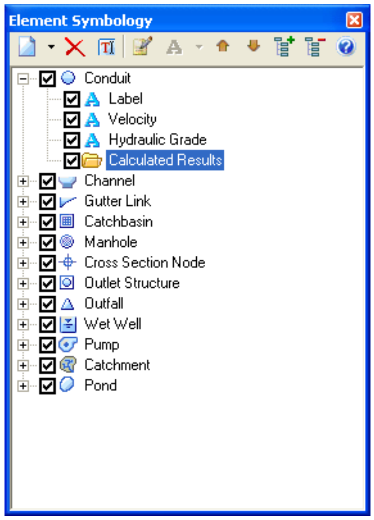
    Note the Initial Offset and Initial Multiplier checkboxes. When these are checked, the settings for the annotation that is currently highlighted in the list pane will be applied to all of the elements with that particular annotation (in this case, all conduits). If you have manually moved some of the annotations in the drawing pane, you should clear the Initial Offset checkbox so that the new settings won't interfere with your manually repositioned annotations.



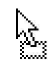
11. In the **EPS Results Browser**, click and slowly drag the time slider to the right to see how the values change over time.

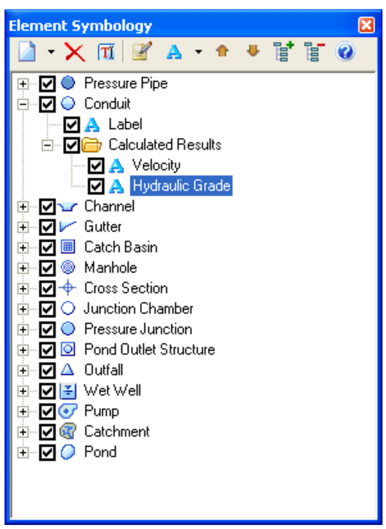
12. In the **Element Symbology** manager, you can create Theme Folders to organize the various annotations for an element type. Highlight **Conduit** and click the **New** button, then select **New Folder** from the shortcut menu that appears.

13. Highlight the newly created folder and click the **Rename** button. Enter the name **Calculated Results.**



14. Click on the **Velocity** annotation label and hold down the mouse button, then drag the mouse cursor to the **Calculated Results** folder. Your mouse cursor will change to a drag object icon. Release the mouse button to place the **Velocity** annotation in the folder. Repeat this procedure with the **HGL** annotation.





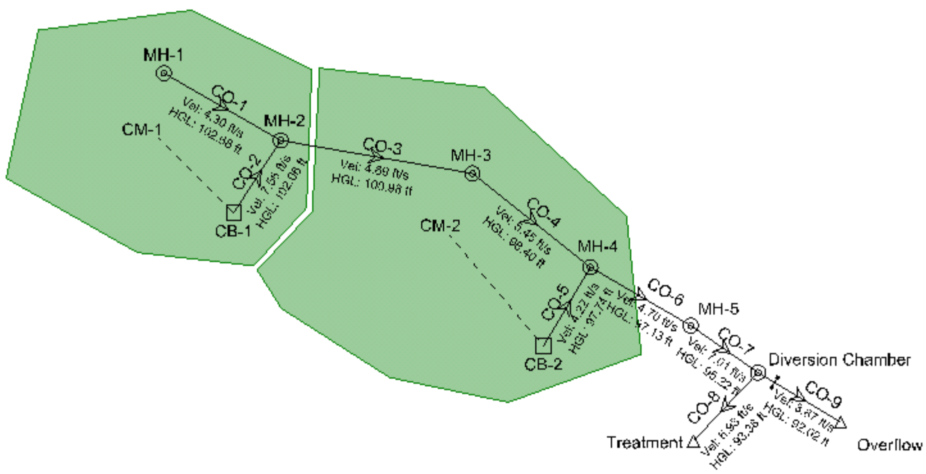
15. The checkboxes next to each node in the Element Symbology manager list control the visibility of the associated object in the drawing pane, as follows:

    The checkbox next to the Conduit node (and the corresponding checkboxes next to each of the other element types) controls the visibility of conduit elements in the drawing pane.

    The checkbox next to the Label node controls the visibility of conduit element labels in the drawing pane.

The checkbox next to a folder controls the visibility of all annotation definitions within that folder. In the case of the Calculated Results folder, it controls the visibility of the Velocity and HGL annotations for conduit elements in the drawing pane.

16. Clear the checkbox next to the **Calculated Results** folder. Note that both the Velocity and HGL annotations disappear from the drawing pane, while the Label annotation is still displayed. Click the checkbox next to the **Calculated Results** folder to turn the annotations back on.
17. The results annotations may now interfere with the visibility of other elements and labels. Manually reposition the element labels if this is the case, so that all of the labels are clearly visible (To learn how to manually reposition labels, see "Part 3: Moving Element Labels" on page 3-62). Your model should now look like this:



This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_5_6.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

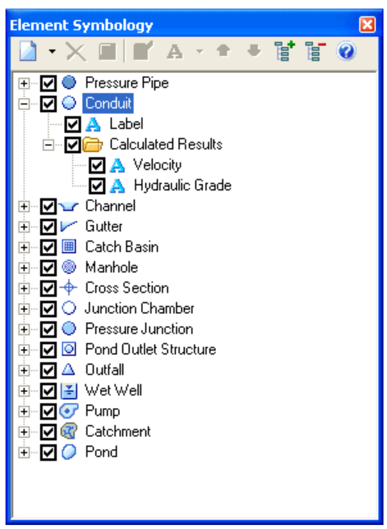
Part 6: Applying Color Coding

Color Coding allows you to assign colors based on ranges of values for a specified attribute to elements in the plan view. Color coding is useful in performing quick diagnostics on the network.

If you've already completed Part 5 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley/**SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_5_6.swg**, then click **Open.**

1. Click the **Compute** button, then close the **Calculation Executive Summary** dialog.

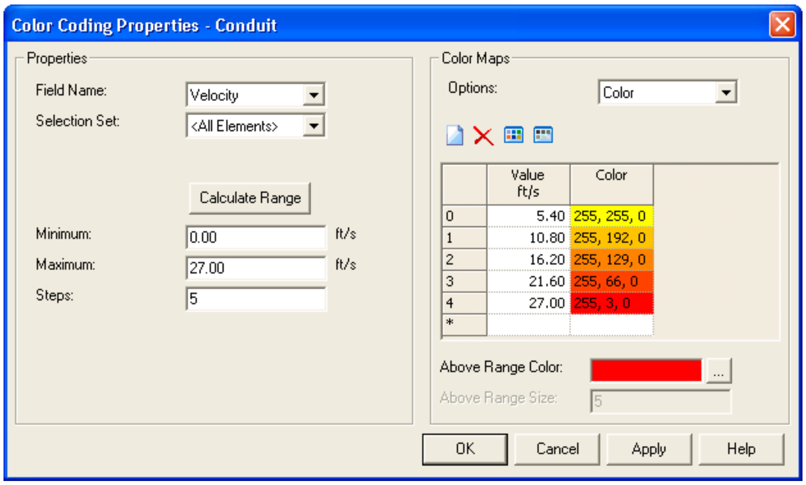
2. Color Coding is assigned through the **Element Symbology** manager. If you are using the default workspace configuration, the Element Symbology manager is located directly below the toolbars on the left side of the dialog. If not, click the **View** menu and select the **Element Symbology** command. Highlight **Conduit** and click the **New** button, then select **New Color Coding** from the shortcut menu that appears.



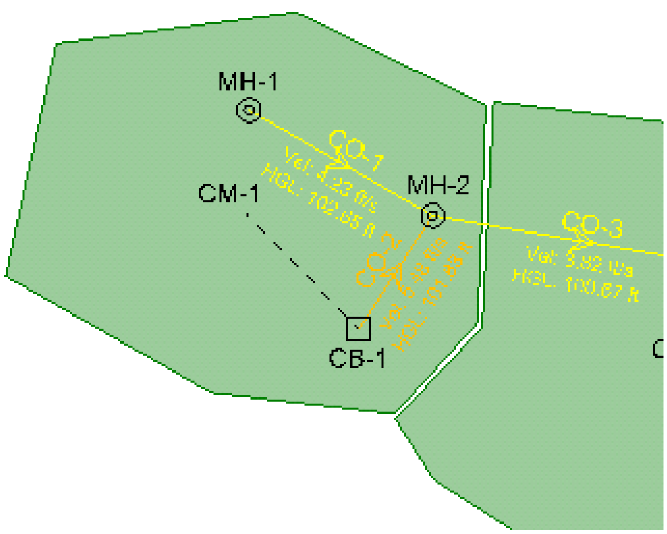
3. In the **Color Coding Properties** dialog that appears, change the **Field Name** to **Velocity**. The **Selection Set** control allows you to apply the current color coding to only those elements contained within a previously defined selection set. Leave this value at **<All Elements>**.
4. Click the **Calculate Range** button.
5. This fills in the **Min.** and **Max.** fields using the highest and lowest calculated values (over the duration of the entire simulation, not just the current time step) for the attribute specified in the **Field Name** menu. The **Steps** field lets you specify how many intermediate points are created between the minimum and maximum values defined by the **Min.** and **Max.** fields (with the min and max values each representing a point counting towards the total as well). Leave the **Steps** value at **5**.

6. Under Color Maps, leave the **Options** field set to **Color**. Click the **Initialize** button. The Color Maps table is now populated with 5 rows (because there were 5 steps in the range) and a different color has been assigned to each step.

7. Click the arrow button in the **Color** column of the first row and select **Yellow**. Click the **Ramp** button. The three middle colors are changed to various shades of orange. The **Ramp** button assigns colors to the intermediate rows to create a gradient between the first and last colors in the table. Click the **OK** button.

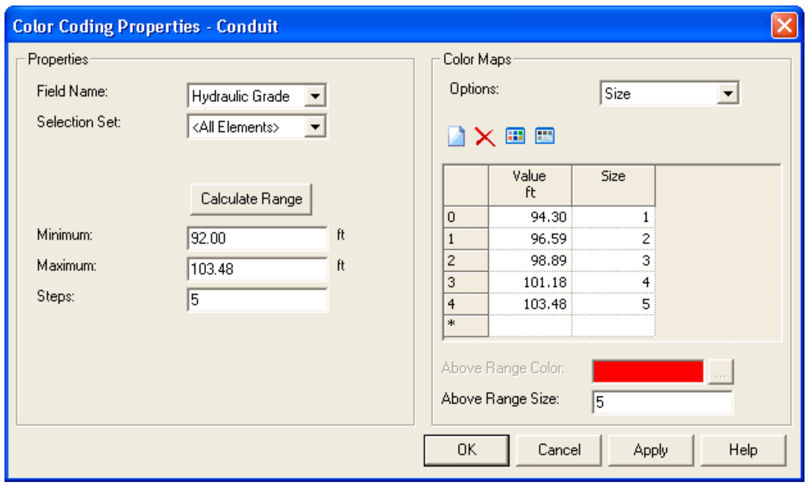


8. In the **EPS Results Browser**, click and slowly drag the time slider to the right. Note that the color of the conduit elements change according to the calculated velocity value for the current time step.

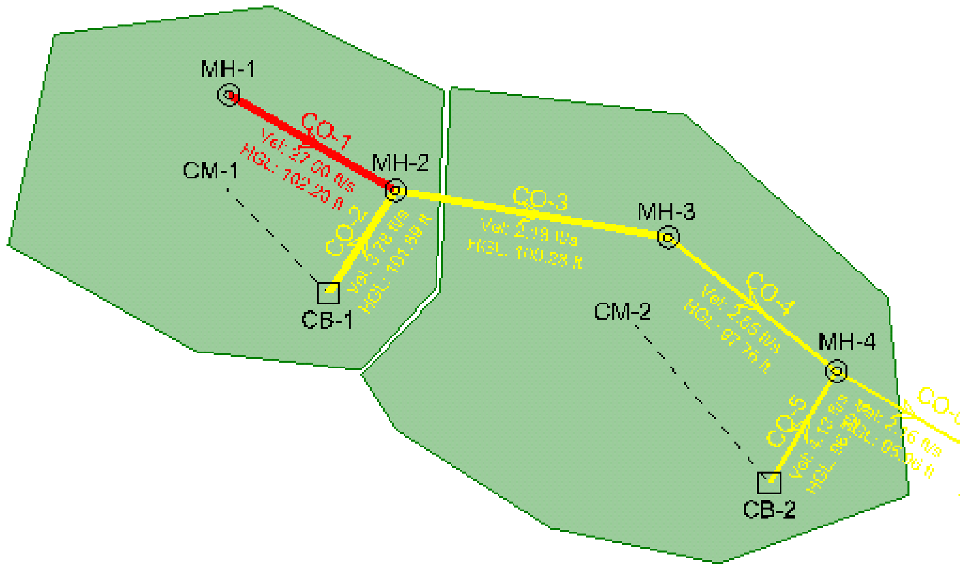


You can also set up color coding to change the size of an element type in the drawing pane according to the value of a specified attribute.

9. In the **Element Symbology** manager, click the **New** button and select **New Color Coding** from the shortcut menu that appears.
10. In the **Color Coding Properties** dialog, change the **Field Name** to **Hydraulic Grade**. Leave the **Selection Set** value at **<All Elements>**. Click the **Calculate Range** button. Leave the **Steps** value at **5**.
11. Under Color Maps, change the **Options** value to **Size**. Click the **Initialize** button. For the first row, leave the **Size** value in the Color Maps table at **1**, the value for the second row at **2**, the third at **3**, the fourth at **4**, and the fifth at **5**. The Size values are a multiplier of the default element symbol size. In the case of link elements like conduits, the value is a multiplier of the default line weight (width). So a Size value of 5 for a conduit means that a conduit displayed at that value will be five times wider than a default conduit. Click the **OK** button.



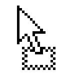
12. In the **EPS Results Browser**, click and slowly drag the time slider to the right. Note that the color of the conduit elements change according to the calculated velocity value, while at the same time the size of the conduits changes according to the calculated HGL for the current time step.

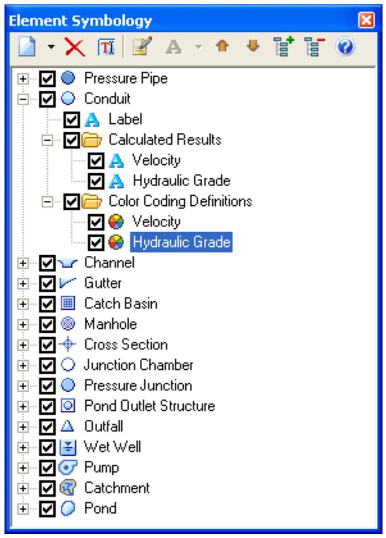


13. In the **Element Symbology** manager, you can create Theme Folders to organize the various color coding definitions for an element type. Highlight **Conduit** and click the **New** button, then select **New Folder** from the shortcut menu that appears.

14. Highlight the newly created folder and click the **Rename** button. Enter the name **Color Coding Definitions**.

15. Click on the **Velocity** color coding label and hold down the mouse button, then drag the mouse cursor to the **Color Coding Definitions** folder. Your mouse cursor will change to a drag object icon. Release the mouse button to place the **Velocity** color coding definition underneath the folder. Repeat this procedure with the **HGL** color coding definition.





16. The checkboxes next to each node in the Element Symbology manager list control the visibility of the associated object in the drawing pane, as follows:

    The checkbox next to the Conduit node (and the corresponding checkboxes next to each of the other element types) controls the visibility of conduit elements in the drawing pane.

    The checkbox next to the Label node controls the visibility of conduit element labels in the drawing pane.

    The checkbox next to a folder controls the visibility of all annotation definitions within that folder. In the case of the Color Coding folder, it controls the visibility of the Velocity and HGL color coding definitions for conduit elements in the drawing pane.

17. Clear the checkbox next to the **Color Coding Definitions** folder. Note that both the Velocity and HGL color codings disappear from the drawing pane, leaving the conduits displayed in the default color and size. Click the checkbox next to the **Color Coding Definitions** folder to turn the color coding definitions back on.

This concludes Lesson 5. The next lesson continues where this one leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next lesson at a later time, you can either save the current project or use the **Lesson_6_1.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/ **Lessons** folder.

Lesson 6: Creating Multiple Storm Events

In Lesson 3, we used a predefined storm event for ease of data entry. In this lesson, we will set up multiple return events from synthetic rainfall data and apply them to the model by creating a new scenario using a new Rainfall Runoff alternative.

It is common for engineers to consider multiple storm events when designing drainage facilities. Multiple rainfall events are modeled in SewerGEMS via rainfall alternatives and scenario management.

This lesson describes the process for defining and applying multiple rainfall events. Two 24-hour events will be applied to the SCS Type II dimensionless rainfall distributions. This procedure can be applied to an unlimited number of design storms for use with different scenarios.

General Organization of Storm Events

It is important to understand the overall hierarchy of design storm data in the context of scenario management. There are basically three major levels of design storm data that you must create and assign:

Storm Events - This tabular data represents the raw data for a single storm event. This data can come from applying rainfall depth to dimensionless rainfall distributions (such as SCS Type I, IA, II, III), or from actual gauged data. Storm Events are entered using the main menu option Analysis / Storm Events.

Rainfall Runoff Alternatives - Each rainfall alternative references an individual Storm Event. Rainfall Runoff alternatives are created and edited by clicking the Alternatives button on the main tool bar to access the Alternatives manager.

Scenarios - Any scenario can reference any Rainfall Runoff Alternative. This hierarchy provides maximum flexibility for use with scenario management. Scenarios are edited by clicking the Scenario button on the main toolbar.

The overall hierarchy discussed here can be illustrated as:

Scenario references > **Rainfall Runoff Alternative** that references > **Storm Event**

EXAMPLE: ENTER MULTIPLE STORM EVENTS FOR THE FOLLOWING DATA, THEN REFERENCE WITH RAINFALL RUNOFF ALTERNATIVES AND SCENARIOS:

10-Year, 24-Hour Depth = 4.8 inches, SCS Type II Dimensionless Distribution

100-Year, 24-Hour Depth = 7.1 inches, SCS Type II Dimensionless Distribution

Part 1: Creating Unique Storm Events for Design Storms

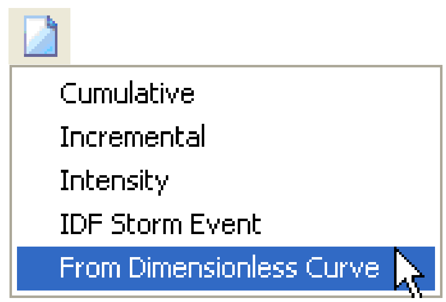
We must first establish our raw data for each unique rainfall event. Follow the sequence below to enter 2 SCS rainfall events.

If you've already completed Lesson 5, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/Program Files/ Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_6_1.swg**, then click **Open.**

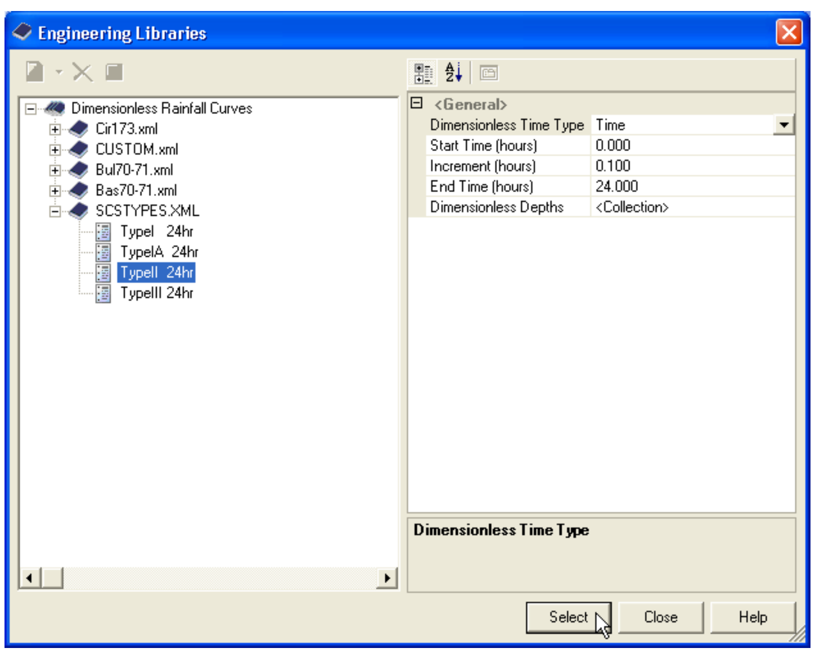
1. Click the **Components** menu and select **Storm Events**. This will display the **Storm Events** manager window.

   We will now enter a SCS rainfall 10-year event.

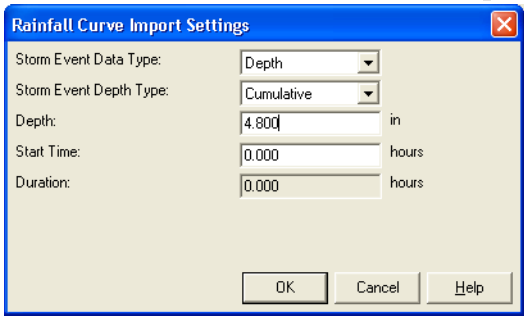
2. In the **Storm Events** manager dialog, click the **New** button and select the **From Dimensionless Curve** command from the submenu that appears.



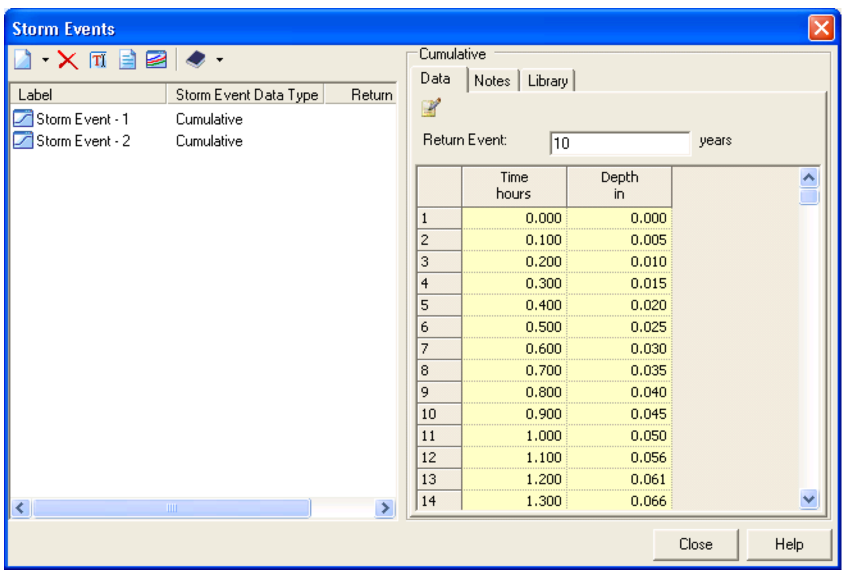
3. In the **Engineering Libraries** dialog that appears, the Dimensionless Rainfall Curves library will be displayed. Expand the **Dimensionless Rainfall Curves** node in the list pane on the left by clicking the **Plus** button, then expand the **SCSTYPES.xml** node so that all four distributions are shown (Types I, IA, II, III). Select **Type II 24hr**, then click the **Select** button.



4. In the **Rainfall Curve Import Settings** dialog that appears, change the **Storm Event Data Type** field to **Depth** and the **Storm Event Depth Type** to **Cumulative**. Enter a value of **4.800** in the **Depth** field, then click **OK**.



5. SewerGEMS V8*i* will generate a cumulative rainfall curve by multiplying 4.8 inches (the depth that was specified in step 4) by the SCS Type II dimensionless distribution (the curve type that was selected in Step 3). The resulting cumulative depth table will be displayed in the **Data** table on the right side of the **Storm Events** dialog. Type **10** in the **Return Event** field above the Data table.



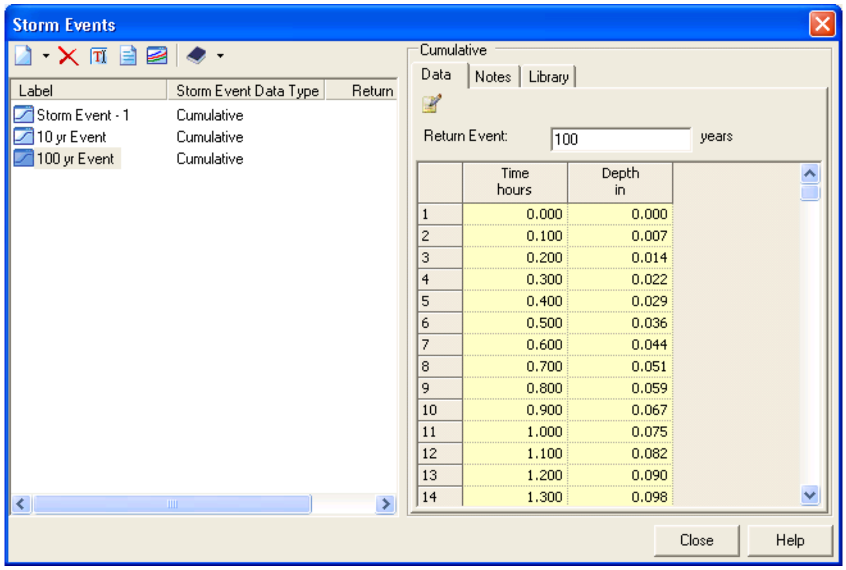
6. Highlight the newly created storm in the list pane on the left side of the dialog and click the **Rename** button. Type in **10 yr Event**.

7. Repeat steps 2 and 3.

8. In the **Rainfall Curve Import Settings** dialog that appears, change the **Storm Event Data Type** field to **Depth** and the **Storm Event Depth Type** to **Cumulative**. Enter a value of **7.100** in the **Depth** field, then click **OK**.

9. Type **100** in the **Return Event** field above the **Data** table.

10. Highlight the newly created storm in the list pane on the left side of the dialog and click the **Rename** button. Type in **100 yr Event**.



11. **Close** the **Storm Events** manager dialog.

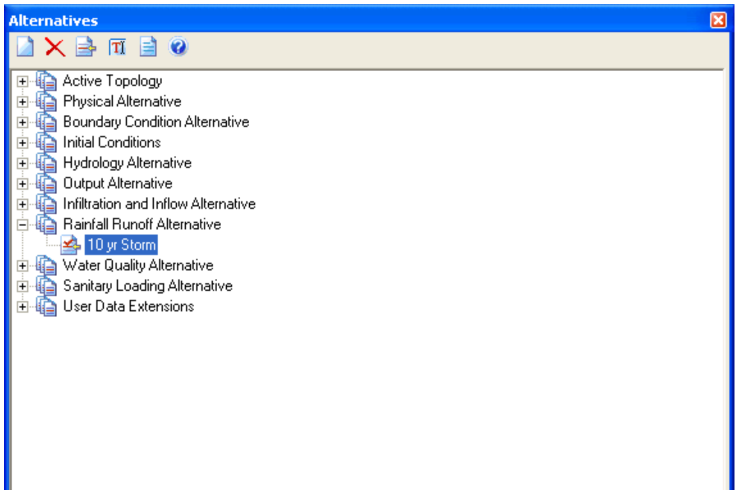
This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_6_2.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

Part 2: Creating Rainfall Runoff Alternatives to Reference Storm Events

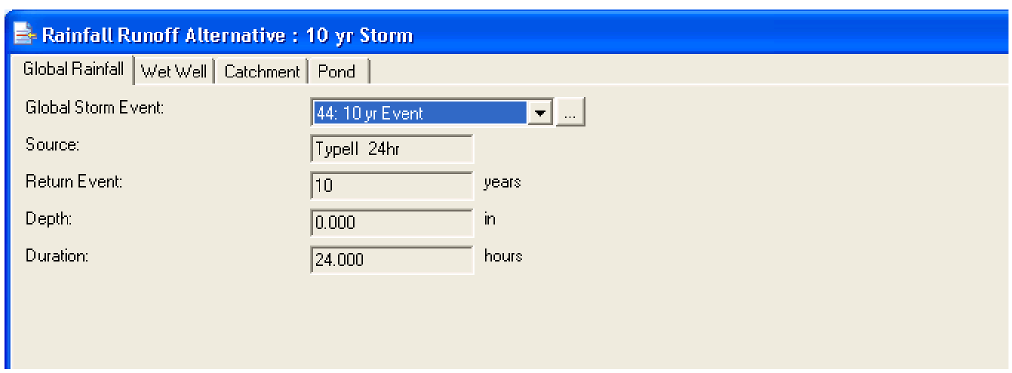
In Part 1 of this lesson, we created our raw rainfall storm event data. In this part of the lesson, we will reference these storms by creating Rainfall Runoff Alternatives.

If you've already completed Part 1 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/ Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_6_2.swg**, then click **Open.**

1. Click the **Analysis** menu and select the **Alternatives** command.
2. In the **Alternatives** manager dialog that appears, expand the **Rainfall Runoff Alternative** node by clicking the plus button next to it. Highlight the **Base Alternative** and click the **Rename** button. Type in **10 yr Storm**.

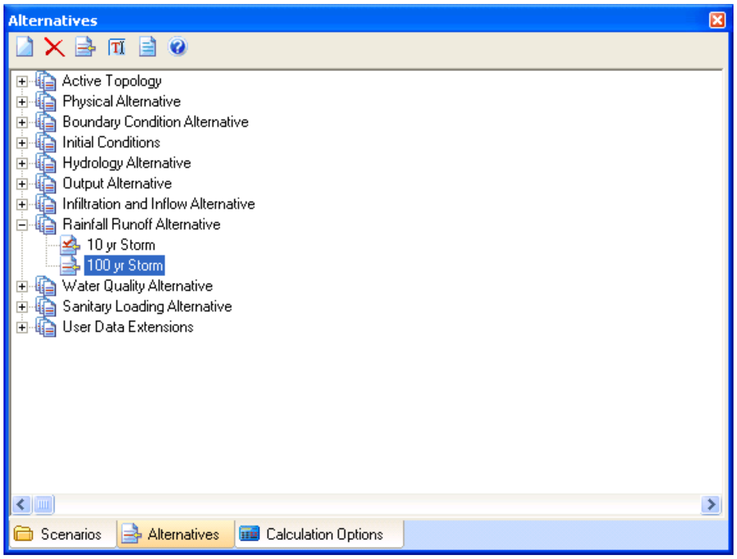


3. Double-click the **10 yr Storm** alternative to open up the **Rainfall Runoff Alternative** editor. Click the **Global Rainfall** list box. You should see the 2 events that were defined in Part 1 of this lesson, along with **Storm Event - 1**, which was used in the earlier lessons. Select the **10 yr Event**. **Close** the **Rainfall Runoff Alternative** editor.

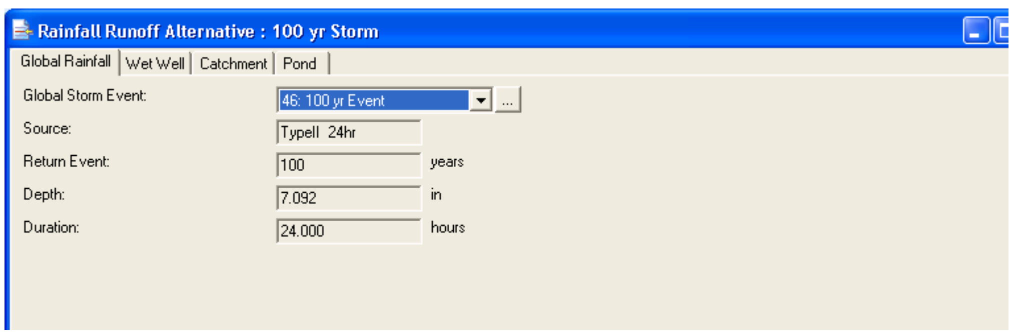


4. In the **Alternatives** manager, highlight the **Rainfall Runoff Alternative** node, then click the **New** button and select the **Base Alternative** command from the submenu that appears.

5. Highlight the newly created base alternative and click the **Rename** button. Type in **100 yr Storm**.

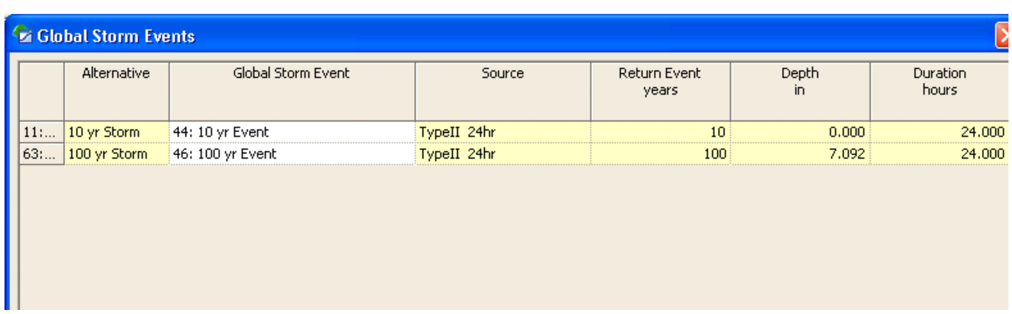


6. Double-click the **100 yr Storm** alternative to open up the **Rainfall Runoff Alternative** editor. Click the **Global Storm Event** list box and select the **100 yr Event**. **Close** the **Rainfall Runoff Alternative** editor.



7. **Close** the **Alternatives** manager.

8. Click the **Components** menu and select the **Global Storm Events** command. The **Global Storm Events** dialog that appears provides a summary of global storm events, their depth, duration, and their original source of distribution data. **Close** the **Global Storm Events** dialog.



This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep the same project open if you plan to continue immediately. If you plan to begin the next part at a later time, you can either save the current project or use the **Lesson_6_3.swg** project located in the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder.

Part 3: Creating Scenarios to Reference Rainfall Runoff Alternatives

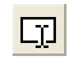
In Part 2 of this lesson, we created Rainfall Runoff Alternatives using Storm Events defined in Part 1. In this lesson, we will create scenarios that reference the Rainfall Runoff Alternatives.

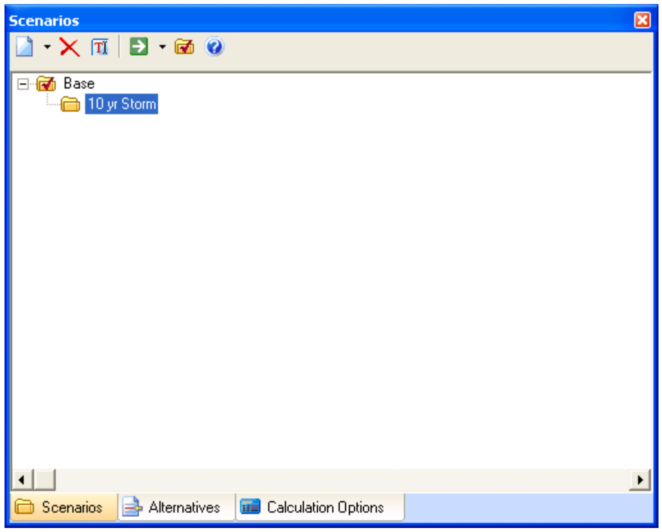
You can reference a single Rainfall Runoff Alternative from an unlimited number of scenarios. For example, you may want to apply the same 100-yr design storm to pre-developed conditions, and two different proposed designs. In each of these three cases, you would reference the same 100-year Rainfall Runoff Alternative. This organization maximizes your flexibility for sharing storm data across different scenarios.

If you've already completed Part 2 of this lesson, you can continue using the same model. Otherwise, begin by clicking the **File/Open** button; then, browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_6_3.swg**, then click **Open.**

1. Click the **Analysis** pulldown menu and select **Scenarios**.
2. In the **Scenarios** manager, highlight the **Base** scenario, then click the **New** button and select **Child Scenario** from the submenu that appears.

3. Highlight the newly created scenario and click the **Rename** button. Type in **10 yr Storm**.

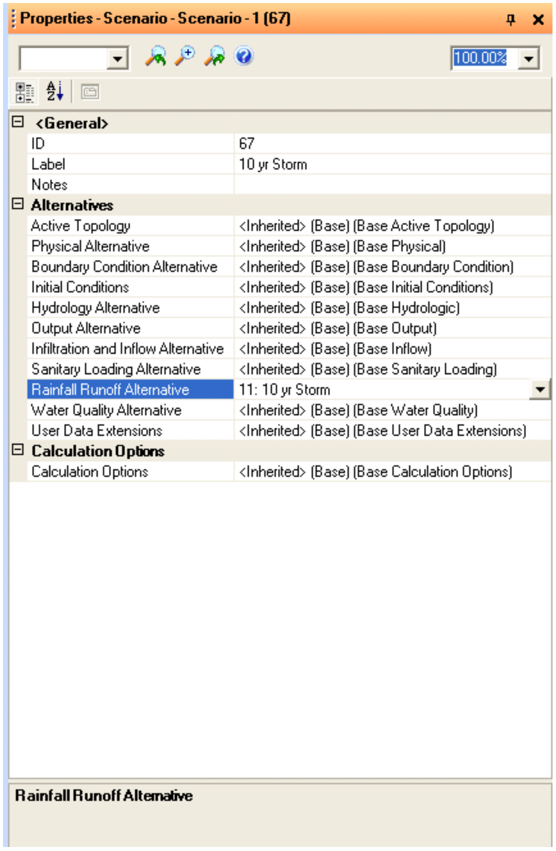




Tip: **When creating design storm scenarios, it is important to include the return event description AND scenario description as part of the child scenario name. By using this convention, you can reference the same design storm from multiple scenarios, while clearly differentiating between each scenario when you click the fall down scenario list on the main tool bar. If you do not add a description in addition to the return event, you could potentially see several "100-Year" events in the fall down scenario list, even though they reference different topological or physical alternatives.**

4. Double-click the **10yr Storm** scenario to bring up the scenario **Property Editor** dialog.

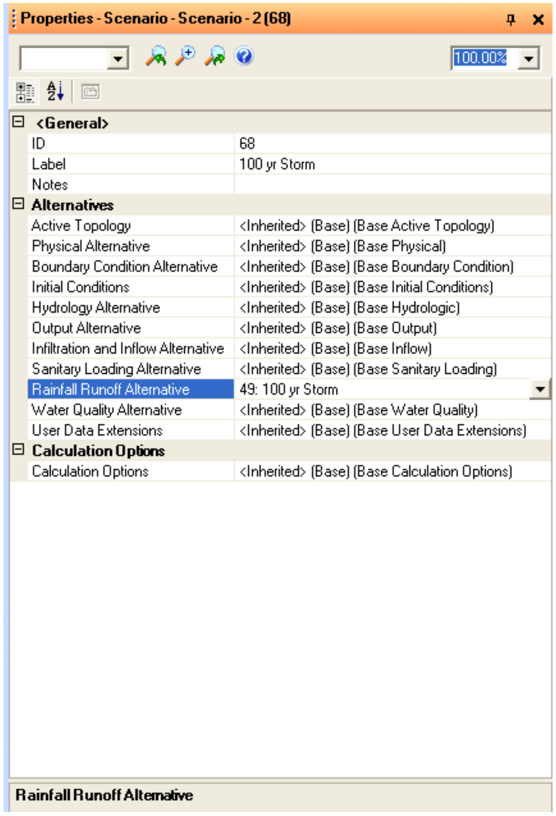
5. In the **Property Editor**, make sure the **Rainfall Runoff Alternative** is set to **10yr Storm**.



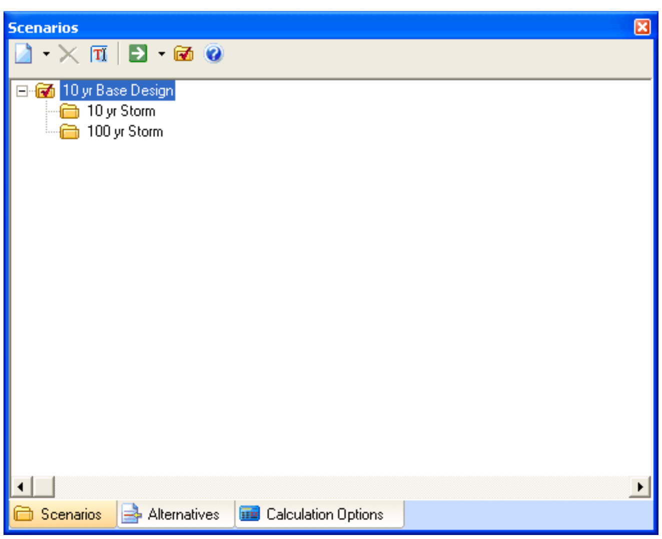
6. In the **Scenarios** manager, highlight the **Base** scenario, then click the **New** button and select **Child Scenario** from the submenu that appears.

7. Highlight the newly created scenario and click the **Rename** button. Type in **100 yr Storm**.

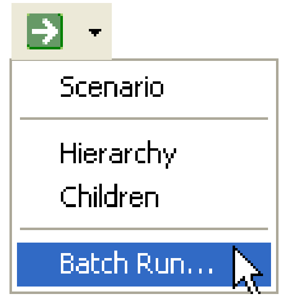
8. Making sure that the **100 yr Storm** scenario is still highlighted in the **Scenarios** manager, set the **Rainfall Runoff Alternative** to **100 yr Storm** in the **Property Editor**.



9. The Base Rainfall Runoff Alternative has been renamed and revised to be a 10 yr Event, so highlight the **Base** scenario in the **Scenarios** manager and click the **Rename** button. Type in **10 yr Base Design**.

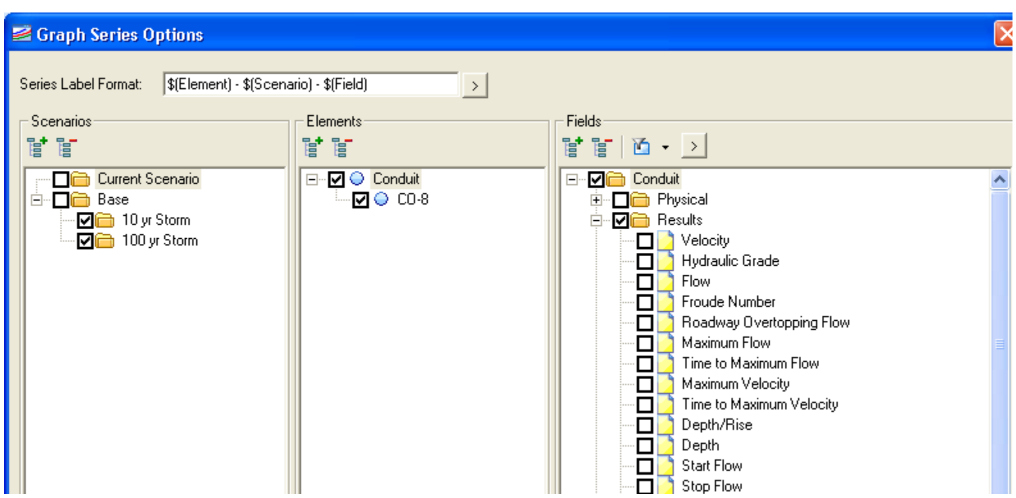


10. In the **Scenarios** manager, click the **Compute** button and select the **Batch Run** command from the submenu that appears.

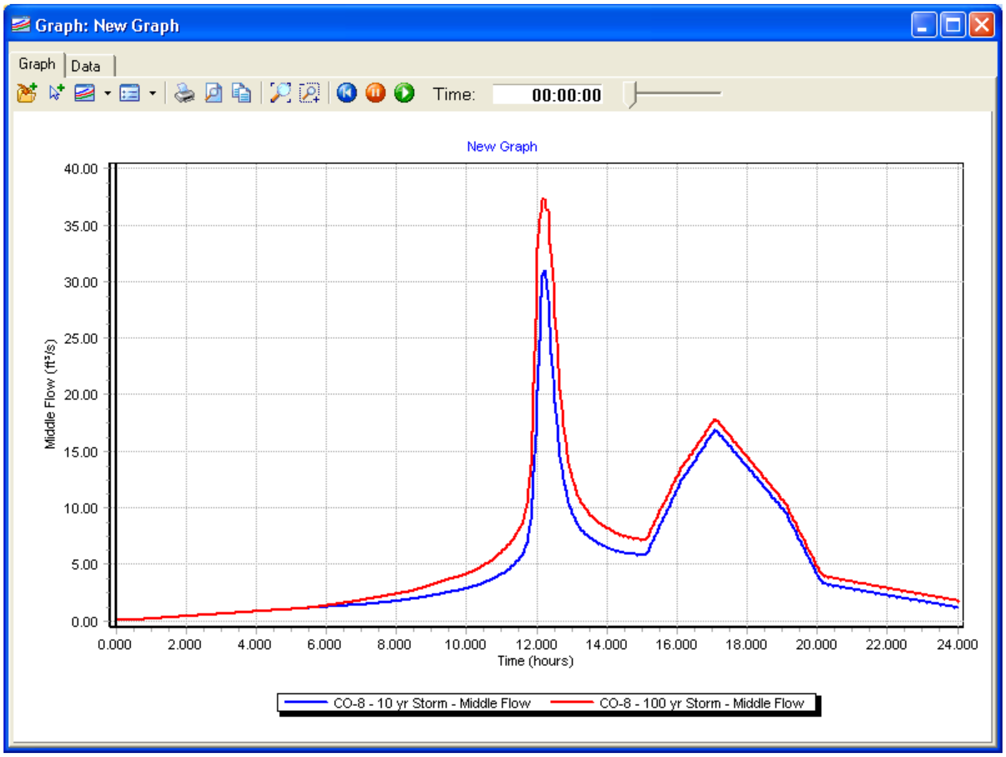


11. The **Batch Run** dialog that appears allows you to calculate a number of scenarios at the same time. Click the checkboxes next to **10 yr Storm** and **100 yr Storm.** Click the **Batch** button.

12. In the **Please Confirm** dialog that appears, click the **Yes** button. After both scenarios have been calculated, click **OK** in the **Information** box that appears.
13. **Close** the **Scenarios** manager.
14. In the **Drawing Pane**, right-click the last conduit before the **Treatment** outfall, **CO-8**, and select the **Graph** command from the shortcut menu that appears.
15. In the **Graph Series Options** dialog that appears, click the **10 yr Storm and 100 yr Storm** checkboxes in the **Scenarios** list pane. Click the **Base** checkbox to clear it. Click the **OK** button.



16. The Graph view dialog now displays the flow for each of the scenarios we computed during the batch run, allowing you to compare the two.



This concludes Lesson 6. The next lesson will use different model files, so save your model even if you plan to continue immediately. Click the **File** pulldown menu and select the **Save As** command. Browse to the **Program Files/Bentley/SewerGEMS/Lessons** folder and enter the name **Lesson_7.swg**, then click **Save**.

Lesson 7: Working With the ArcMap Client

In this lesson, we will discuss using the ArcMap client to:

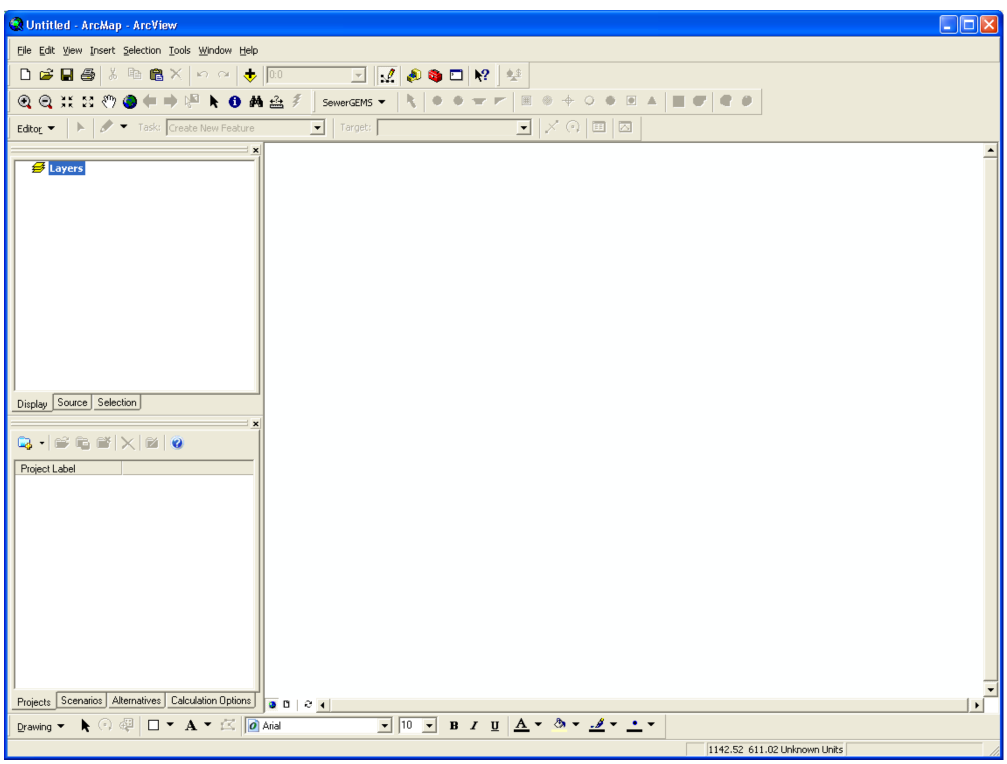
- Customize the ArcMap interface
- Create a new project
- Lay out a model
- Create a project from an existing Bentley SewerGEMS V8*i* project
- Use GeoTables to perform ArcMap functions on live Bentley SewerGEMS V8*i* data

Part 1: Customizing the ArcMap Interface

In this part of the lesson, we will customize the ArcMap interface to display important Bentley SewerGEMS V8*i* dialogs while providing a large area in the display pane for model layout and viewing.

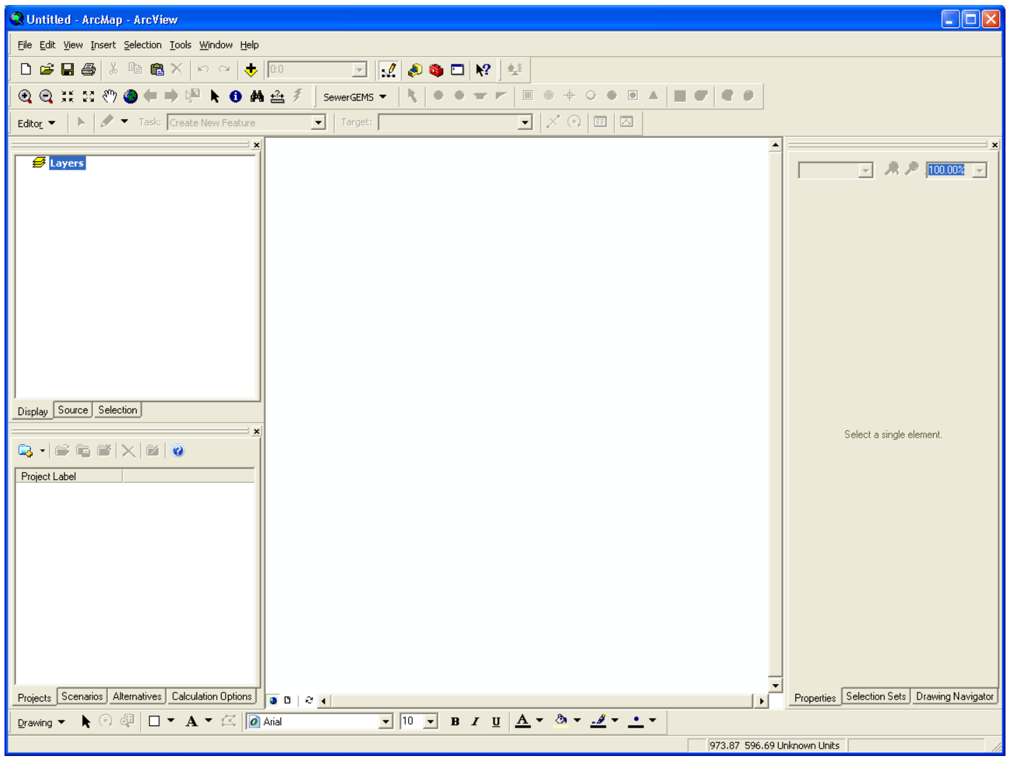
1. Start ArcMap.
2. The first time you start ArcMap after installing Bentley SewerGEMS V8*i*, the Bentley SewerGEMS V8*i* toolbar will be floating (undocked). Dock it in the area of the ArcMap toolbars by clicking on the heading bar and holding the mouse button, then drag it to an empty area of the ArcMap toolbar area and release the mouse button.

3. Click the SewerGEMS V8*i* menu on the SewerGEMS V8*i* toolbar and hover the mouse cursor over the **View** menu, then select the **Project Manager** command.
4. Click the heading bar of the **Project Manager** dialog, hold down the mouse button, and drag the dialog over the middle of the ArcMap Display/Source/Selection dialog, then release the mouse button. Your interface should now look like this:



5. Click the SewerGEMS V8*i* menu on the SewerGEMS V8*i* toolbar and hover the mouse cursor over the **View** menu, then select the **Properties** command.

6. Click the heading bar of the **Properties** dialog, hold down the mouse button, and drag the dialog to the right edge of the ArcMap window, then release the mouse button. Your interface should now look like this:



This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep ArcMap open if you plan to continue immediately. If you plan to begin the next part at a later time, you can close ArcMap and the interface changes you made will be retained next time you open ArcMap.

Part 2: Creating a New Project in ArcMap

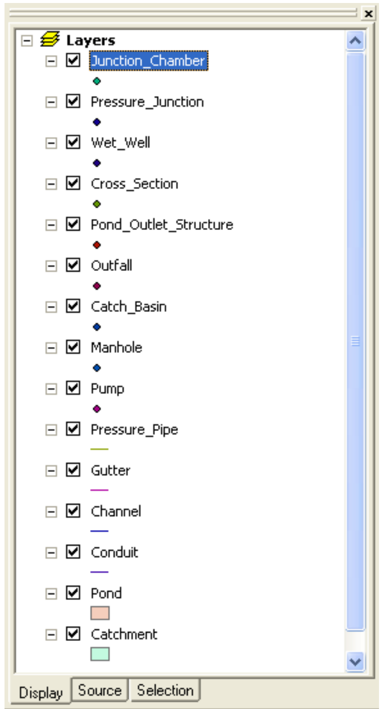
An ArcMap Bentley SewerGEMS V8*i* project consists of:

- **A** Bentley SewerGEMS V8*i* **.mdb file**—this file contains all modeling data, and includes everything needed to perform a calculation.
- **A** Bentley SewerGEMS V8*i* **.swg file**—this file contains data such as annotation and color-coding definitions.
- **A geodatabase association**—a project must be linked to a new or existing geodatabase.

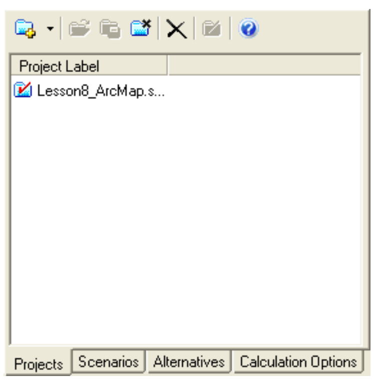
In this part of the lesson, we will be creating an entirely new project and associating it to a new geodatabase. If ArcMap is not already open, start it up now. If you have not yet completed Part 1 of this lesson, do so now so that all of the necessary SewerGEMS V8*i* tools and dialogs are available to you.

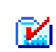
1. In the **Project Manager**, click the **Add Project** button and select **Add New Project** from the submenu that appears.
2. In the **Save As** dialog that appears, browse to the **Bentley/SewerGEMS/Lessons** folder and enter the name **Lesson8_ArcMap**.
3. Click the **Save** button.
4. In the **Attach Geodatabase** dialog that appears, click the **Attach Geodatabase** button. 
5. The **Import Into or Create New Geodatabase** dialog that appears allows you to either select an existing geodatabase to associate the new project with, or to create a new one. Create a new geodatabase for the project by entering the file name **Lesson8_ArcMap_GDB**, then click the **Save** button.
6. In the **Attach Geodatabase** dialog, leave the default **Dataset Name** of **Lesson8_ArcMap**.
7. Click the **OK** button.

8. ArcMap layers for each Bentley SewerGEMS V8*i* element type are automatically created.



9. In addition, the project is listed in the Project Manager.



10. Note the icon next to the project name. This icon indicates that the project is the current project. Only one project can be the current project at one time; if there were more than one project open, only one would display this icon. 

This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep ArcMap open if you plan to continue immediately. If you plan to begin the next part at a later time, you can close ArcMap - the project we will use for the next part of the lesson has been created and there is no need to save the ArcMap .mxd because the project is empty.

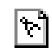
Part 3: Laying out a Model In ArcMap

Laying out a model in the ArcMap client differs from laying one out in Stand-Alone in a couple of different respects. This part of the lesson describes the procedure for laying out polygon, node, and link elements in ArcMap.

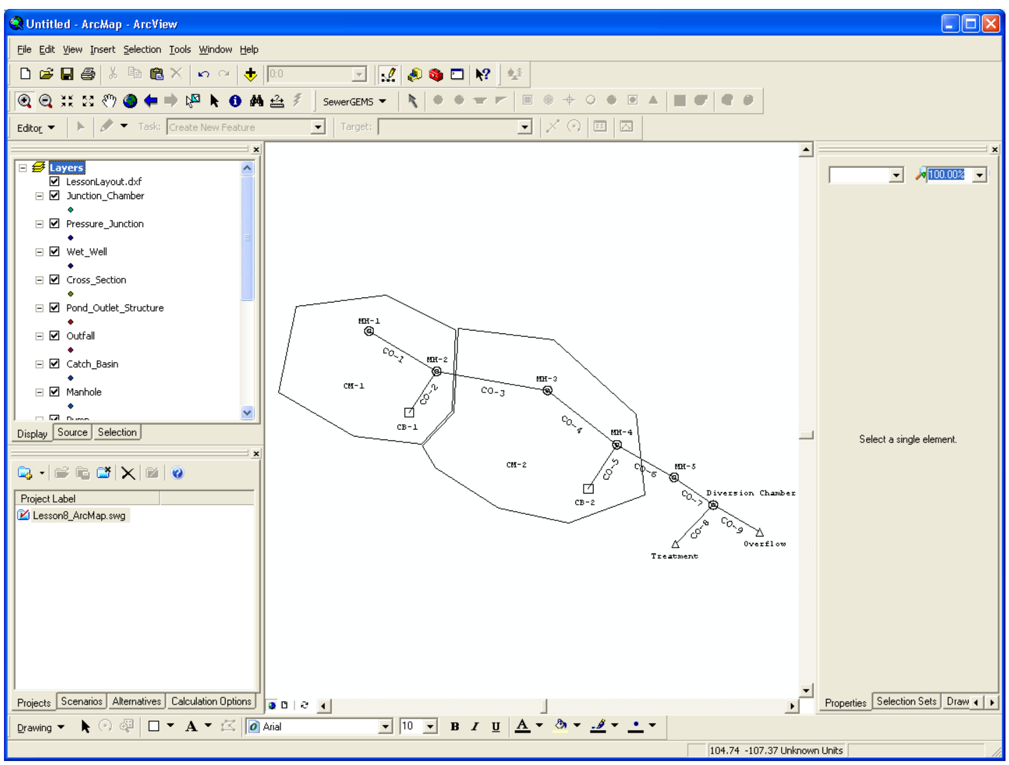
If ArcMap is not already open, start it up now. If you have not yet completed Parts 1 and 2 of this lesson, do so now so that all of the necessary Bentley SewerGEMS V8*i* tools and dialogs are available to you, and so that you have a new project to work with.

1. Click the **Add Data** button.

2. In the **Add Data** dialog that appears, browse to the **Bentley**/SewerGEMS V8*i*/**Lessons** directory. Highlight **LessonLayout.dxf**. The Add Data dialog will display two separate files with this name - use the one with the icon pictured to the right.



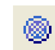
3. Click the **Add** button.



4. To add or move elements, you must be in an edit session. Click the **Editor** button and select the **Start Editing** command from the submenu that appears.

5. Note that the SewerGEMS V8*i* toolbar becomes active. Click the **Catchment** button.

6. Click on one of the corners of the catchment outline for **CM-1**. Drag the mouse to the next corner, and click again.
7. Continue laying out the catchment boundaries by clicking each corner until you click the last one. Right-click and select **Finish Sketch** from the submenu that appears.
8. Click the **Manhole** button.



9. Click on the location indicated by the dxf background for **MH-1** to place a manhole there.
10. Click on the location indicated by the dxf background for **MH-2** to place a manhole there.
11. Click the **Conduit** button.

12. Click on **MH-1**, then click on **MH-2**. Right-click and select **Finish Sketch** to lay out the conduit.
13. Click the **Editor** button and select the **Stop Editing** command. A prompt will appear, asking **Do you want to save your edits?** Click the **Yes** button.
14. We won't be using this project for the remainder of the lesson. In the **Project Manager**, highlight **Lesson8_ArcMap.swg** and click the **Remove Project** button.
15. Note the warning that appears. Be careful with the Remove Project command, because as the warning indicates, the operation irreparably breaks the geodatabase connection. You will still be able to open the project in Stand-Alone mode, but not in ArcMap.
16. Right-click the **Layers** node in the ArcMap **Display** dialog and click the **Select All Layers** command.
17. Right-click one of the selected layers and click the **Remove** command to delete the layers and clear the map.

This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep ArcMap open if you plan to continue immediately. If you plan to begin the next part at a later time, you can close ArcMap - we'll be starting a new project so there is nothing that needs to be saved.

Part 4: Creating A New ArcMap Project From An Existing Bentley SewerGEMS V8i Project

In this part of the lesson, we will be creating a new project in the ArcMap client and associating it to a new geodatabase. If ArcMap is not already open, start it up now. If you have not yet completed Part 1 of this lesson, do so now so that all of the necessary Bentley SewerGEMS V8*i* tools and dialogs are available to you.

1. Click the **Add Project** button and select the **Add Existing Project** command from the submenu that appears.
2. In the **Open** dialog that appears, browse to the **Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_7_Final.swg**, and click the **Open** button.
3. In the **Attach Geodatabase** dialog that appears, click the **Attach Geodatabase** button.

4. The **Import Into or Create New Geodatabase** dialog that appears allows you to either select an existing geodatabase to associate the new project with, or to create a new one. Create a new geodatabase for the project by entering the file name **Lesson7_Final_ArcMap.mdb**, then click the **Save** button.
5. In the **Attach Geodatabase** dialog, leave the default **Dataset Name** of **Lesson_7_Final**.
6. Click the **OK** button.
7. ArcMap layers are created for each element type, and the network is displayed in the drawing pane.

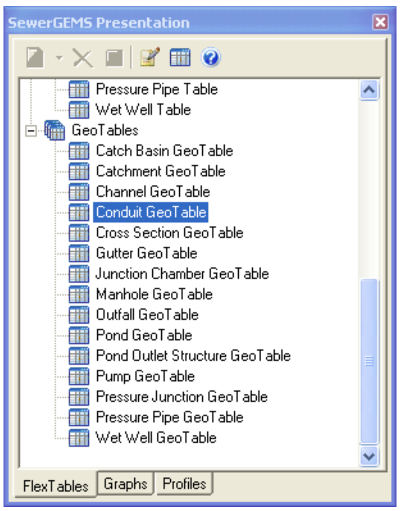
This concludes this part of the lesson. The next part continues where this part leaves off, so you can keep ArcMap open if you plan to continue immediately. If you plan to begin the next part at a later time, you can close ArcMap - the project we will use for the next part of the lesson has been created and there is no need to save the ArcMap .mxd since no changes were made to the map settings.

Part 5: Using GeoTables

GeoTables allow you to use the viewing and rendering tools provided by the ArcMAP environment on all of your Bentley SewerGEMS V8*i* data, including both calculated results and input.

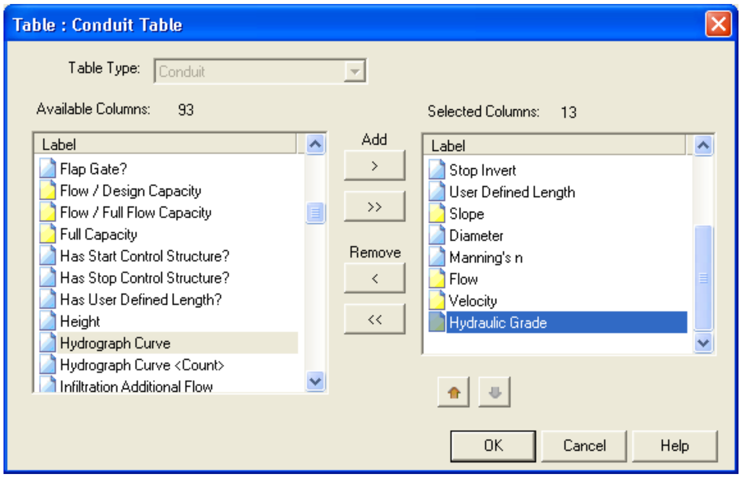
If ArcMap is not already open, start it up now. If you have not yet completed Parts 1 and 4 of this lesson, do so now so that all of the necessary Bentley SewerGEMS V8*i* tools and dialogs are available to you, and so that you have a completed project to work with.

1. Click the SewerGEMS V8*i* menu on the SewerGEMS V8*i* toolbar and hover the mouse cursor over the **Analysis** menu, then select the **Compute** command.
2. Click the SewerGEMS V8*i* menu on the SewerGEMS V8*i* toolbar and hover the mouse cursor over the **View** menu, then select the **FlexTables** command. Leave the FlexTable manager dialog floating (not docked), and position it so that you can still see the map display.
3. In the FlexTables dialog, scroll down to the **GeoTables** node and expand it if necessary by clicking the plus button. Double-click the **Conduit** node under the GeoTables heading.



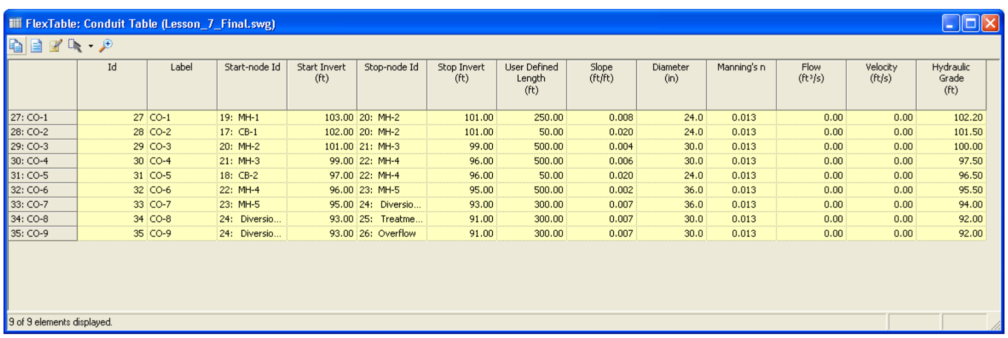
4. In the **FlexTable: Conduit GeoTable** dialog that appears, note the attributes that are currently included. The attributes in the GeoTable are the only ones that are available for use with ArcGIS functions and commands. Click the **Edit** button.

5. The **Table: Conduit GeoTable** dialog that appears allows you to add and remove attribute columns to/from the GeoTable. To add attributes to the GeoTable, you must move the desired attributes from the Available list to the Selected list. Find **Hydraulic Grade** in the **Available** list and double-click it to move it to the **Selected** list.



6. Click **OK**.

7. Note that the **FlexTable: Conduit GeoTable** dialog now contains a column for **Hydraulic Grade**, and the calculated HGL is displayed for each conduit. Close the dialog.


FlexTable: Conduit Table (Lesson_7_Final.swg)

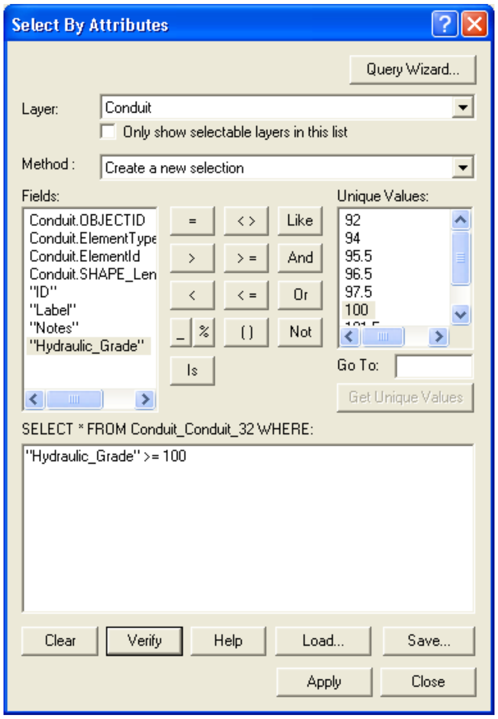
	Id	Label	Start-node Id	Start Invert (ft)	Stop-node Id	Stop Invert (ft)	User Defined Length (ft)	Slope (ft/ft)	Diameter (in)	Manning's n	Flow (ft³/s)	Velocity (ft/s)	Hydraulic Grade (ft)
27: CO-1	27	CO-1	19: MH-1	103.00	20: MH-2	101.00	250.00	0.008	24.0	0.013	0.00	0.00	102.20
28: CO-2	28	CO-2	17: CB-1	102.00	20: MH-2	101.00	50.00	0.020	24.0	0.013	0.00	0.00	101.50
29: CO-3	29	CO-3	20: MH-2	101.00	21: MH-3	99.00	500.00	0.004	30.0	0.013	0.00	0.00	100.00
30: CO-4	30	CO-4	21: MH-3	99.00	22: MH-4	96.00	500.00	0.006	30.0	0.013	0.00	0.00	97.50
31: CO-5	31	CO-5	18: CB-2	97.00	22: MH-4	96.00	50.00	0.020	24.0	0.013	0.00	0.00	96.50
32: CO-6	32	CO-6	22: MH-4	96.00	23: MH-5	95.00	500.00	0.002	36.0	0.013	0.00	0.00	95.50
33: CO-7	33	CO-7	23: MH-5	95.00	24: Diversio...	93.00	300.00	0.007	36.0	0.013	0.00	0.00	94.00
34: CO-8	34	CO-8	24: Diversio...	93.00	25: Treatme...	91.00	300.00	0.007	30.0	0.013	0.00	0.00	92.00
35: CO-9	35	CO-9	24: Diversio...	93.00	26: Overflow	91.00	300.00	0.007	30.0	0.013	0.00	0.00	92.00

9 of 9 elements displayed.

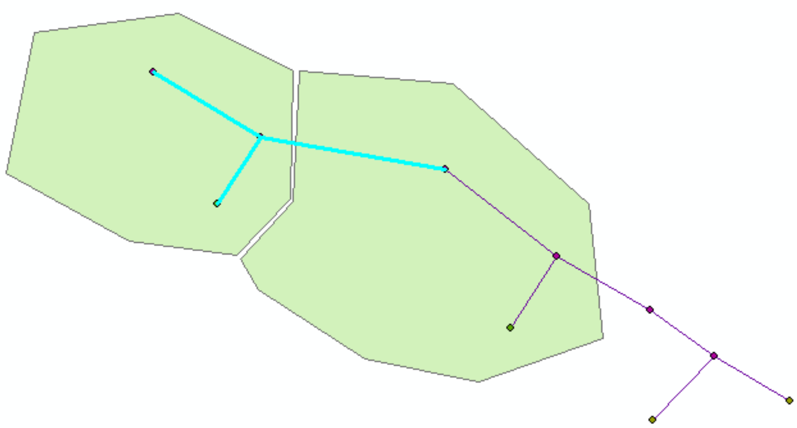
8. The calculated Hydraulic Grade is now available for use with ArcMap commands. Click the ArcMap **Selection** menu and choose the **Select By Attributes** command.

9. In the **Select By Attributes** dialog that appears, click the **Layer** pulldown menu and select **Conduit**. Double-click **"Hydraulic_Grade"** in the **Fields** list to add it to the query statement pane. Click the >= button to add it to the query. Click the **Get Unique Values** button, then double-click **100** in the **Unique Values** list.

10. Click the **Verify** button; you should receive the message **The expression was successfully verified**. Click the **Apply** button.

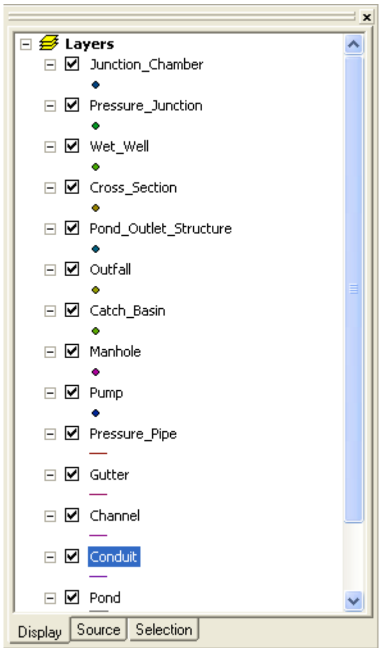


11. Click the **Close** button. Note that three of the conduits in the drawing pane are selected.



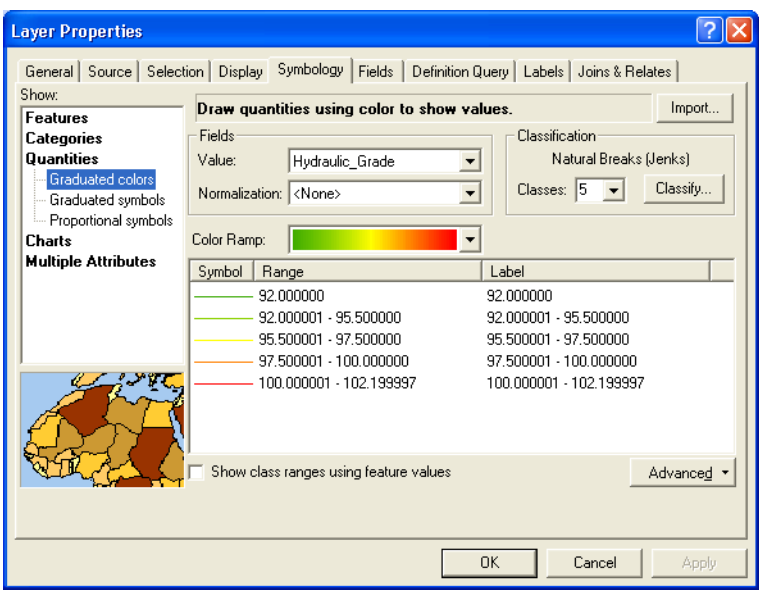
12. Click the ArcGIS **Selection** menu and choose the **Clear Selected Features** command.

13. You can also apply ArcMap symbology settings based on Bentley SewerGEMS V8*i* attributes that have been added to the appropriate GeoTable. Double click the **Conduit** layer in the ArcMap **Layers** dialog.

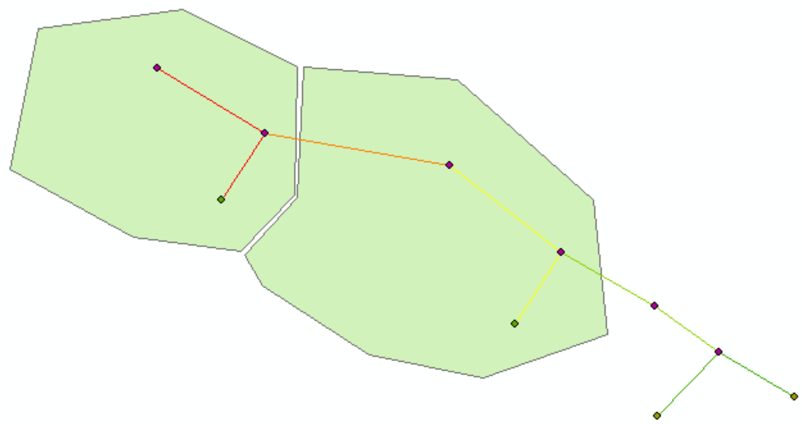


14. In the **Layer Properties** dialog that appears, click the **Symbology** tab.

15. Click **Quantities** in the **Show:** list, then highlight **Graduated Colors**. Click the **Value:** pulldown menu and select **Hydraulic_Grade**. Change the Color Ramp setting if desired. Leave the **Normalization** and **Classes** fields set to their defaults.



16. Click the **OK** button.
17. Note that the conduits are now color-coded according to their calculated hydraulic grade.



Lesson 8: Adding Hydrographs Using the RTK Runoff Method

The RTK method is used to generate a hydrograph based on precipitation data. It forms the hydrograph by combining triangular hydrographs from three components of flow:

- Rapid inflow
- Moderate infiltration
- Slow infiltration

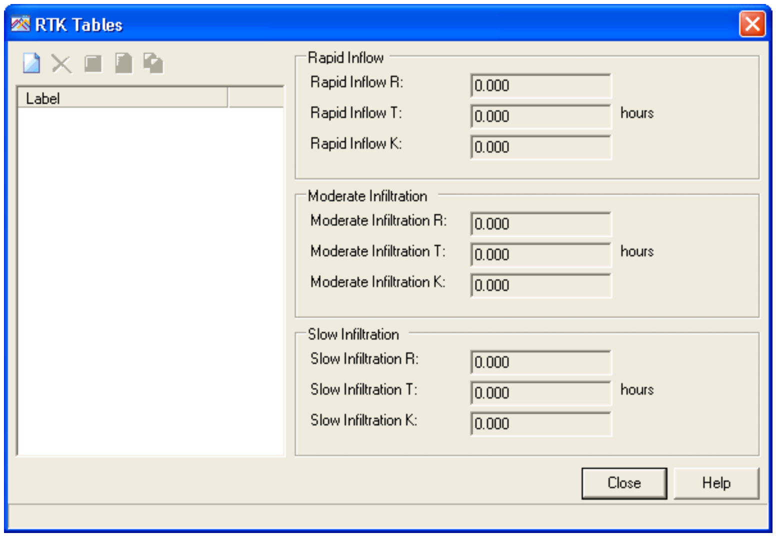
In this lesson, we will define RTK tables and assign them to the catchments in a model.

Let's begin by starting up Bentley SewerGEMS V8*i*. If SewerGEMS V8*i* is already open, skip step one and click the **File** pulldown menu and select the **Open** command instead, then proceed to step two.

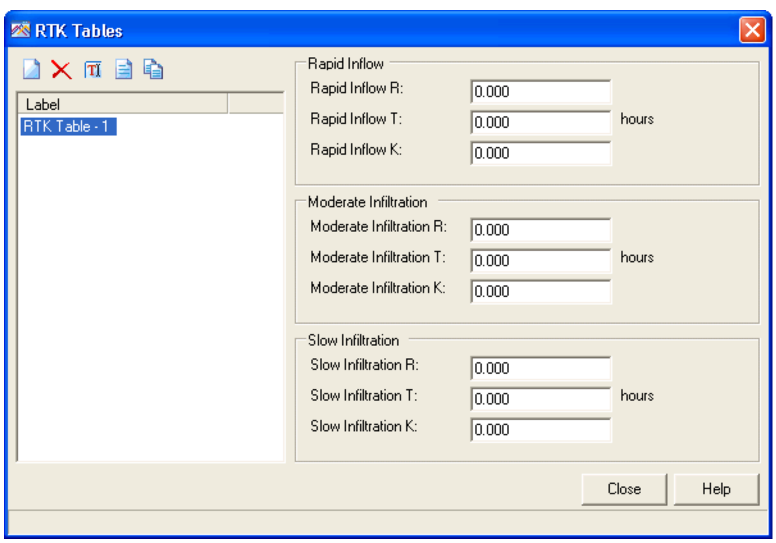
1. In the Welcome to Bentley SewerGEMS V8*i* dialog that appears, click the **Open Existing Project** button.
2. Browse to the **C:/Program Files/Bentley**/SewerGEMS V8*i*/**Lessons** folder, highlight **Lesson_8.swg**, then click **Open**.

First, create the RTK tables that will be applied to the catchments in the model. The RTK parameters are a property of each catchment. However, it is not uncommon for many catchments with similar characteristics to share the same RTK parameters. Therefore, the RTK parameters are entered in a named RTK table and that table can be shared among many catchments.

3. Click the **Components** menu and select the **RTK Tables** command.



4. In the **RTK Tables** dialog that appears, click the **New** button.



5. Click the newly created table to select it if it's not already highlighted, and click the **Rename** button.

6. Enter the name **CM-1**.

7. With **CM-1** still highlighted, enter the RTK values for the rapid inflow component of flow. The R, T, and K values for rapid inflow represent the following attributes:
   - R—Fraction of precipitation that enters the collection system for rapid inflow.
   - T—The time from the precipitation pulse to the peak of rapid inflow of the hydrograph.
   - K—The ratio of the time to peak to time to end of hydrograph for rapid inflow.

   Enter the values of R, T, and K for the rapid inflow component of flow in the corresponding fields of the RTK Tables dialog as follows:

Table 3-10: Rapid Inflow RTK Values for CM-1

Attribute	Value
Rapid Inflow R	0.020
Rapid Inflow T	2.000
Rapid Inflow K	1.400

8. Next, with **CM-1** still highlighted, enter the RTK values for the moderate infiltration component of flow. The R, T, and K values for moderate infiltration represent the following attributes:
   - R—Fraction of precipitation that enters the collection system for moderate infiltration.
   - T—The time from the precipitation pulse to the peak of moderate infiltration of the hydrograph.
   - K—The ratio of the time to peak to time to end of hydrograph for moderate infiltration.

9. Enter the values of R, T, and K for the moderate infiltration component of flow in the corresponding fields of the RTK Tables dialog as follows:

Table 3-11: Moderate Infiltration RTK Values for CM-1

Attribute	Value
Moderate Infiltration R	0.050
Moderate Infiltration T	3.000
Moderate Infiltration K	1.600

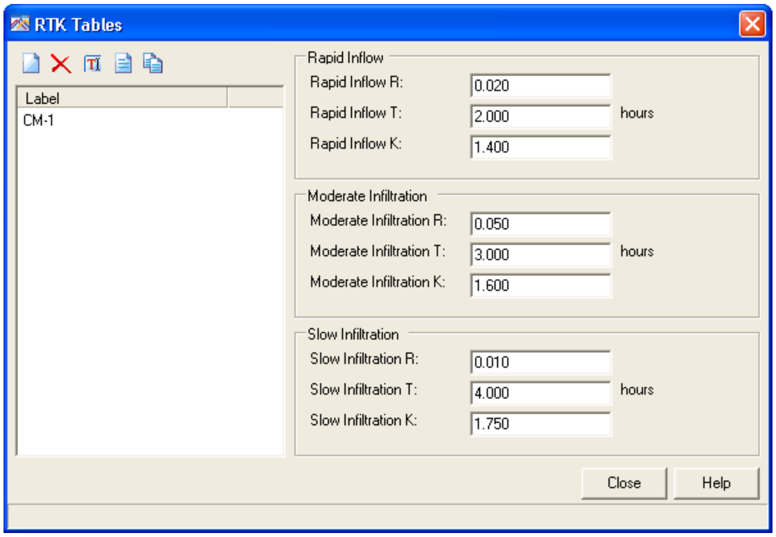
10. Finally, with **CM-1** still highlighted, enter the RTK values for the slow infiltration component of flow. The R, T, and K values for slow inflow represent the following attributes:
    - R—Fraction of precipitation that enters the collection system for slow infiltration.
    - T—The time from the precipitation pulse to the peak of slow infiltration of the hydrograph.
    - K—The ratio of the time to peak to time to end of hydrograph for slow infiltration.

11. Enter the values of R, T, and K for the slow infiltration component of flow in the corresponding fields of the RTK Tables dialog as follows:

Table 3-12: Slow Infiltration RTK Values for CM-1

Slow Infiltration R	0.010
Slow Infiltration T	4.000
Slow Infiltration K	1.750

12. The RTK table for CM-1 should now look like this:

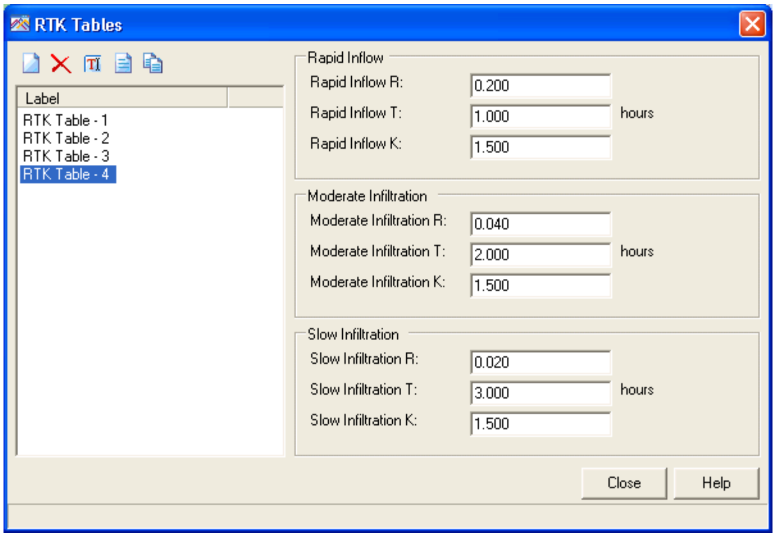


13. Follow steps 4-11 to create three more tables for CM-2, CM-3, and CM-4 using the following values:

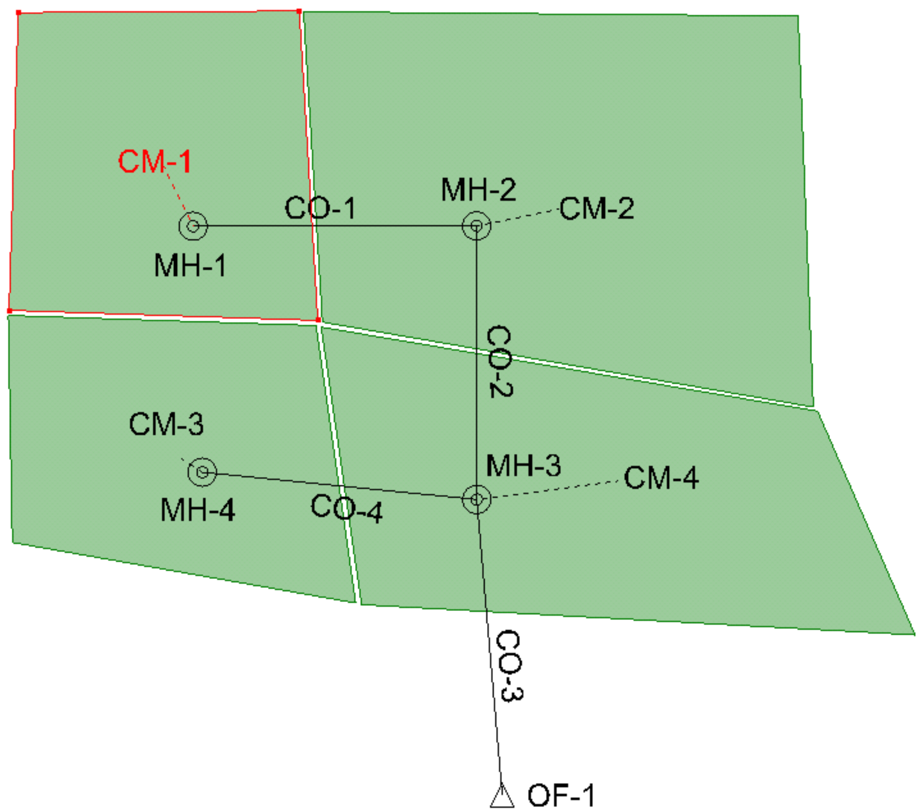
Table 3-13:

Attribute	CM-2 Value	CM-3 Value	CM-4 Value
Rapid Inflow R	0.001	0.120	0.200
Rapid Inflow T	0.200	1.000	1.000
Rapid Inflow K	2.000	1.500	1.500
Moderate Infiltration R	0.002	0.040	0.040
Moderate Infiltration T	1.000	2.000	2.000
Moderate Infiltration K	2.000	1.500	1.500
Slow Infiltration R	0.140	0.020	0.020
Slow Infiltration T	2.000	3.000	3.000
Slow Infiltration K	1.750	1.500	1.500

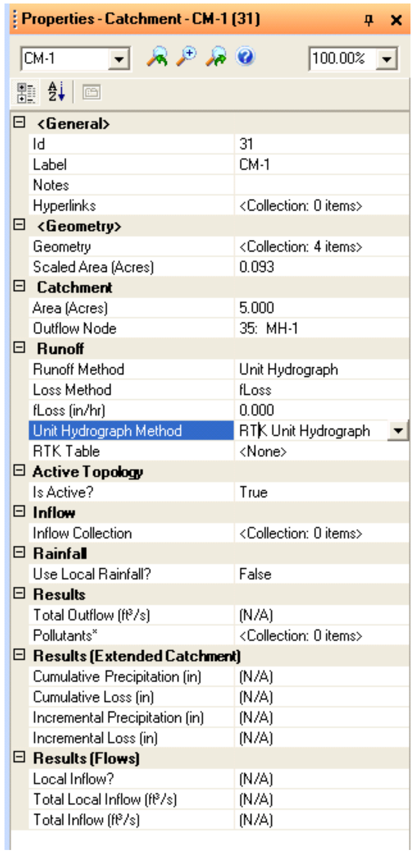
14. The RTK Tables dialog should now look like this:



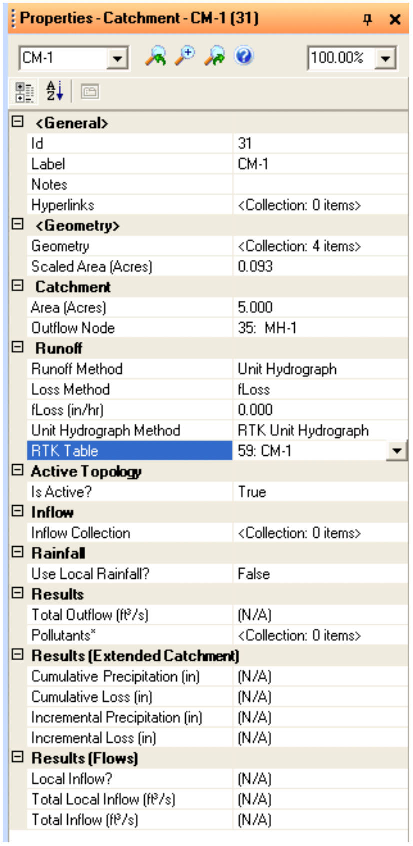
15. The RTK tables can now be assigned to the catchments in the model. Click the Close button to close the RTK Tables dialog. In the drawing pane, highlight catchment CM-1.



16. With **CM-1** highlighted, find the **Unit Hydrograph Method** attribute in the **Property Editor**. Click the pulldown menu and change the value to **RTK Unit Hydrograph**.



17. Click the pulldown menu in the **RTK Table** attribute field of the **Property Editor** and select **CM-1**.



18. Repeat steps 15-17 for catchments CM-2, CM-3, and CM-4, assigning the correspondingly named RTK tables to each catchment.

19. Click the Compute button and review the results using Reports, FlexTables, Graphs, Profiles, Annotation, and Color Coding as described in "Lesson 5: Presenting Calculated Results" on page 3-84.

This concludes the QuickStart Lessons. For more information on any of Bentley SewerGEMS V8*i*'s functions, you can right-click or press the F1 key to access the context-sensitive online help at any time.

Chapter

Starting a Project 4

Welcome Dialog Box

When you first start Bentley SewerGEMS V8*i*, the Welcome dialog box appears. The Welcome dialog box contains the following controls:

Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Create New Project	Creates a new Bentley SewerGEMS V8*i* project. When you click this button, an untitled Bentley SewerGEMS V8*i* project is created.
Open Existing Project	Opens an existing project. When you click this button, the Windows Select Bentley SewerGEMS V8*i* Project to Open dialog box appears, allowing you to browse to the project to be opened.
Open from ProjectWise	Open an existing Bentley SewerGEMS V8*i* project from ProjectWise. You are prompted to log into a ProjectWise datasource if you are not already logged in.
Show This Dialog at Start	When selected, the Welcome dialog box appears whenever you start Bentley SewerGEMS V8*i*. Clear this box if you do not want the Welcome dialog box to appear whenever you start Bentley SewerGEMS V8*i*.

You can access the Welcome dialog box at any time from the Help menu in Bentley SewerGEMS V8*i*.

Projects

All data for a model are stored in Bentley SewerGEMS V8*i* as a project. Bentley SewerGEMS V8*i* project files have the file name extension .swg. Bentley Sewer-GEMS V8*i* lets you open more than one project at a time. You can assign a title, date, notes and other identifying information about each project using the Project Properties dialog box. You can have up to five Bentley SewerGEMS V8*i* projects open at one time.

Starting a New Project

To start a new project, select **File > New** or press **Ctrl+N**. An untitled project is opened in the drawing pane.

Opening an Existing Project

To open an existing project, select **File > Open** or press **Ctrl+O**. A dialog box appears allowing you to browse for the project you want to open.

Displaying Multiple Projects

To switch between multiple open projects, click the appropriate tab at the top of the drawing pane. The file name of the project is displayed on the tab.

Setting Project Properties

The Project Properties dialog box let you enter project-specific information to help identify the project. Project properties are stored with the project.

To set project properties:

Select **File > Project Properties**, enter information in the Project Properties dialog box and click **OK**.

Project Properties Dialog Box

The dialog box contains the following text fields and controls:

Title	Lets you type a title for the project.
File Name	Displays the file name for the current project. If you have not saved the project yet, the file name is listed as “Untitled.swg.”
Engineer	Lets you type name of the project engineer.

Company	Lets you type the name of your company.
Date	Click this field to display a calendar, which lets you use your mouse to set a date for the project.
Notes	Lets you type additional information about the project.

Setting Options

You can change global settings for Bentley SewerGEMS V8*i* in the Options dialog box. The Options dialog box contains four tabs, each of which lets you change a different group of global settings.

Click one of the following links to learn more about the Options dialog box:

- "Options Dialog Box - Global Tab" on page 4-147
- " Options Dialog Box - Project Tab"
- "Options Dialog Box - Drawing Tab" on page 4-151
- "Options Dialog Box - Units Tab" on page 4-152
- "Options Dialog Box - Labeling Tab" on page 4-155
- "Options Dialog Box - ProjectWise Tab" on page 4-156

Options Dialog Box - Global Tab

The Global tab lets you change general program settings for the Bentley SewerGEMS V8*i* stand-alone editor, including whether or not to display the status pane, as well as window color and layout settings.

The Global tab contains the following controls:

General Settings

Backup Levels	Indicates the number of backup copies that are retained when a project is saved. The default value is 1. **Note: The higher this number, the more .BAK files (backup files) are created, thereby using more hard disk space on your computer.**
Show recently used files	The checkbox turns the list of recently opened files on and off. The File menu has the ability to display a list of recently opened files, providing shortcuts that let you quickly access projects. When this check box is cleared, these shortcuts are not available from the File menu. When the box is checked, you can specify a number of files between 1 and 15 to show by typing the number in the adjacent field.
Compact Database After	When selected, the Bentley SewerGEMS V8*i* database is automatically compacted when you save a particular file the specified number of times.
Status Pane	When selected, activates the Status Pane display at the bottom of the Bentley SewerGEMS V8*i* stand-alone editor. This check box is selected by default.
Show Welcome Page on Startup	When selected, activates the Welcome dialog that appears when you first start Bentley SewerGEMS V8*i*. This check box is selected by default.
Zoom Extents on Open	When this box is checked a Zoom Extents operation is performed upon fiel open, so that the entire network is displayed in the drawing pane.

Window Color Settings

Background Color	Displays the color that is currently assigned to the drawing pane background. You can change the color by clicking the ellipsis button (...) to open the Color dialog box.
Foreground Color	Displays the color that is currently assigned to elements and labels in the drawing pane. You can change the color by clicking the ellipsis button (...) to open the Color dialog box.
Read Only Background Color	Displays the color that is currently assigned to data field backgrounds. You can change the color by clicking the ellipsis button (...) to open the Color dialog box.
Read Only Foreground Color	Displays the color that is currently assigned to data field text. You can change the color by clicking the ellipsis button (...) to open the Color dialog box.
Selection	Displays the color that is currently assigned to elements that are selected in the drawing pane. You can change the color by clicking the ellipsis button (...) to open the Color dialog box.

Layout Settings

Display Inactive Topology	When selected, activates the display of inactive element in the drawing pane in the color defined in lthe adjacent color box. When not selected, inactive elements will not be visible in the drawing pane. This check box is selected by default.
Auto Refresh	Activates Auto Refresh. When Auto-Refresh is active, the drawing pane automatically updates whenever changes are made to the Bentley SewerGEMS V8*i* datastore. This check box is selected by default.

Sticky Tool Palette	When selected, activates the Sticky Tools feature. When Sticky Tools is activated, the drawing pane cursor does not reset to the Select tool after you create a node or finish a pipe run in your model, allowing you to continue dropping new elements into the drawing without re-selecting the tool. When Sticky Tools is not activated, the drawing pane cursor resets to the Select tool after you create a node. This check box is selected by default.
Select Polygons by Edge	When selected, lets you select polygons in your model at their edges instead of anywhere inside the polygon. This check box is cleared by default.
Selection Handle Size In Pixels	Specifies, in pixels, the size of the handles that appear on selected elements. Enter a number from 1 to 10.
Default Drawing Style for New Projects	Choose the style in which elements are displayed in the drawing pane. Under GIS style, the size of element symbols in the drawing pane will remain the same regardless of zoom level. Under CAD style, element symbols will appear larger or smaller depending on zoom level.

Options Dialog Box - Project Tab

Geospatial Settings

Spatial Reference	Used for integration with Projectwise Geospatial. You can leave the field blank if there is no spatial information.

Element Labeling Options

Element Identifier Format	Specifies the format in which reference fields are used. Reference fields are fields that link to another element or support object (pump definitions, patterns, controls, zones, etc.).

Result Files

Specify Custom Results File Path?	When checked, allows you to edit the results file path and format by enabling the other controls in this section.
Root Path	Allows you to specify the root path where results files are stored. You can type the path manually or choose the path from a Browse dialog by clicking the ellipsis (...) button.
Path Format	Allows you to specify the path format. You can type the path manually and use predefined attributes from the menu accessed with the [>] button..
Path	Displays a dynamically updated view of the custom result file path based on the settings in the Root Path and Path Format fields.

Options Dialog Box - Drawing Tab

This tab contains drawing layout and display settings. You can set the scale that you want to use as the finished drawing scale for the plan view output. Drawing scale is based upon engineering judgment and the destination sheet sizes to be used in the final presentation. The Drawing tab contains the following controls:

Drawing Scale Settings

Drawing Mode	Drop-down list that lets you select either **Scaled** or **Schematic** mode for models in the drawing pane.
Horizontal Scale Factor 1 in. =:	Controls the scale of the plan view. This value affects the text height when printing-to-scale and does not affect the scaled length or area results.

Annotation Multipliers Settings

Symbol Size Mulitplier	Increases or decreases the size of your symbols by the factor indicated. For example, a multiplier of 2 would result in the symbol size being doubled. The program selects a default symbol height that corresponds to 4.0 ft. (approximately 1.2 m) in actual-world units, regardless of scale.

Text Height Multiplier	Increases or decreases the default size of the text associated with element labeling by the factor indicated. The program automatically selects a default text height that displays at approximately 2.5 mm (0.1 in) high at the user-defined drawing scale. A scale of 1.0 mm = 0.5 m, for example, results in a text height of approximately 1.25 m. Likewise, a 1 in. = 40 ft. scale equates to a text height of around 4.0 ft.

Pipe Text Setting

Align Text with Pipes	Turns text alignment on and off. When this check box is selected, labels are aligned to their associated pipes. When the check box is cleared, labels are displayed horizontally near the center of the associated pipe.
Color Element Annotations	When this box is checked, color coding settings are applied to the element annotation.

Options Dialog Box - Units Tab

The Units tab lets you modify the unit settings for the current project. The Units tab contains the following controls:

Save As	Lets you save the current unit settings as a separate .xml file. This file allows you to reuse your Units settings in another project. When the button is clicked, a Windows Save As dialog box appears, allowing you to enter a name and specify the directory location of the .xml file.
Load	Lets you load a previously created Units project .xml file, thereby transferring the unit and format settings that were defined in the previous project. When the button is clicked, a Windows Load dialog box appears, allowing you to browse to the location of the desired .xml file.
Reset Defaults - SI	Resets the unit and formatting settings to the original factory defaults for the System International (Metric) system.

Reset Defaults - US	Resets the unit and formatting settings to the original factory defaults for the Imperial (U.S.) system.
Default Unit System for New Project	Lets you specify the unit system that is used globally across the project. Note that you can locally change any number of attributes to use system other than the one specified here.

Units Table

The units table contains the following columns:

- **Label**—Displays the parameter measured by the unit.
- **Unit**—Displays the type of measurement. To change the unit of an attribute type, click the choice list and click the unit you want. This option also allows you to use both U.S. customary and SI units in the same worksheet.
- **Display Precision**—Sets the rounding of numbers and number of digits displayed after the decimal point. Enter a negative number for rounding to the nearest power of 10: (-1) rounds to 10, (-2) rounds to 100, (-3) rounds to 1000, and so on. Enter a number from 0 to 15 to indicate the number of digits after the decimal point.
- **Format Menu**—Lets you select the display format used by the current field. Choices include:
  - **Scientific**—Converts the entered value to a string of the form "-d.ddd...E+ddd" or "-d.ddd...e+ddd", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative.
  - **Fixed Point**—Abides by the display precision setting, and automatically enters zeros after the decimal place to do so. With a display precision of 3, an entered value of 3.5 displays as 3.500.
  - **General**—Truncates any zeros after the decimal point, regardless of the display precision value. With a display precision of 3, the value that would appear as 5.200 in Fixed Point format displays as 5.2 when using General format. The number is also rounded. So, an entered value of 5.35 displays as 5.4, regardless of the display precision.
  - **Number**—Converts the entered value to a string of the form "-d,ddd,ddd.ddd...", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. Thousand separators are inserted between each group of three digits to the left of the decimal point.

Options Dialog Box - Labeling Tab

The Element Labeling tab is used to specify the automatic numbering format of new elements as they are added to the network. You can save your settings to an .xml file for later use. The Element Labeling tab contains the following controls:

Save As	Lets you save your element labeling settings to an element label project file, which is an. xml file.
Load	Lets you open an existing element label project file.
Reset	Fills in the **Next** column for each element based on the labels already used in the model and the increment value that has been set. So, for example, if the model contains conduits CO-1, CO-2, and CO-3 and the Increment value is 1, clicking Reset will fill in the Next column with CO-4 for the Conduit row.

Labeling Table

The labeling table contains the following columns:

- **Element**—Shows the type of element to which the label applies.
- **On**—Lets you turn automatic element labeling on and off for the associated element type.
- **Next**—Lets you enter the integer you want to use as the starting value for the ID number portion of the label. Bentley SewerGEMS V8*i* generates labels beginning with this number and chooses the first available unique label.
- **Increment**—Lets you enter the integer that is added to the ID number after each element is created to yield the number for the next element.
- **Prefix**—Lets you enter the letters or numbers that appear in front of the ID number for the elements in your network.
- **Digits**—Lets you enter the minimum number of digits that the ID number has. For instance, 1, 10, and 100 with a digit setting of two would be 01, 10, and 100.
- **Suffix**—Lets you enter the letters or numbers that appear after the ID number for the elements in your network.
- **Preview**—Lets you see what the label looks like, based on the information you have entered in the previous fields.

Options Dialog Box - ProjectWise Tab

Note: These settings affect ProjectWise users only.

The ProjectWise tab contains options for using SewerGEMS V8*i* with ProjectWise.

This tab contains the following controls:

Default Datasource	Displays the current ProjectWise datasource. If you have not yet logged into a datasource, this field will display <login>. To change the datasource, click the **Ellipses (...)** button to open the Change Datasource dialog box. If you click **Cancel** after you have changed the default datasource, the new default datasource is retained.
Update server on Save	When this is checked, any time you save your SewerGEMS V8*i* project locally using the File > Save menu command, the files on your ProjectWise server will also be updated and all changes to the files will immediately become visible to other ProjectWise users. This option is turned off by default.

Note: This option, when turned on, can significantly affect performance, especially for large, complex projects.

For more information about using SewerGEMS V8*i* with ProjectWise, see "Considerations for ProjectWise Users" on page 4-158.

Considerations for ProjectWise Users

Bentley ProjectWise provides managed access to SewerGEMS V8*i* content within a workgroup, across a distributed organization, or among collaborating professionals. When ProjectWise is integrated with SewerGEMS V8*i*, project files can be accessed quickly, checked out for use, and checked back in directly from within SewerGEMS V8*i*.

If ProjectWise is installed on your system, SewerGEMS V8*i* automatically installs all the components necessary for you to use ProjectWise to store and share your SewerGEMS V8*i* projects.

To learn more about ProjectWise, refer to the ProjectWise online help.
To learn more about using ProjectWise with SewerGEMS V8*i*, see the following topics:

- "General Guidelines for using ProjectWise" on page 4-158
- "Performing ProjectWise Operations" on page 4-159

General Guidelines for using ProjectWise

Follow these guidelines when using SewerGEMS V8*i* with ProjectWise:

- Use the File > ProjectWise commands to perform ProjectWise file operations, such as Save, Open, and Change Datasource.
- The first time you choose one of the File > ProjectWise menu commands in your current SewerGEMS V8*i* session, you are prompted to log into a ProjectWise datasource. The datasource you log into remains the current datasource until you change it using the File > ProjectWise > Change Datasource command.
- Use SewerGEMS V8*i*'s File > New command to create a new project. The project is not stored in ProjectWise until you select File > ProjectWise > Save As.
- Use SewerGEMS V8*i*'s File > Open command to open a local copy of the current project.
- Use SewerGEMS V8*i*'s File > Save command to save a copy of the current project to your local computer.
- When you Close a project already stored in ProjectWise using File > Close, you are prompted to select one of the following options:
  - **Check In**—Updates the project in ProjectWise with your latest changes and unlocks the project so other ProjectWise users can edit it.
  - **Unlock**—Unlocks the project so other ProjectWise users can edit it but does not update the project in ProjectWise. Note that this will abandon any changes you have made since the last server update.

- **Leave Out**—Leaves the project checked out so others cannot edit it and retains any changes you have made since the last server update to the files on your local computer. Select this option if you want to exit Bentley Sewer-GEMS V8*i* but continue working on the project later.

- In the SewerGEMS V8*i* Options dialog box, there is a ProjectWise tab with the Update server on Save check box. This option, when turned on, can significantly affect performance, especially for large, complex projects. When this is checked, any time you save your SewerGEMS V8*i* project locally using the File > Save menu command, the files on your ProjectWise server will also be updated and all changes to the files will immediately become visible to other ProjectWise users. This option is turned off by default.
- In this release of SewerGEMS V8*i*, calculation result files are not managed inside ProjectWise. A local copy of results is maintained on your computer, but to ensure accurate results you should recalculate projects when you first open them from ProjectWise.
- SewerGEMS V8*i* projects associated with ProjectWise appear in the Most Recently Used Files list (at the bottom of the File menu) in the following format:

  pwname://PointServer:_TestDatasource/Documents/TestFolder/Test1.prj

Performing ProjectWise Operations

You can quickly tell whether or not the current SewerGEMS V8*i* project is in Project-Wise or not by looking at the title bar and the status bar of the SewerGEMS V8*i* window. If the current project is in ProjectWise, “pwname://” will appear in front of the file name in the title bar, and a ProjectWise icon will appear on the far right side of the status bar, as shown below.

You can perform the following ProjectWise operations from within SewerGEMS V8*i*:

To save an open SewerGEMS V8*i* project to ProjectWise:

1. In SewerGEMS V8*i*, select **File** > **ProjectWise** > **Save As**.
2. If you haven’t already logged into ProjectWise, you are prompted to do so. Select a ProjectWise datasource, type your ProjectWise user name and password, then click **Log in**.

3. In the ProjectWise Save Document dialog box, enter the following information:
    a. Click **Change** next to the Folder field, then select a folder in the current ProjectWise datasource in which to store your project.
    b. Type the name of your SewerGEMS V8*i* project in the Name field. We recommend that you keep the ProjectWise name the same as or as close to the SewerGEMS V8*i* project name as possible.
    c. Keep the default entries for the rest of the fields in the dialog box
    d. Click **OK**.

To open a SewerGEMS V8*i* project from a ProjectWise datasource:

1. Select **File** > **ProjectWise** > **Open**.
2. If you haven't already logged into ProjectWise, you are prompted to do so. Select a ProjectWise datasource, type your ProjectWise user name and password, then click **Log in**.
3. In the ProjectWise Select Document dialog box, perform these steps:
    a. From the Folder drop-down menu, select a folder that contains SewerGEMS V8*i* projects.
    b. In the Document list box, select a SewerGEMS V8*i* project.
    c. Keep the default entries for the rest of the fields in the dialog box
    d. Click **Open**.

To copy an open SewerGEMS V8*i* project from one ProjectWise datasource to another:

1. Select **File** > **ProjectWise** > **Open** to open a project stored in ProjectWise.
2. Select **File** > **ProjectWise** > **Change Datasource**.
3. In the ProjectWise Log in dialog box, select a different ProjectWise datasource, then click **Log in**.
4. Select **File** > **ProjectWise** > **Save As**.
5. In the ProjectWise Save Document dialog box, change information about the project as required, then click **OK**.

To make a local copy of a SewerGEMS V8*i* project stored in a ProjectWise datasource:

1. Select **File** > **ProjectWise** > **Open**.
2. If you haven't already logged into ProjectWise, you are prompted to do so. Select a ProjectWise datasource, type your ProjectWise user name and password, then click **Log in**.

3. Select **File > Save As**.
4. Save the SewerGEMS V8*i* project to a folder on your local computer.

To change the default ProjectWise datasource:

1. Start SewerGEMS V8*i*.
2. Select **File > ProjectWise > Change Datasource**.
3. In the ProjectWise Log in dialog box, type the name of ProjectWise datasource you want to log into, then click **Log in**.

To use background layer files with ProjectWise:

- Using File > ProjectWise > Save As—If there are background files, you are prompted with two options: you can copy the background layer files to the project folder for use by the project, or you can remove the background references and manually reassign them once the project is in ProjectWise to other existing ProjectWise documents.
- Using File > ProjectWise > Open—This works the same as the normal Project-Wise > Open command, except that background layer files are not locked in ProjectWise for the current user to edit. The files are intended to be shared with other users at the same time.
- To add a background layer file reference to a project that exists in Project Wise—The ProjectWise Select Document dialog box opens, and you can choose any existing ProjectWise document. You must have previously added these back-ground layer files as described in the first bullet above, or by using the Project-Wise Explorer.
- When you remove a background layer file reference from a project that exists in ProjectWise, the reference to the file is removed but the file itself is not deleted from ProjectWise.
- Using File > Save As—When you use File > Save As on a project that is already in ProjectWise and there are background layer files, you are prompted with two options: you can copy all the files to the local project folder for use by the project, or you can remove the background references and manually reassign them after you have saved the project locally.

Using ProjectWise with Bentley SewerGEMS V8*i* for AutoCAD

Bentley SewerGEMS V8*i* for AutoCAD maintains a one to one relationship between the AutoCAD drawing (.dwg) and the Bentley SewerGEMS V8*i* project file. When using ProjectWise with this data, we recommend that you create a Set in the Project-Wise Explorer. Included in this set should be the AutoCAD drawing (example.dwg), the Bentley SewerGEMS V8*i* database (example.swg.mdb), the Bentley SewerGEMS V8*i* project file (example.swg), and optionally for stand-alone, the stand-alone drawing setting file (example.swg.dwh).

If you use the Set and the ProjectWise Explorer for all of your check-in / check-out procedures, you will maintain the integrity of this relationship. We recommended that you do not use the default ProjectWise integration in AutoCAD, as this will only work with the .dwg file.

Using ProjectWise with Bentley SewerGEMS V8*i* for MicroStation

When using ProjectWise with a MicroStation SewerGEMS V8*i* project, we recommend that you create a Set in the ProjectWise Explorer. Included in this set should be the MicroStation drawing (example.dgn), the Bentley SewerGEMS V8*i* database (example.swg.mdb), the Bentley SewerGEMS V8*i* project file (example.swg), and optionally for stand-alone, the stand-alone drawing setting file (example.swg.dwh).

If you use the Set and the ProjectWise Explorer for all of your check-in / check-out procedures, you will maintain the integrity of this relationship. We recommended that you do not use the default ProjectWise integration in AutoCAD, as this will only work with the .dgn file.

Importing Data From Other Models

Bentley SewerGEMS V8*i* lets you import data from a variety of data sources, including Bentley SewerGEMS V8*i*, EPA SWMM, SewerCAD, and StormCAD.

- “Importing Data from a CivilStorm Database” on page 4-162
- “Importing Data from SewerCAD” on page 4-163
- “Importing a StormCAD Exchange Database” on page 4-164
- “Importing StormCAD V8i” on page 4-165
- “Importing Data from Bentley Wastewater” on page 4-166

Importing Data from a CivilStorm Database

You can import data from another SewerGEMS V8*i* project by importing the SewerGEMS V8*i* database (.MDB) file. You might want to do this if you need to rebuild your model, or if you want to open a SewerGEMS V8*i* project sent to you by another SewerGEMS V8*i* user.

When you import a SewerGEMS V8*i* database (.MDB) file, the model will notice that the model does not have any element symbology definitions (annotations and color-coding), project-level options, text customizations, or border and line customizations that may have been associated with the project. These are stored in the project’s .CSD and .DWH files, which are not imported.

Note: SewerGEMS can only import CivilStorm databases created in CivilStorm version 8.9.20.9 or later. If the database you want to import was created in an older version, you will need to open the file in a newer version of CivilStorm before the import into SewerGEMS will work.

To import data from a CivilStorm project:

1. Select **File** > **Import** > **CivilStorm Database**.
2. In the Select Database File to Import dialog box, select the CivilStorm .mdb file (database file). The data will display as a new project in CivilStorm.

Importing Data from SewerCAD

You can import data from a SewerCAD model into SewerGEMS V8*i*. When you installed Bentley SewerGEMS V8*i*, an additional menu command was added to the SewerCAD File > Export menu that allows you to export SewerCAD project data. You must use this command to export your SewerCAD data before you can import it into SewerGEMS V8*i*.

To import SewerCAD data into SewerGEMS V8*i*:

1. Open SewerCAD, then open the model you want to import into SewerGEMS V8*i*.
2. Select **File** > **Export** > **SewerCAD Exchange Database**.
3. Select the file name for the exported data. The file name extension for a SewerCAD Exchange Database is .SWR.MDB.
4. In SewerGEMS V8*i*, select **File** > **Import** > **SewerCAD Exchange Database**.
5. Select the .SWR.MDB file that you created when you exported your model data from SewerCAD.

Note: A pressure junction in Bentley SewerGEMS V8*i* has a specific ground elevation; in SewerCAD, there is no specific ground elevation for a presure junction. Bentley SewerGEMS V8*i* adds 5 feet to the SewerCAD invert elevation value to arrive at the appropriate Bentley SewerGEMS V8*i* ground elevation.

Related Topics

- "Importing Data from a SewerGEMS Database" on page 4-165
- "Importing a StormCAD Exchange Database" on page 4-164

Importing a StormCAD Exchange Database

You can import an exchange database from StormCAD v5.6 into SewerGEMS V8*i*. When you installed Bentley SewerGEMS V8*i*, an additional menu command was added to the StormCAD File > Export menu that allows you to export StormCAD project data. You must use this command to export your StormCAD data before you can import it into SewerGEMS V8*i*.

During the export from StormCAD, you are prompted to define the following ranges:

- Grade Inlet Hydrologic Rating Table—the range over which the hydrologic rating curve is developed for each grade inlet in the model. The export process will automatically generate these relationships for each inlet in the model.
- Intensity Duration Inlet Rating Definition—the range over which the IDF equation or table is developed for each inlet in the model. The export process will automatically convert the IDF equation or table into a data table that follows the specified range definition.

To import a StormCAD exchange database into SewerGEMS V8*i*:

1. Open StormCAD, then open the model you want to import into SewerGEMS V8*i*.
2. Select **File > Export > StormCAD Exchange Database**.
3. In the GEMS Exchange File Export dialog box, enter the following:
   a. Define the grade inlet hydrologic rating table (the range over which the hydrologic rating curve is developed for each grade inlet in the model) by entering a maximum discharge and a flow increment.
   b. Define the intensity duration inlet rating (the range over which the IDF equation or table is developed for each inlet in the model) by entering a maximum duration and a duration increment.
4. Select the file name for the exported data. The file name extension for a StormCAD Exchange Database is .STM.MDB.
5. In SewerGEMS V8*i*, select **File > Import > StormCAD Exchange Database**.
6. Select the .STM.MDB file that you created when you exported your model data from SewerCAD.

- "Importing Data from a SewerGEMS Database" on page 4-165

Importing StormCAD V8*i*

You can import a model created in StormCAD V8*i* into SewerGEMS V8*i*. Note that, unlike the StormCAD Exchange Database import, the StormCAD V8*i* import does not require you to create an exchange database file in StormCAD first - SewerGEMS V8*i* handles the StormCAD V8*i* edition model files natively.

After importing the StormCAD model database, the model will notice that the model does not have any element symbology definitions (annotations and color-coding), project-level options, text customizations, or border and line customizations that may have been associated with the project. These are stored in the project's .swg and .dwh files, which are not imported.

Because of differences between StormCAD and SewerGEMS V8*i*, some data will either not be imported or will be imported using near-equivalent substitutes, including the following:

- Only Mannings roughness values will be imported.
- AASHTO and Headloss Flow Curves from StormCAD will not be imported into SewerGEMS V8*i*.
- Catalog Inlets and Percent Capacity Inlets' are converted to Inflow-Capture curves.
- For StormCAD gutters, SewerGEMS V8*i* will set the shape as trapezoidal, set the appropriate side slopes, leave the bottom width = 0 and set an appropriate Manning's N. The depth value must be entered by the user. Additionally, gutters are set to inactive automatically upon import. You will need to make the gutters active and fill in missing data before the gutter can be included in the calculations.
- StormCAD has a Headloss Alternative, while SewerGEMS V8*i* does not. Headloss information from the StormCAD Headloss Alternative is imported into a child Physical alternative inheriting from the Base Physical alternative in SewerGEMS V8*i*. This child alternative contains the headloss data.

When a StormCAD model is imported, an Inlet Flow Settings dialog will appear. The data entered here is required to define the inlet flow-capture curves. The two pieces of data required in this dialog are as follows:

- **Flow Increment**: The flow increment value should be a value that will give the flow-capture curves satisfactory resolution.
- **Maximum Flow**: The value entered here should exceed the highest flow expected at any inlet.

Note: The flow-capture curves are only for on-grade inlets Sag inlets are set to "full capture".

To import a StormCAD V8*i* model

1. Click the File menu and select Import > StormCAD V8*i* Edition.
2. In the Select Database File to Import dialog, browse to the swg.mdb file for the StormCAD model you are importing. Highlight the file and click the Open button.
3. In the Inlet Flow Settings dialog, enter vlaues for the Flow Increment and Maximum Flow, then click OK.

Importing Data from Bentley Wastewater

You can import data from a Bentley Wastewater data source into SewerGEMS V8*i*. Before importing this data into SewerGEMS V8*i*, you must first export it from Bentley Wastewater into a set of output files. These output files can then be imported into a SewerGEMS V8*i* model.

To export Bentley Wastewater data to a set of output files:

1. In the **Bentley Wastewater** toolbar, click the **Export > Data** command.
2. In the dialog that appears, select all of the listed element types.
3. Click the **File** button and select a destination output file.
4. Use the MicroStation **Place Fence** tool and draw a fence surrounding the model.
5. Click on the **Start** button in the Export Data dialog and click inside the fence you created in step 4.

To import Bentley Wastewater data into SewerGEMS V8*i* from a set of output files:

1. In SewerGEMS V8*i*, click the **File** menu and select the **Import > Bentley Wastewater Import** command.
2. Follow the steps in the **Bentley Wastewater Import Wizard** that appears.

Bentley Wastewater Import Wizard

The Bentley Wastewater Import Wizard will guide you through the process of configuring the settings needed to import the data contained in the Bentley Wastewater output files into a SewerGEMS V8*i* model.

The wizard consists of the following steps:

- “Step 1: Bentley Wastewater Import” on page 4-167

- “Step 2: Bentley Wastewater Data Source” on page 4-167
- “Step 3: Data Source Table Names” on page 4-168
- “Step 4: Unit Options” on page 4-168
- “Step 5: Import Options” on page 4-169

Step 1: Bentley Wastewater Import

This step displays the filename and directory location of the target SewerGEMS V8*i* model for the import operation. No user input is required for this step.

Related Topics

- “Bentley Wastewater Import Wizard” on page 4-166
- “Step 1: Bentley Wastewater Import” on page 4-167
- “Step 2: Bentley Wastewater Data Source” on page 4-167
- “Step 3: Data Source Table Names” on page 4-168
- “Step 4: Unit Options” on page 4-168
- “Step 5: Import Options” on page 4-169

Step 2: Bentley Wastewater Data Source

This step allows you to specify the data source and geometry file that contains the Bentley Wastewater data to be imported.

The step consists of the following controls:

Select A Data Source Type—This control consists of two pulldown menus. The value selected in the first pulldown menu will determine the choices available in the second menu. For most of the Data Source Types, only one option will be available in the second menu. When the OLEDB data source type is chosen, the second menu will contain a number of OLEDB database types.

Select Data Source—This control allows you to choose the data source of the type selected in the Select A Data Source Type menus. Click the Browse button to bring up an Open dialog that will allow you to specify the data source.

Select Geometry Data File—This control allows you to choose the .dat file that contains the Bentley Wastewater geometry data. Click the Browse button to bring up an Open dialog that will allow you to specify the data file.

Note: The Select Geometry Data File field is optional. If the user wishes to import pipe geometry data which contains the pipe vertices (bends) into the model, then this field should point to the data file that can be created during the export pipe geometry data process. However, if the user chooses not to import geometry data, this field can be left blank and the pipes will be imported without any bends associated with them.

Update only the elements specified in the geometry data file—When this box is checked, only data for those elements contained within the specified geometry data file will be imported. This option allows you to import just a subset of the original Bentley Wastewater model.

Step 3: Data Source Table Names

This step allows you to specify the tables within the Bentley Wastewater data source that correspond to the various SewerGEMS V8*i* element types. Each of the menus in this step allow you to choose a database table that contains the data for each of the associated elements.

- "Bentley Wastewater Import Wizard" on page 4-166
- "Step 1: Bentley Wastewater Import" on page 4-167
- "Step 2: Bentley Wastewater Data Source" on page 4-167
- "Step 3: Data Source Table Names" on page 4-168
- "Step 4: Unit Options" on page 4-168
- "Step 5: Import Options" on page 4-169

Step 4: Unit Options

This step allows you to define the units used for various attributes of the model and the network elements. The units specified should match those used in the Bentley Wastewater model being imported.

- "Bentley Wastewater Import Wizard" on page 4-166
- "Step 1: Bentley Wastewater Import" on page 4-167
- "Step 2: Bentley Wastewater Data Source" on page 4-167
- "Step 3: Data Source Table Names" on page 4-168
- "Step 4: Unit Options" on page 4-168
- "Step 5: Import Options" on page 4-169

Step 5: Import Options

This step allows you to specify other options that will be applied to the SewerGEMS V8*i* model. The menu allows you to specify the friction method that will be used. Bentley Wastewater models can contain point nodes. If the checkbox in this step is checked, these point nodes will be imported as SewerGEMS V8*i* manhole elements.

- “Bentley Wastewater Import Wizard” on page 4-166
- “Step 1: Bentley Wastewater Import” on page 4-167
- “Step 2: Bentley Wastewater Data Source” on page 4-167
- “Step 3: Data Source Table Names” on page 4-168
- “Step 4: Unit Options” on page 4-168
- “Step 5: Import Options” on page 4-169

- “Importing Data from SewerCAD” on page 4-163

Exporting Data

You can export your SewerGEMS V8*i* data as a SWMM .INP file, or export the graphical representation of your model as a .DXF file.

Exporting a .DXF File

You can export your SewerGEMS V8*i* model as a .DXF file if you plan to edit the file in AutoCAD or another program. When you export a .DXF file, you export only the graphical (vector) representation of the model. The DXF file is an ASCII file.

To export the current project to a .DXF file:

1. Select **File** > **Export** > **DXF**.
2. Type the name of the DXF file, then click **Save**.

   You may now open the DXF file in another program.

Exporting to SWMM 5

You can export your SewerGEMS V8*i* model data to a SWMM 5 .INP file.

To export the current project to a SWMM 5 file:

1. Select **File > Export > SWMM v5**.
2. In the Select SWMM v5 File to Export dialog that appears, browse to the directory that you want to save the file to.
3. Type the name of the .INP file, then click **Save**.

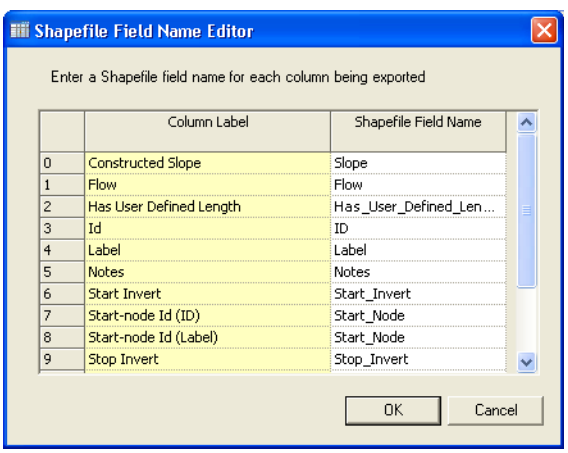
Note: Only catchments using the EPA SWMM Runoff Method will be exported using this command.

Exporting to Shapefile

It is possible to export model elements and data to create a shapefile. Unlike the other export features in SewerGEMS V8*i*, the export to shapefile operation occurs in a FlexTable as opposed to the File > Export menu. Shapefiles must be created one element type at a time. That means there will be a separate shapefile to junctions, pipes, tanks, etc.

To create a shapefile, open the FlexTable for the type of element. Use selection sets or filtering to reduce the size of the FlexTable to what is desired in the shapefile. Use the table edit feature to eliminate any columns that are not desired.

When the FlexTable is in the correct form, pick the first button at the top left of the table which is the Export button. A drop down list will appear, pick Export to Shapefile. The user is asked for the name of shapefile and path. When the user names the file and hits Save, the dialog below appears.



It is important to ensure that any shapefile field names are less than or equal to 10 characters. The default name for shapefile field is the name of the column in the FlexTable. (If the user changes the name to something different from the FlexTable column name, the editor remembers it when other shapefiles are created from this table.) Once the names are acceptable, hit OK to create the shapefile. A shapefile consisting of .dbf, .shx and .shp files are created.

Chapter

Using Modelbuilder 5

ModelBuilder lets you use your existing GIS asset to construct a new Bentley SewerGEMS V8*i* model or update an existing Bentley SewerGEMS V8*i* model. ModelBuilder supports a wide variety of data formats, from simple databases (such as Access and DBase), spreadsheets (such as Excel or Lotus), GIS data (such as shapefiles, coverages, ESRI ArcGIS Geodatabases, and ArcGIS Geometric Networks), to high end data stores (such as Oracle, and SQL Server), and more.

Using ModelBuilder, you map the tables and fields contained within your data source to element types and attributes in your Bentley SewerGEMS V8*i* model. The result is that a Bentley SewerGEMS V8*i* model is created, either in stand-alone mode or in an existing ArcMap project.

> **Note: ModelBuilder lets you bring a wide range of data into your model. However, some data is better suited to the use of the more specialized Bentley SewerGEMS V8*i* modules. For instance, LoadBuilder offers many powerful options for incorporating loading data into your model.**

ModelBuilder is the first tool you will use when constructing a model from GIS data. The steps that you take at the outset will impact how the rest of the process goes. Take the time now to ensure that this process goes as smoothly and efficiently as possible. The following topics are included:

Preparing to Use ModelBuilder

- **Determine the purpose of your model**—Once you establish the purpose of your model, you can start to make decisions about how detailed the model should be.
- **Get familiar with your data**—If you obtained your GIS data from an outside source, you should take the time to get acquainted with it. Review spatial and attribute data directly in your GIS environment. Do the nodes have coordinate information, and do the pipes have start and stop nodes specified? If not, the best method of specifying network connectivity must be determined.

  Contact those involved in the development of the GIS to learn more about the GIS tables and associated attributes. Find out the purpose of any fields that may be of interest, ensure that data is of an acceptable accuracy, and determine units associated with fields containing numeric data.

  Ideally, there will be one GIS source data table for each Bentley SewerGEMS V8*i* element type. This isn't always the case, and there are two other possible scenarios:

  Many GIS tables for one element type—In this case, there may be several tables in the GIS/database corresponding to a single GEMS modeling element . In this case each data source table must be individually mapped to the Bentley SewerGEMS V8*i* element, or the tables must be combined into a single table in the GIS/database before running ModelBuilder.

  One GIS table containing many element types—In this case, there may be entries that correspond to several Bentley SewerGEMS V8*i* modeling elements in one GIS/database table. You should separate these into individual tables before running ModelBuilder. The one case where a single table can work is when the features in the table are ArcGIS subtypes. ModelBuilder handles these subtypes by treating them as separate tables when setting up mappings. See "Subtypes" on page 5-186 for more information.

  Note: If you are working with an ArcGIS data source, see "ESRI ArcGIS Geodatabase Support" on page 5-184 for additional information.

- **Preparing your data**—When using ModelBuilder to get data from your GIS into your model, you will be associating rows in your GIS to elements in Bentley SewerGEMS V8*i*. Your data source needs to contain a Key/Label field that can be used to uniquely identify every element in your model. The data source tables should have identifying column labels, or ModelBuilder will interpret the first row of data in the table as the column labels. Be sure data is in a format suited for use in ModelBuilder. Use powerful GIS and Database tools to perform Database Joins, Spatial Joins, and Update Joins to get data into the appropriate table, and in the desired format.

Note: When working with ID fields, the expected model input is the Bentley SewerGEMS V8*i* ID. After creating these items in your Bentley SewerGEMS V8*i* model, you can obtain the assigned ID values directly from your Bentley SewerGEMS V8*i* modeling file. Before synchronizing your model, get these Bentley SewerGEMS V8*i* IDs into your data source table (e.g., by performing a database join).

One area of difficulty in building a model from GIS data is the fact that unless the GIS was created solely to support modeling, it most likely contains much more detailed information than is needed for modeling. This is especially true with regard to the number of piping elements. It is not uncommon for the GIS to include every service line and hydrant lateral. Such information is not needed for most modeling applications and should be removed to improve model run time, reduce file size, and save costs.

ModelBuilder Connections Manager

ModelBuilder is available within both the Stand-Alone and ArcMap client interfaces. To access the ModelBuilder Connections Manager:

- In Stand-Alone: Click the Tools menu and select the ModelBuilder command.
- In ArcMap: Click the Bentley SewerGEMS V8*i* menu, click the Tools menu, then select the ModelBuilder command.

The ModelBuilder Connections manager allows you to create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process.

At the center of this window is the **Connections List** which displays the list of connections that you have defined.

There is a toolbar located along the top of the Connections list.

The set of buttons on the left of the toolbar allow you to manage your connections:

	**New**	Create a new connection using the ModelBuilder Wizard.
	**Edit**	Edit the selected connection using the ModelBuilder Wizard.
	**Rename**	Rename the selected connection.
	**Duplicate**	Create a copy of the selected connection.
	**Delete**	Permanently Remove the selected connection.

The button on the right of the toolbar allows you to either build or synchronize a model. Click the menu arrow associated with this button to access the following options:

- **Build New Model—**Starts the ModelBuilder build process using the selected connection. You will be prompted to interactively specify a new filename.
- **Synchronize Existing Model**—Starts the ModelBuilder synchronize process using the selected connection. You will be prompted to interactively specify an existing Bentley SewerGEMS V8*i* model filename.

Note: If you set up a ModelBuilder mapping to an Access mdb, it requires a primary key for that table.

After specifying your target, ModelBuilder will perform the selected operation. During the process, a progress-bar will be displayed indicating the step that Model-Builder is currently working on.

When ModelBuilder completes, you will be presented with a summary window that outlines important information about the build process. We recommend that you save this summary so that you can refer to it later.

ModelBuilder Wizard

The ModelBuilder Wizard assists in the creation of ModelBuilder connections. The Wizard will guide you through the process of selecting your data source and mapping that data to the desired input of your model.

Tip: The ModelBuilder Wizard can be resized, making it easier to preview tables in your data source. In addition, Step 1 and Step 3 of the wizard offer a vertical split bar, letting you adjust the size of the list located on the left side of these pages.

There are 5 steps involved:

- "Step 1—Specify Project" on page 5-175
- "Step 2—Specify Data Source" on page 5-175
- "Step 3—Specify Spatial Options" on page 5-176
- "Step 4—Specify Field Mappings for each Table/Feature Class" on page 5-177
- "Step 5—Build Operation Confirmation" on page 5-179

Step 1—Specify Project

In this step, the path to the Bentley SewerGEMS V8*i* project is defined.

The following controls are available:

- Bentley SewerGEMS V8*i* **Model Path—**This field displays the path and file name of the Bentley SewerGEMS V8*i* model that the current build operation will be applied to.
- **Browse—**This button opens a browse dialog, allowing you to specify the Bentley SewerGEMS V8*i* model that the build operation will be applied to.

Step 2—Specify Data Source

In this step, the data source type and location are specified. After selecting your data source, the desired database tables can be chosen and previewed.

The following fields are available:

- **Data Source type** (drop-down list)—This field allows you to specify the type of data you would like to work with.

Note: If your specific data source type is not listed in the Data Source type field, try using the OLE DB data source type. OLE DB can be used to access many database systems (including ORACLE, and SQL Server, to name a few).

- **Data Source** (text field)—This read-only field displays the path to your data source.
- **Browse** (button)—This button opens a browse dialog box that allows you to interactively select your data source.

Note: Some Data Source types expect you to choose more than one item in the Browse dialog box. For more information, see “Multi-select Data Source Types” on page 5-180**.**

- **Table/Feature Class** (list)—This pane lists the tables/feature classes that are contained within the data source. Use the check boxes (along the left side of the list) to specify the tables you would like to include.

Tip: The list can be resized using the split bar (located on the right side of the list).

Right-click to Select All or Clear the current selection in the list.

ModelBuilder has built in support for ArcGIS Subtypes. For more information, see “ESRI ArcGIS Geodatabase Support” on page 5-184**.**

- **Preview Pane**—A tabular preview of the highlighted table is displayed in this pane when the **Show Preview** check box is enabled.

Step 3—Specify Spatial Options

In this step you will specify the spatial options to be used during the ModelBuilder process. The spatial options will determine the placement and connectivity of the model elements. The fields available in this step will vary depending on the data source type.

Specify the Coordinate Unit of your data source (drop-down list)—This field allows you to specify the coordinate unit of the spatial data in your data source.

The remaining fields are only available when working with the spatial data sources such as shapefiles and ArcGIS Geodatabases.

- **Establish connectivity using spatial data** (check box)—When this box is checked, ModelBuilder will connect pipes to nodes that fall within a specified tolerance of a pipe endpoint.

- **Tolerance** (numeric field)—This field dictates how close a node must be to a pipe endpoint in order for connectivity to be established. The **Tolerance** field is available only when the **Establish connectivity using spatial data** box is checked.

  Note: Pipes will be connected to the closest node within the specified tolerance.

  The unit associated with the tolerance is dictated by the Specify the Coordinate Unit of your data source field.

  For more information, see "Specifying Network Connectivity in ModelBuilder" on page 5-186**.**

- **Create nodes if none found at pipe endpoint** (check box)—When this box is checked, ModelBuilder will create a pressure junction at any pipe endpoint that: a) doesn't have a connected node, and b) is not within the specified tolerance of an existing node. This field is active only when the Establish connectivity using spatial data box is checked.

Step 4—Specify Field Mappings for each Table/Feature Class

In this step, data source tables are mapped to the desired modeling element types, and data source fields are mapped to the desired model input attributes. You will assign mappings for each Table/Feature Class that appears in the list; **Step 1** of the wizard can be used to exclude tables, if you wish.

- **Tables** (list)—This pane, located along the left side of the dialog box, lists the data source Tables/Feature Classes to be used in the ModelBuilder process. Select an item in the list to specify the settings for that item.

  Tip: The list can be resized using the split bar.

  There are two toolbar buttons located directly above Tables list (these buttons can be a great time saver when setting up multiple mappings with similar settings).

  – **Copy Mappings** (button)—This button copies the mappings (associated with the currently selected table) to the clipboard.

  – **Paste Mappings** (button)—This button applies the copied mappings to the currently selected table.

- **Settings Tab**—The Settings tab allows you to specify mappings for the selected item in the Tables list.

  The top section of the Settings tab allows you to specify the common data mappings:

– **Table Type** (drop-down list)—This field allows you to specify the target modeling element type that the source table/feature class represents. For example, a source table that contains pipe data should be associated with the Pressure Pipe element type.

Note: Each table in your data source can be assigned to a single Element Type. For this reason, individual tables in your data source should contain only data that will be assigned to elements of the same type. For instance, the source data table that will be designated as the Pressure Junction element type should contain only data that will be applied to Pressure Junctions, not data relating to pressure valves, tanks, etc. The exception to this general rule is when the source table contains subtypes, each of which can be assigned to different elements. For more information see “Subtypes” on page 5-186**.**

Tip: Shapefiles can be converted into Geodatabase format if you would like to make use of Subtypes. This can be useful if a single data source table needs to be mapped to multiple Bentley SewerGEMS V8*i* element types.

– **Key/Label Field** (drop-down list)—This required field allows you to associate a row in this table to a particular element in the model. The model references each element using a unique alphanumeric label. Your data source must have a field that can be used to uniquely identify all elements in the model.

Note: When working with ArcGIS data sources, OBJECTID is not a good choice for Key field (because OBJECTID is only unique for that particular Feature Class). If you do not have a field that can be used to uniquely identify each element, you may use the <label> field (which is automatically generated by ModelBuilder for this purpose).

These optional fields are available for Pipe element types:

– **Start/Stop**—Select the fields in your pipe table that contain the Label of the start and stop nodes. For more information, see “Specifying Network Connectivity in ModelBuilder” on page 5-186.

Note: When working with an ArcGIS Geometric Network data source, these fields will be set to <auto> (indicating that ModelBuilder will automatically determine connectivity from the geometric network).

These fields are available for Node element types:

– **X/Y Field**—These fields are used to specify the node X and Y coordinate data.

Note: The Coordinate Unit setting in Step 2 of the wizard allows you to specify the units associated with these fields.

When working with ArcGIS Geodatabase or shapefile data sources, these fields will be set to <auto> (indicating that ModelBuilder will automatically determine node geometry from the data source).

These optional fields are available for Pump element types:

– **Suction Element (drop-down list)**—For tables that define pump data, select a pipe label or other unique identifier to set the suction element of the Pump.

– **Downstream Edge** (drop-down list)—For tables that define pump or valve data, select a pipe label or other unique identifier to set the direction of the Pump.

The bottom section of the Settings tab allows you to specify additional data mappings.

– **Field**—Field refers to a field in the selected data source. The Field list displays the associations between fields in the database to attributes in the model.

– **Attribute** (drop-down list)—Attribute refers to a Bentley SewerGEMS V8*i* attribute. Use the Attribute drop-down list to map the highlighted field to the desired attribute.

– **Unit** (drop-down list)—This field allows you to specify the units of the values in the database (no conversion on your part is required).

To map a field in your GIS to a particular Bentley SewerGEMS V8*i* attribute:

1. In the **Field** list, select the item you would like to update.
2. In the **Attribute** drop-down list, select the desired Bentley SewerGEMS V8*i* attribute.
3. In the **Unit** drop-down list, specify the unit of this field in your data source.

To remove the mapping for a particular field:

1. Select the field you would like to update.
2. In the **Attribute** drop-down list, select **<none>**.

– **Preview Tab**—The Preview tab displays a tabular preview of the currently highlighted source data table when the Show Preview check box is checked.

Step 5—Build Operation Confirmation

In this step, you are prompted to build a new model or update an existing model.

To build a new model, click the **Yes** radio button under *Would you like to build the model now?*.

If you choose No, you will be returned to the ModelBuilder Manager dialog. The connection you defined will appear in the list pane. To build the model from the ModelBuilder Manager, highlight the connection and click the Build Model button.

Reviewing Your Results

At the end of the ModelBuilder process, you will be presented with statistics, and a list of any warning/error messages reported during the process. You should closely review this information, and be sure to save this data to disk where you can refer to it later.

Note: Refer to the section titled “ModelBuilder Warnings and Error Messages” on page 5-181 **to determine the nature of any messages that were reported.**

Multi-select Data Source Types

When certain Data Source types are chosen in Step 1 of the ModelBuilder Wizard (see “Step 2—Specify Data Source” on page 5-175), multiple items can be selected for inclusion in your ModelBuilder connection.

After clicking the **Browse** button to interactively specify your data source, use standard Windows selection techniques to select all items you would like to include in the connection (e.g., Ctrl+click each item you would like to include).

The following are multi-select Data Source types:

- ArcGIS Geodatabase Features
- Shapefiles
- DBase, FoxPro, HTML Export, and Paradox.

Exporting X/Y Coordinates

SewerGEMS V8*i* X/Y coordinates can be exported to an external data tables, such as spreadsheets, using ModelBuilder.

1. Add fields to the external data table to accept the coordinate data.
2. In “Step 5—Build Operation Confirmation” on page 5-179 of the ModelBuilder Wizard, click the **No** radio button under **Build Model Now?**, then click **Finish**.
3. In the **ModelBuilder Manager**, highlight the connection and click the **Sync Out** button.

ModelBuilder Warnings and Error Messages

Errors and warnings that are encountered during the ModelBuilder process will be reported in the ModelBuilder Summary.

For more information, see:

- "Warnings" on page 5-182
- "Error Messages" on page 5-183

Warnings

Warning messages include:

1. *Some rows were ignored due to missing key-field values.*

   ModelBuilder encountered missing data (e.g., null or blank) in the specified Key/Label field for rows in your data source table. Without a key, ModelBuilder is unable to associate this source row with a target element, and must skip these items. This can commonly occur when using a spreadsheet data source. To determine where and how often this error occurred, check the Statistics page for the message **<x> row(s) ignored due to missing key-field values**.

2. *Unable to create pipe <element>; start and/or stop node could not be found.*

   Pipes can only be created if its start and stop nodes can be established. If you are using Explicit connectivity, a node element with the referenced start or stop label could not be found. If you are using implicit connectivity, a node element could not be located within the specified tolerance. For more information, see "Specifying Network Connectivity in ModelBuilder" on page 5-186.

3. *Unable to update pipe <element> topology; (start or stop) node could not be found.*

   This error occurs when synchronizing an existing model, and indicates that the pipe connectivity could not be updated. For more information, see warning message #2 (above).

4. *The downstream edge for <element> could not be found.*

   ModelBuilder was unable to set a Pump direction because a pipe with the referenced label could not be found.

5. *Directed Node <element> direction is ambiguous.*

   ModelBuilder was unable to set the direction of the referenced pump or valve because direction could not be implied based on the adjacent pipes (e.g. there should be one incoming and one outgoing pipe).

Error Messages

Note: If you encounter these errors or warnings, we recommend that you correct the problems in your original data source and re-run ModelBuilder (when applicable).

Error messages include:

1. *Unable to assign <attribute> for element <element>.*

   Be sure that the data in your source table is compatible with the expected Bentley SewerGEMS V8*i* format. For more information, see "Using Modelbuilder" on page 5-171.

2. *Unable to create <element type> <element>.*

   This message indicates that an unexpected error occurred when attempting to create a node element.

3. *Unable to create pipe <element> possibly due to start or stop connectivity constraints.*

   This message indicates that this pipe could not be created, because the pump or valve already has an incoming and outgoing pipe. Adding a third pipe to a pump or valve is not allowed.

4. *Unable to update pipe <element> topology; possibly due to start element connectivity constraints.*

   This error occurs when synchronizing. For more information, see error message #3 (above).

5. *Operation terminated by user.*

   You pressed the Cancel button during the ModelBuilder process.

6. *Unable to create < element>; pipe start and stop must be different.*

   This message indicates that the start and stop specified for this pipe refer to the same node element.

7. *Unable to update <element> topology; pipe start and stop must be different.*

   This message indicates that the start and stop specified for this pipe refer to the same node element.

8. *Unable to update the downstream edge for <element>.*

   An unexpected error occurred attempting to set the downstream edge for this pump or valve.

9. *Nothing to do. Some previously referenced tables may be missing from your data source.*

This data source has changed since this connection was created. Verify that tables/feature-classes in your data source have not been renamed or deleted.

10. *One or more input features fall outside of the XYDomain.*

    This error occurs when model elements have been imported into a new geodatabase that has a different spatial reference from the elements being created. Elements cannot be created in ArcMAP if they are outside the spatial bounds of the geodatabase.

    The solution is to assign the correct X/Y Domain to the new geodatabase when it is being created:

    a. In the Attach Geodatabase dialog that appears after you initialize the Create New Project command, click the Change button.

    b. In the Spatial Reference Properties dialog that appears, click the Import button.

    c. Browse to the datasource you will be using in ModelBuilder and click Add.

    d. Back in the Spatial Reference Properties dialog, click the x/Y Domain tab. The settings should match those of the datasource.

    e. Use ModelBuilder to create the model from the datasource.

ESRI ArcGIS Geodatabase Support

ModelBuilder was built using ArcObjects, and supports the following ESRI ArcGIS Geodatabase functionality. See your ArcGIS documentation for more information about ArcObjects. For more information, see:

- "Geodatabase Features" on page 5-185
- "Geometric Networks" on page 5-185
- "ArcGIS Geodatabase Features versus ArcGIS Geometric Network" on page 5-185
- "Subtypes" on page 5-186
- "SDE (Spatial Database Engine)" on page 5-186

Geodatabase Features

ModelBuilder provides direct support for working with Geodatabase features. A feature class is much like a shapefile, but with added functionality (such as subtypes).

The geodatabase stores objects. These objects may represent nonspatial real-world entities, such as manufacturers, or they may represent spatial objects, such as pipes in a network. Objects in the geodatabase are stored in feature classes (spatial) and tables (nonspatial).

The objects stored in a feature class or table can be organized into subtypes and may have a set of validation rules associated with them. The ArcInfo™ system uses these validation rules to help you maintain a geodatabase that contains valid objects.

Tables and feature classes store objects of the same type—that is, objects that have the same behavior and attributes. For example, a feature class called WaterMains may store pressurized water mains. All water mains have the same behavior and have the attributes ReferenceID, Depth, Material, GroundSurfaceType, Size, and PressureRating.

Geometric Networks

ModelBuilder has support for Geometric Networks, and a new network element type known as Complex Edge. When you specify a Geometric Network data source, ModelBuilder automatically determines the feature classes that make up the network. In addition, ModelBuilder can automatically establish model connectivity based on information in the Geometric Network.

ArcGIS Geodatabase Features versus ArcGIS Geometric Network

Note: See your ArcGIS documentation for more information about Geometric Networks and Complex Edges.

When working with a Geometric Network, you have two options for constructing your model—if your model contains Complex Edges, then there is a distinct difference. A Complex Edge can represent a single feature in the Geodatabase, but multiple elements in the Geometric Network.

For example, when defining your Geometric Network, you can connect a lateral to a main without splitting the main line. In this case, the main line will be represented as a single feature in the Geodatabase but as multiple edges in the Geometric Network.

Depending on the data source type that you choose, ModelBuilder can see either representation. If you want to include every element in your system, choose ArcGIS Geometric Network as your data source type. If you want to leave out laterals and you want your main lines to be represented by single pipes in the model, choose **ArcGIS Geodatabase Features** as your data source type.

Subtypes

Tip: Shapefiles can be converted into Geodatabase Feature Classes if you would like to make use of Subtypes. See your ArcGIS documentation for more information.

If multiple types of Bentley SewerGEMS V8*i* elements have their data stored in a single geodatabase table, then each element must be a separate ArcGIS subtype. For example, in a valve table PRVs may be subtype 1, PSVs may be subtype 2, FCVs may be subtype 3, and so on. With subtypes, it is not necessary to follow the rule that each GIS/database feature type must be associated with a single type of GEMS model element. Note that the subtype field must be of the integer type (e.g., 1, 2) and not an alphanumeric field (e.g., PRV). For more information about subtypes, see ArcGIS Help.

ModelBuilder has built in support for subtypes. After selecting your data source, feature classes will automatically be categorized by subtype. This gives you the ability to assign mappings at the subtype level. For example, ModelBuilder allows you to exclude a particular subtype within a feature class, or associate each subtype with a different element type.

SDE (Spatial Database Engine)

ModelBuilder lets you specify an SDE Geodatabase as your data source. See your ESRI documentation for more information about SDE.

Specifying Network Connectivity in ModelBuilder

When importing spatial data (ArcGIS Geodatabases or shapefile data contain spatial geometry data that ModelBuilder can use to establish network connectivity by connecting pipe ends to nodes, creating nodes at pipe endpoints if none are found.), ModelBuilder provides two ways to specify network connectivity:

- **Explicit connectivity**—based on pipe **Start** node and **Stop** node (see "Step 4—Specify Field Mappings for each Table/Feature Class" on page 5-177).
- **Implicit connectivity**—based on spatial data. When using implicit connectivity, ModelBuilder allows you to specify a **Tolerance**, and provides a second option allowing you to **Create nodes if none found** (see "Step 3—Specify Spatial Options" on page 5-176).

The method that you use will vary depending on the quality of your data. The possible situations include (in order from best case to worst case):

- You have pipe start and stop information—Explicit connectivity is definitely the preferred option.
- You have some start and stop information—Use a combination of explicit and implicit connectivity (use the **Spatial Data** option, and specify pipe Start/Stop fields). If the start or stop data is missing (blank) for a particular pipe, Model-Builder will then attempt to use spatial data to establish connectivity.
- You do not have start and stop information—Implicit connectivity is your only option. If your spatial data is good, then you should reduce your **Tolerance** accordingly.
- You do not have start and stop information, and you do not have any node data (e.g., you have GIS data that defines your pipes, but you do not have data for nodes)—Use implicit connectivity and specify the **Create nodes if none found** option; otherwise, the pipes cannot be created.

> **Note:** **If pipes do not have explicit Start/Stop nodes and "Establish connectivity using spatial data" is not checked, the pipes will not be connected to the nodes and a valid model will not be produced.**

Other considerations include what happens when the coordinates of the pipe ends do not match up with the node coordinates. This problem can be one of a few different varieties:

1. **Both nodes and pipe ends have coordinates, and pipes have explicit Start/Stop nodes**—In this case, the node coordinates are used, and the pipe ends are moved to connect with the nodes.
2. **Nodes have coordinates but pipes do not have explicit Start/Stop nodes**—The nodes will be created, and the specified tolerance will be used to connect pipe ends within this tolerance to the appropriate nodes. If a pipe end does not fall within any node's specified tolerance, a new node can be created using the **Create nodes if none found** option.
3. **Pipe ends have coordinates but there are no junctions**—New nodes must be created using the **Create nodes if none found** option. Pipe ends are then connected using the tolerance that is specified.

Another situation of interest occurs when two pipes cross but aren't connected. If, at the point where the pipes cross, there are no pipe ends or nodes within the specified tolerance, then the pipes will not be connected in the model. If you intend for the pipes to connect, then pipe ends or junctions must exist within the specified tolerance.

Sample Spreadsheet Data Source

Note: Database formats (such as MS Access) are preferable to simple spreadsheet data sources. The sample below is intended only to illustrate the importance of using expected data formats.

Here are two examples of possible data source tables. The first represents data that is in the correct format for an easy transition into ModelBuilder, with no modification. The second table will require adjustments before all of the data can be used by ModelBuilder.

Table 5-1: Correct Data Format for ModelBuilder

Label	Roughness_C	Diam_in	Length_ft	Material_ID	Subtype
P-1	120	6	120	3	2
P-2	110	8	75	2	1
P-3	130	6	356	2	3
P-4	100	10	729	1	1

Table 5-2: Data Format Needs Editing for ModelBuilder

P-1	120	.5	120	PVC	Phase2
P-2	110	.66	75	DuctIron	Lateral
P-3	130	.5	356	PVC	Phase1
P-4	100	.83	729	DuctIron	Main
P-5	100	1	1029	DuctIron	Main

In "Table 5-2: Data Format Needs Editing for ModelBuilder"on page 5-188, no column labels have been specified. ModelBuilder will interpret the first row of data in the table as the column labels, which can make the attribute mapping step of the ModelBuilder Wizard more difficult unless you are very familiar with your data source setup.

"Table 5-1: Correct Data Format for ModelBuilder"on page 5-188 is also superior to "Table 5-2: Data Format Needs Editing for ModelBuilder"on page 5-188 in that it clearly identifies the units that are used for unitized attribute values, such as length and diameter. Again, unless you are very familiar with your data source, unspecified units can lead to errors and confusion.

Finally, "Table 5-2: Data Format Needs Editing for ModelBuilder"on page 5-188 is storing the Material and Subtype attributes as alphanumeric values, while Model-Builder uses integer ID values to access this input. This data is unusable by Model-Builder in alphanumeric format, and must be translated to an integer ID system in order to read this data.

Handling Collection and Curve Data in Modelbuilder

ModeBuilder has the ability to import Collection and Curve data, such as pump curves, hydrographs, and IDF curves, among others. In the model, these data types are always associated to either Domain elements (pumps, pipes, …) or Components (pump definitions, hydrographs, …). You can just import the collection data, and ModelBuilder will create a default domain/component element if it doesn't yet exist, and if the creation option to automatically create referenced elements is enabled. But a better technique would be to import the domain/component element from an external table, and then import the collection data into those created domain/component elements. This gives you the ability to import the most amount of detail.

The external tabular data for a collection curve must have a label field, where the label field contains string values that match the label of the associated domain/component element. For example, if two pattern curves are defined in an external table, the table would look something like the following:

Table 5-3: Sample Tabular Data

Label	Time from Start	Multiplier	Order
Normal	1.2	.8	1
Normal	1.7	.7	2
Normal	12	.2	3
Normal	17	.5	4
High	5	1.0	1
High	19	.85	3
High	12	.65	2

This would assign 4 entries to the 'Normal' pattern, and 3 entries to the 'High' pattern. The Order field is optional, and is discussed below. The same approach applies to nodes, for such things as a variable area tank curve, or junction demands. In these cases, the label field would contain the name of the node that collection entries are being added into.

ModelBuilder also includes an advanced feature to allow precise ordering of the collection records. For some collections, the order of the records does not matter and this feature isn't needed. For other cases, order of the records is meaningful. For these types of collections, there is a Sort By Field in the ModelBuilder mapping form. By default, it uses the record order as the records exist in the external data source. If the records are not ordered correctly in the data source, then the external table must have an additional field that contains numeric values. These values will represent the order that the records should be imported. So for the above example data, normally the records would be imported in the order entered (5, 19, then 12 for the High pattern). However the user can set the Sort By Field in the ModelBuilder form to use the 'Order' external field, and this will import the records based on either Ascending 'Order' values (5, 12, 19) or Descending 'Order' values (19, 12, 5).

Chapter

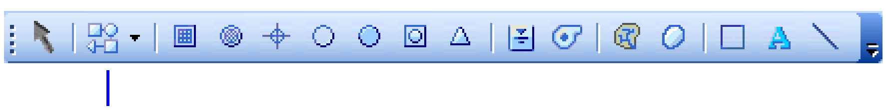
Creating Your Model 6

Elements and Element Attributes

You use the Layout toolbar to add elements to your model and edit the attributes of elements using the Property Editor, one of the dockable managers in Bentley Sewer-GEMS V8*i* stand-alone editor.

- “Pressure Junctions” on page 6-212
- “Junction Chambers” on page 6-211

Link Elements



Layout tool

Link elements connect the other elements to form the sewer network. The link elements are the conveyance elements that carry flow through the network to its eventual discharge point at an outlet. You can add any of the following link elements to your model, depending on the link element’s location within the network:

- Pressure pipes
- Conduits
- Channels
- Gutters

When you click the Layout tool on the Layout toolbar, you select the type of link element to add (pressure pipe, conduit, channel, or gutter), then select an element. You can place multiple elements with different kinds of connections using the Layout tool.

- "Pressure Pipe Attributes" on page 15-821

Entering Additional Data to Link Elements

There are several dialog boxes that are available from the Property Editors that let you enter additional data for link elements.

- "Defining a Control Structure in a Conduit" on page 6-192
- "Adding a Minor Loss Collection to a Pressure Pipe" on page 6-196
- "Defining the Geometry of a Link Element" on page 6-198
- "Defining the Cross-Sectional Shape of a Link Element" on page 6-199
- "Defining Manning's n vs. Depth Curves" on page 6-201
- "Defining Manning's n vs. Flow Curves" on page 6-202

Defining a Control Structure in a Conduit

In SewerGEMS V8*i*, you can attach a control structure, such as a weir or orifice, at either the upstream end or the downstream end of a conduit, or at both ends of the conduit. A control can also have a flap gate which allows flow to travel in only one direction. You define control structures for conduits in the Conduit Control Structures dialog box, which is accessible from the Property Editor for a conduit.

Note: For more information about control structures in SewerGEMS V8*i*, see "Flow Control Structures" on page 14-748.

To define a control structure in a conduit:

1. Click a conduit in your model to display the Property Editor, or right-click a conduit and select **Properties** from the shortcut menu.
2. If the conduit has a start control structure, do the following:
   a. In the Physical: Control Structure section of the Property Editor for the conduit, select the type of start control structure by selecting **Side** or **Inline** from the Start Control Structure Type submenu.
   a. If the conduit contains a start control structure, set the Has Start Control Structure? field to **True**.
3. If the control structure contains a flap gate, set the Flap Gate field to **True**.
4. If the conduit contains a stop control structure, set the Has Stop Control Structure? field to **True**.

5. To define a control structure, click the **Ellipses (...)** button next to the Start Control Structure field and/or the Stop Control Structure, then perform the following steps:
    a. In the Conduit Control Structure dialog box, click the **New** button, then select the type of control structure you want to create from the submenu (Functional, Orifice, Depth-Flow Curve, or Weir).
    b. For a Functional control structure, specify whether the structure has a flap gate or not, then enter values for Crest Elevation, Coefficient, and Exponent.
    c. For an Orifice control structure, specify whether the structure has a flap gate or not, enter values for Crest Elevation and Orifice Coefficient, select the Orifice Type (Bottom Outlet or Side Outlet) and Orifice Shape (Circular or Rectangular), then enter values for Diameter for a Circular orifice or Height and Width for a Rectangular orifice.
    d. For a Depth-Flow Curve orifice, specify whether the structure has a flap gate or not, enter a value for Crest Elevation, then enter Depth and Flow values in the Depth-Flow table.
    e. For a Weir control structure, specify whether the structure has a flap gate or not, enter values for Crest Elevation, Weir Coefficient, and Structure Top Elevation, then select the Weir Type (Inline, Side, V-Notch, or Trapezoidal) and enter values for the selected Weir Type.
    f. Repeat Steps a - e for each additional component you want to add to the control structure, then click **Close**.

6. Perform the following optional steps:
    – To delete a component from the control structure, select the item in the list pane then click **Delete**.
    – To rename a control structure, select the control structure you want to rename, click **Rename**, then type the new name.
    – To view a report on the control structure, click **Report**.
    – To view a graph of a Depth-Flow curve, click **Graph** above the Depth-Flow curve table.

Conduit Control Structure Dialog Box

The Conduit Control Structure dialog box lets you add and edit control components to the control structure of a conduit.

	**New**	Lets you create a new component for your control structure. These attributes are **Functional**, **Orifice**, **Depth-Flow Curve**, and **Weir**.
	**Delete**	Lets you delete the selected attribute.
	**Rename**	Lets you rename the control structure. By default, the structure is given a unique name, but you can set any other unique name that you prefer.
	**Report**	Lets you create and view a report of the control structure.

Depending on the type of control structure you select, the Conduit Control Structure dialog box contains the following controls:

Functional

Has Flap Gate?	Select this check box if the component includes a flap gate, clear it if not. When checked, reverse flow is not allowed through the structure.
Crest Elevation	The structure elevation at which flow starts to occur.
Coefficient	Set the coefficient.
Exponent	Set the exponent.

Orifice

Has Flap Gate?	Select this check box if the component includes a flap gate, clear it if not. When checked, reverse flow is not allowed through the structure.
Orifice Coefficient	Set the coefficient.
Orifice Type	Select the type of orifice from the drop-down list: **Side Outlet** or **Bottom Outlet**. A Bottom Outlet type refers to a drop orifice (flow drops through it). A Side Outlet type refers to a normal flow-by orifice

Orifice Shape	Choose a circular or rectangular orifice.
Orifice Diameter	Set the diameter of a circular orifice.
Orifice Height	Set the height for a rectangular orifice.
Orifice Width	Set the width for a rectangular orifice.

Depth-Flow Curve

Has Flap Gate?	Select this check box if the component includes a flap gate, clear it if not. When checked, reverse flow is not allowed through the structure.
Depth-Flow	Set the depth and flow values that define the depth-flow curve. The depth-flow curve consists of a table of the depth and pairs that define the depth-flow curve. Depth in this case refers to the water depth above the invert on the upstream side of the control structure and flow is the flow through/over the control structure.
New	Click this button to add new rows to the depth-flow table.
Delete	Click this button to remove selected rows from the depth-flow table. You can only remove one row at a time, if you have selected more than one row, only the last row you selected is removed
Graph	Click this button to view a graph of the Depth-Flow curve.

Weir

Has Flap Gate?	Select this check box if the component includes a flap gate, clear it if does not. When checked, reverse flow is not allowed through the structure.
Side Slope	Set the side slope of the weir.
Weir Coefficient	Set the weir coefficient, which is also known as the coefficient of flow. This value is used to account for variables that are not otherwise directly accounted for in the weir equation (e.g., vena contracta).

Structure Top Elevation	Set the structure top elevation for the weir.
Weir Type	Select the type of weir from the drop-down list: inline, side, V-notch, or trapezoidal.
Number of Contractions	Set the number of contractions for an inline weir. This field is available only when you select Inline Weir as the Weir Type.
Weir Length	Set the length of the weir.
Weir Angle	Set the weir angle for a V-notch weir. This field is available only when you select V-Notch Weir as the Weir Type.
Weir End Coefficient	Discharge coefficient for flow through the triangular ends of a trapezoidal weir. Typical values are 2.4 - 2.8 US (1.35 - 1.55 SI). This field is available only when you select Trapezoidal Weir as the Weir Type.
Weir Side Slope	Set the slope of the side of a trapezoidal weir. This field is available only when you select Trapezoidal Weir as the Weir Type.

Adding a Minor Loss Collection to a Pressure Pipe

Pressure pipes can have an unlimited number of minor loss elements associated with them. SewerGEMS V8*i* provides an easy-to-use table for editing these minor loss collections in the Minor Loss Collection dialog box.

Note: For more information on minor losses, see “Minor Losses” on page 14-747**.**

To add a minor loss collection to a pressure pipe:

1. Click a pressure pipe in your model to display the Property Editor, or right-click a pressure pipe and select **Properties** from the shortcut menu.
2. In the Physical: Minor Losses section of the Property Editor, click the **Ellipses (...)** button next to the Minor Loss Coefficient field.
3. In the Minor Loss Collection dialog box, each row in the table represents a single minor loss type and its associated headloss coefficient. For each row in the table, perform the following steps:

a. Type the number of minor losses of the same type to be added to the composite minor loss for the pipe in the Quantity column, then press the Tab key to move to the Minor Loss column.

b. Click the **Ellipses (...)** button in the Minor Loss column to display the Minor Loss Libraries in the Engineering Libraries.

c. Click the plus signs to expand the Minor Loss Libraries, then select the desired minor loss type and click the **Select** button. The minor loss type and its associated headloss coefficient appears in the table in the Minor Loss Collection dialog box. Note that the Headloss Coefficient column in the table is not editable.

Note: You can edit the values of a minor loss type in the Engineering Libraries in the Editor pane.

4. When you are finished adding minor losses to the table, click **Close**. The composite minor loss coefficient for the minor loss collection appears in the Property Editor.
5. Perform the following optional steps:
   - To delete a row from the table, select the row label then click **Delete**.
   - To view a report on the minor loss collection, click **Report**.
6. You can override the headloss coefficient for the minor loss collection by typing a custom value in the Minor Loss Coefficient field of the Property Editor.

Minor Loss Collection Dialog Box

The Minor Loss Collection dialog box contains buttons and a minor loss table. The dialog box contains the following controls:

	**New**	This button creates a new row in the table.
	**Delete**	This button deletes the currently highlighted row from the table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.

The table contains the following columns:

Column	Description
Quantity	The number of minor losses of the same type to be added to the composite minor loss for the pipe.
Minor Loss	The type of minor loss element. Clicking the Ellipses button next to this field displays the Minor Loss Engineering Libraries, where you select an existing minor loss type to be included in your minor loss collection.
Headloss Coefficient	The headloss coefficient for a single minor loss element of the specified type.

Defining the Geometry of a Link Element

You define the geometry of a link element by entering the location and angle of bends for the selected link element. You enter X vs. Y points that plot the shape of the polyline that represents the element in the Polyline Vertices dialog box .

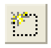
To define the geometry of a link element:

1. Click a link element in your model to display the Property Editor, or right-click a link element and select **Properties** from the shortcut menu.
2. In the Geometry section of the Property Editor, click the **Ellipses (...)** button next to the Geometry field.
3. In the Polyline Vertices dialog box, click the **New** button to add a new row to the table.
4. Type values for X and Y points for each row in the table.
5. To remove rows from the table, click the **Delete** button.
6. Click **OK**.

Polyline Vertices Dialog Box

This dialog box contains the *X* vs. *Y* table that allows you to define any number of points that plot the shape of the polyline representing the selected link element.

The dialog box contains the following controls:

	**New**	This button creates a new row in the table.
	**Delete**	This button deletes the currently highlighted row from the table.

Defining the Cross-Sectional Shape of a Link Element

You define the cross-sectional shapes of link elements in their respective Property Editor as follows:

- Define the cross-sectional shape of a conduit section by entering data in the Physical section of the element’s Property Editor.
- Define the cross-sectional shape of an irregular conduit section by entering Station vs. Depth data in the Station-Depth Curve dialog box.
- Define the cross-sectional shape of an irregular channel or gutter section by entering Station vs. Elevation data in the Station-Elevation Curve dialog box.
- Define the cross-sectional shape of a trapezoidal channel or gutter section by entering data in the Physical section of the element’s Property Editor.
- Define the circular shape of a pressure pipe by entering data in the Physical section of the element’s Property Editor.

You access the curve dialog boxes in the selected link element’s Property Editor.

Note: Although you can have complex channels, the algorithm does not support split flows or bridges. If you split the channel within the cross section, a constant water surface across the cross section is assumed.

When the elevation of the water surface exceeds the highest elevation in the table, the last two unsubmerged points are linearly extrapolated to create a new, wider channel.

To define the cross-sectional shape of a link element:

1. Display the Property Editor for the link element:
    a. For a conduit, gutter, or pressure pipe, click the link element in your model, or right-click the link element and select **Properties** from the shortcut menu.
    b. For a channel, click the connecting cross-section node in your model, or right-click the channel and select **Properties** from the shortcut menu.

2. In the Physical section of the Property Editor for the selected link element, define the cross-section of the selected link element as follows:
   – For a cross-section (channel link element) or a gutter, select either **Trapezoidal Channel** or **Irregular Channel** as the Section Type. For trapezoidal channels, enter data in the appropriate fields. If you select Irregular Channel, the Station-Elevation Curve field becomes available. Click the **Ellipses (...)** button next to the Station-Elevation Curve field to display the Station-Elevation Curve dialog box, then type values for station and elevation in the table.
   – For a conduit, select a section type, then enter data in the appropriate fields. If you select Irregular Channel as the Section Type, the Station-Depth Curve field becomes available. Click the **Ellipses (...)** button next to the Station-Depth Curve field to display the Station-Depth Curve dialog box, then type values for station and depth in the table.
   – For a pressure pipe, which always has a circular section shape, enter data in the appropriate fields.

Station-Elevation Curve/Depth Dialog Box

This dialog box allows you to enter Station vs. Elevation data for the cross-sectional shape of a cross-section or a gutter element, or Station vs. Depth data for the cross-sectional shape of a conduit.

The dialog box contains the station vs. elevation table along with the following controls:

	**New**	This button creates a new row in the station-elevation table.
	**Delete**	This button deletes the currently highlighted row from the station-elevation table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the station-elevation curve defined by the points in the table

The table contains the following columns:

Column	Description
Station	This field allows you to define the cross-sectional distance at the current curve point. You can enter these in any order that defines the channel (e.g., from left-to-right, from right-to-left, with an upstream or downstream perspective).
Elevation/Depth	This field allows you to define the elevation for the current curve point for a cross-section or gutter, or depth for a conduit. This value can be a negative number.

Defining Manning’s n vs. Depth Curves

You can define the roughness type for a conduit or a channel by creating a Manning’s n vs. Depth curve. You define the curve for a channel in the Property Editor for the connecting cross section node.

To define a Manning’s n vs. Depth curve for a conduit or a channel:

1. Display the Property Editor for the link element:
    a. For a conduit, click the conduit in your model, or right-click the conduit and select **Properties** from the shortcut menu.
    b. For a channel, click the connecting cross-section node in your model, or right-click the channel and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, select **Manning’s n - Depth Curve** as the Roughness Type. The Manning’s n-Depth Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Manning’s n - Depth Curve field.
4. In the Manning’s n - Depth Curve dialog box, each row in the table represents a depth value and its associated Manning’s n roughness value. For each row in the table, perform the following steps:
    a. Type the depth in the Depth column, then press the **Tab** key to move to the Manning’s n column.
    b. Type a value in the Manning’s n column or click the **Ellipses (...)** button in the column to display the Material Libraries in the Engineering Libraries.
    c. Click the plus signs to expand the Material Libraries, then select the desired material and click the **Select** button. The Manning’s n roughness value associated with the selected material appears in the table.

Note: You can edit the values of a minor loss type in the Engineering Libraries in the Editor pane.

5. Click the **New** button to add a row to the table.
6. Repeat Steps 4 and 5 for each new row of values in the table.
7. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
8. Click **OK** to close the dialog box and save the curve data in the Property Editor.

Manning's n–Depth Curve Dialog Box

This dialog box lets you define a Manning's n vs. Depth table for conduits and channels. The dialog box contains the Depth vs. Manning's table and the following buttons:

	**New**	Creates a new row in the table.
	**Delete**	Deletes the currently highlighted row from the table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.

The table contains the following columns:

Column	Description
Depth	Lets you define the depth of the depth vs. Manning's curve point.
Manning' n	Lets you define the roughness value at the specified depth in the Depth vs. Manning's table.

Defining Manning's n vs. Flow Curves

You can define the roughness type for a conduit or a channel by creating a Manning's n vs. Flow curve. You define the curve for a channel in the Property Editor for the connecting cross section node.

To define a Manning's n vs. Flow curve for a conduit or a channel:

1. Display the Property Editor for the link element:
   a. For a conduit, click the conduit in your model, or right-click the conduit and select **Properties** from the shortcut menu.
   b. For a channel, click the connecting cross-section node in your model, or right-click the channel and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, select **Manning's n - Flow** as the Roughness Type. The Manning's-Flow Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Manning's n - Flow Curve field.
4. In the Manning's n - Flow Curve dialog box, each row in the table represents a depth value and its associated Manning's n roughness value. For each row in the table, perform the following steps:
   a. Type the depth in the Flow column, then press the **Tab** key to move to the Manning's n column.
   b. Type a value in the Manning's n column or click the **Ellipses (...)** button in the column to display the Material Libraries in the Engineering Libraries.
   c. Click the plus signs to expand the Material Libraries, then select the desired material and click the **Select** button. The Manning's n roughness value associated with the selected material appears in the table.

   Note: You can edit the values of a minor loss type in the Engineering Libraries in the Editor pane.
5. Click the **New** button to add a row to the table.
6. Repeat Steps 4 and 5 for each new row of values in the table.
7. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
8. Click **OK** to close the dialog box and save the curve data in the Property Editor.

Manning's n–Flow Curve Dialog Box

This dialog box lets you define Flow vs. Manning's n tables for conduits and channels.

The dialog box contains the flow vs. Manning's table and the following buttons:

	New	Creates a new row in the table.
	Delete	Deletes the currently highlighted row from the table.
	Report	Opens a print preview window containing a report that details the input data for this dialog box.

The table contains the following columns:

Column	Description
Flow	Lets you define the flow of the flow vs. Manning's curve point.
Manning' n	Lets you define the roughness value at the specified flow in the Flow vs. Manning's table.

C-Depth Table Dialog Box

This dialog box allows you to enter Depth vs. C data for a weir associated with a conduit.

The dialog box contains the Depth vs. C table along with the following controls:

	New	This button creates a new row in the Depth vs. C table.
	Delete	This button deletes the currently highlighted row from the Depth vs. C table.
	Report	Opens a print preview window containing a report that details the input data for this dialog box.
	Graph	Opens a graph window plotting the Depth vs. C curve defined by the points in the table.

The table contains the following columns:

Column	Description
Depth	This field allows you to define the depth at the current curve point. You can enter these in any order that defines the weir (e.g., from left-to-right, from right-to-left, with an upstream or downstream perspective).
C	This field allows you to define the C value for the current curve point.

Depth Width Curve Dialog Box

A depth-width curve is a method of describing a closed conduit that is not a standard shape. The Depth Width Curve dialog box lets you enter points on a depth-width curve for a conduit whose Section Type is defined as Irregular Closed Section.

The dialog box contains the depth vs.width table and the following controls:

	**New**	This button creates a new row in the depth-width table.
	**Delete**	This button deletes the currently highlighted row from the depth-width table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the depth-width curve defined by the points in the table

The table contains the following columns:

Column	Description
Depth	Lets you define the depth of the curve point.
Width	Lets you define the width of the conduit at a specific depth.

Sections Results Dialog Box

The Section Results dialog box shows the calculated flow variables at the start, middle and end of a pipe, conduit, or channel section. You can view the data but you cannot edit it while in the dialog.

The dialog box contains a table displaying the section results and the following control:

	**Report**	Opens a print preview window containing a report of the sections results.

The table displays the following section results:

- Section Distance
- Section Velocity
- Section Flow
- Section Hydraulic Grade
- Section Depth
- Section Flow-Width
- Section Flow-Area
- Section Is Overflowing?
- Section Froude Number

To open this dialog box, go to the Results section of the Property Editor for a pipe, conduit, or a channel after the model has been calculated. Then click the **Ellipsis** (…) button in the Section Results field.

What Happens When the Water Level Exceeds the Top Elevation of an Open Channel?

When the hydraulic grade line (HGL) exceeds the channel top elevation, the last width defined for the channel (or the cross section node) is extended vertically to no limit. So there is no overflow for these. When the channel is bounded by a manhole, overflow occurs at the manhole.

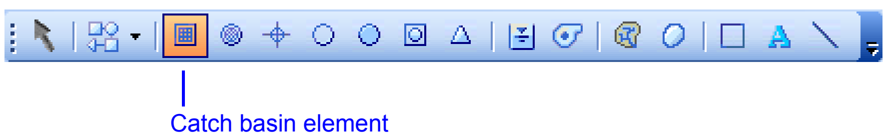
How Do Cross Section Nodes Control the Shape of Channel Cross-Sections?

When you connect a channel between two cross section nodes, the cross-section of the channel is interpolated between the two cross section nodes. When there is only one cross-section node at either the upstream or downstream end of a channel, then the channel will have a constant cross-section, as defined by that one cross-section node.

If there is a cross section node at each end of a channel, then the channel will start with a cross section as defined in the upstream cross-section node, and will make a transition to the cross-section defined in the downstream cross section node.

When you connect a channel to a conduit at a cross-section node, a transition is added between the channel and the conduit. You can specify the type of transition in the Property Editor for the cross-section node as either Gradual or Abrupt. If you specify Abrupt, the top width of the channel cross-section is used as the length of the transition part. If you select Gradual, you enter a value for the Transition Length. If the Transition Length is larger than the top width of the cross-section node, the Transition Length value is used as the length of the transition part.

Catch Basins



Catch basins convey surface water into a storm sewer pipe system. A catch basin (a.k.a., storm drain inlet, curb inlet) is an inlet to the storm drain system that typically includes a grate or curb inlet where stormwater enters the catch basin and a sump to capture sediment, debris and associated pollutants. They are also used in combined sewer watersheds to capture floatables and settle some solids.

When you click the catch basin element on the Layout toolbar, your mouse cursor changes into a catch basin element symbol. Clicking in the drawing pane while this tool is active causes a catch basin element to be placed at the location of the mouse cursor.

Inlet Type

The inflow to a catch basin does not all enter the basin. The flow that actually enters the basin is referred to as its "capture."

A catch basin may:

- Capture all the flow that comes to it, which is referred to as "full capture."
- Capture all of the flow up to a "maximum capacity," and you specify the maximum flow.
- Capture flow in accordance with some curve called an inflow vs. capture curve. For more information on inflow vs. capture curves, see "Adding Inflow vs. Capture Data to a Catch Basin" on page 6-208.

Any inflow that is not captured goes to a gutter. If there is no gutter, the inflow that is not captured is lost from the system.

Adding Inflow vs. Capture Data to a Catch Basin

You can add an Inflow vs. Capture curve to any catch basin in your model. The Inflow vs. Capture curve plots the total inflow against the total captured flow for a series of data points that you define.

To add Inflow vs. Capture data to a catch basin:

1. Click a catch basin in your model to display the Property Editor, or right-click a catch basin and select **Properties** from the shortcut menu.
2. In the Inlet section of the Property Editor, select **Inflow-Capture Curve** as the Inlet Type. The Inflow-Capture Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Inflow-Capture Curve field.
4. In the Inflow-Capture Curve dialog box, each row in the table represents a data point on the Inflow-Capture curve. Type values for the Total Inlet Flow and Inlet Capture for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Inflow-Capture Curve Dialog Box

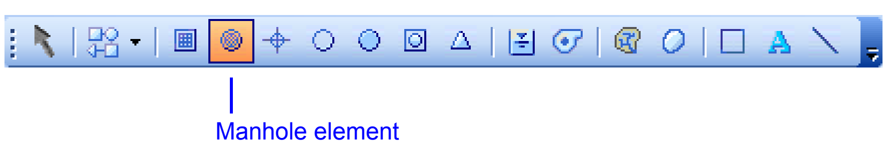
This dialog box allows you to define Total Inflow vs. Inlet Capture tables for catchments. The dialog box contains the inflow vs. capture table along with the following controls:

	New	Creates a new row in the inflow-capture curve table.
	Delete	Deletes the currently highlighted row from the inflow-capture curve table.
	Report	Opens a print preview window containing a report that details the input data for this dialog box.
	Graph	Opens a graph window plotting the inflow-capture curve defined by the points in the table.

The table contains the following columns:

Column	Description
Total Inlet Inflow	Lets you define the inflow at the current curve point. Total inlet flow is the cumulative flow from all catchments and other loads that actually reaches the catch basin.
Inlet Capture	Lets you define the total captured flow for the current curve point. Inlet capture is the portion of the total inlet flow that actually enters the catch basin and is passed downstream.

Manholes



Manholes are placed in a sewer system to provide access for inspection, maintenance, and emergency service. Manholes should be placed at sewer junctions (i.e., tees, wyes, and crosses), upstream terminal ends of sewers, and locations where there is a change in sewer grade or direction. Manholes are locations where loads enter the gravity portion of the sewer system.

When you click the manhole element on the Layout toolbar, your mouse cursor changes into a manhole element symbol. Clicking in the drawing pane while this tool is active causes a manhole element to be placed at the location of the mouse cursor.

Adding Surface Depth vs. Area Data to a Catch Basin or a Manhole

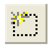
You can add a Surface Depth vs. Area curve to any catch basin or manhole in your model. The Surface Depth vs. Area curve plots depth against area for a series of data points that you define.

To add Surface Depth vs. Area data to a catch basin or a manhole:

1. Display the Property Editor for a catch basin or manhole by clicking the element in the Drawing Pane, or by right-clicking the element then selecting **Properties** from the shortcut menu.
2. In the Physical: Surface Storage section of the Property Editor, select **Surface Depth- Area Curve** as the Surface Storage Type. The Surface Depth-Area Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Surface Depth-Area Curve field.
4. In the Surface Depth-Area Curve dialog box, each row in the table represents a data point on the Surface Depth-Area curve. Type values for the Depth and Area for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Surface Depth-Area Curve Editor

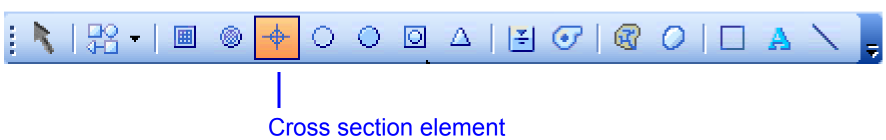
This dialog box allows you to define Depth vs. Area tables for manholes and catch basins. The dialog box contains the depth-area table and the following buttons:

	**New**	Creates a new row in the depth-area table.
	**Delete**	Deletes the currently highlighted row from the depth-area table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the surface depth-area curve defined by the points in the table.

The table contains the following columns:

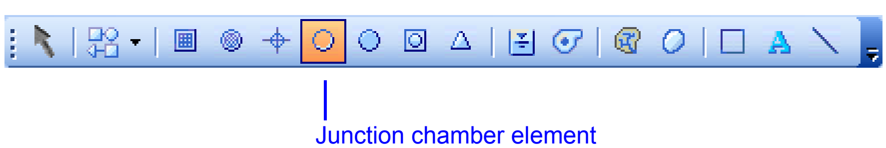
Column	Description
Depth	Lets you enter the depth data for the curve.
Area	Lets you enter the area data for the curve.

Cross Sections



When you click the cross section element on the Layout toolbar, your mouse cursor changes into a cross section element symbol. Clicking in the drawing pane while this tool is active causes a cross section element to be placed at the location of the mouse cursor.

Junction Chambers



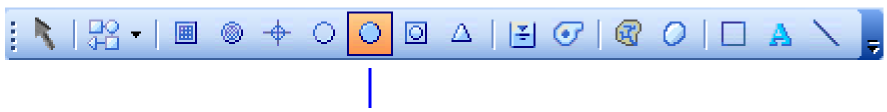
Junction chambers are locations where upstream flows in a gravity system combine. No loads enter the sewer at these points.

When you click the junction chamber element on the Layout toolbar, your mouse cursor changes into a junction chamber element symbol. Clicking in the drawing pane while this tool is active causes a junction chamber element to be placed at the location of the mouse cursor.

Related Topics

- "Junction Chamber Attributes" on page 15-905

Pressure Junctions



Pressure junction element

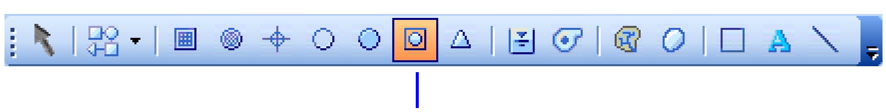
Pressure junctions are connections between two or more pressure pipes of varying characteristics. Loads may enter a pressure portion of a network through a pressure junction.

When you click the pressure junction on the Layout toolbar, your mouse cursor changes into a pressure junction element symbol. Clicking in the drawing pane while this tool is active causes a pressure junction element to be placed at the location of the mouse cursor.

Related Topics

- "Pressure Junction Attributes" on page 15-910

Pond Outlet Structures



Outlet structure element

When you click the outlet structure element on the Layout toolbar, your mouse cursor changes into a outlet structure element symbol. Clicking in the drawing pane while this tool is active causes a outlet structure element to be placed at the location of the mouse cursor.

Note: If there are multiple discharges locations serving a pond then they must all be modeled in the same manner; that is, they must either all be modeled with outlet control structures, or all modeled without outlet control structures.

Defining Composite Outlet Structures

SewerGEMS V8*i* lets you define composite outlet structures for pond outlet structures in your model. A composite outlet structure can contain any combination of orifices, risers, and weirs. You define these outlet structures in the Composite Outlet Structures dialog box.

To define a composite outlet structure:

1. Click a pond outlet structure in your model to display the Property Editor, or right-click a pond outlet structure and select **Properties** from the shortcut menu.
2. In the Pond Outlet section of the Property Editor, select **Yes** in the Has Control Structure field. The Control Structure field becomes available.
3. Click the **Ellipses (...)** button next to the Control Structure field. The Composite Outlet Structures dialog box appears, displaying the existing composite outlet structure associated with the selected pond outlet structure.
4. To create a new composite outlet structure, perform these steps:
    a. Click **New**, then select the type of outlet structure component you want to add (**Orifice**, **Riser**, or **Weir**).
    b. The right side of the dialog box displays settings for the composite outlet structure and for each individual component in the composite outlet structure. With the composite outlet structure selected in the list pane, enter values for Tolerance Settings and ICPM Settings.
    c. Select each individual component in the list pane, then enter values for that component in the fields on the right.
    d. If you define a weir as an Irregular Weir, you must define the cross-sectional shape of the irregular weir by entering X (Station) vs. Y (Depth) data. To do this, click the **Cross Section** button, enter X and Y values in the Irregular Cross Section dialog box, then click **OK to close that dialog box**.

5. To edit an existing composite outlet structure, perform these steps:
   a. If necessary, edit the Tolerance Settings and ICPM Settings for the selected composite outlet structure.
   b. Click the plus sign (+) next to the outlet structure in the list pane to display its individual components.
   c. Select the component (Orifice, Riser, or Weir), then edit the values for the component on the right side of the dialog box.
   d. If you define a weir as an Irregular Weir, you must define the cross-sectional shape of the irregular weir by entering X (Station) vs. Y (Depth) data. To do this, click the **Cross Section** button, enter X and Y values in the Irregular Cross Section dialog box, then click **OK to close that dialog box**.
   e. To add a new component to an existing composite outlet structure, select the outlet structure in the list pane then click **New** and select **Orifice**, **Riser**, or **Weir** from the submenu. You can also right click an existing composite structure in the list pane, then select **New > Orifice**, **New > Riser**, or **New > Weir** from the shortcut menu.

6. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To rename an existing composite outlet structure, click **Rename**, then type the new name.
   – To view a report on the composite outlet structure, click **Report**.
   – To view a plot of the composite outlet structure, click **Graph**. The graph will display multiple curves, one for each component in the composite outlet structure.

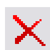
7. Click **OK** to close the dialog box and save your data in the Property Editor.

Composite Outlet Structures Dialog Box

This dialog box allows you to define and manage composite outlet structures for pond outlet structures. The dialog box is divided into three parts: the composite structure list pane on the left, the common settings section in the top-right, and the structure editor section in the bottom-right.

The Composite Outlet Structure List Pane, located on the left side of the dialog box, displays the composite outlet structures that are associated with the selected pond outlet structure. Clicking the plus sign next to each composite outlet structure displays the individual structures that make up the composite outlet structure. Highlighting a structure in the list causes the sections on the right side of the dialog box to display the data associated with the highlighted structure.

The dialog box contains the following controls:

	**New**	Creates a new composite outlet structure that uses an automatically created label. • **Orifice**—Adds a new orifice structure to the current composite outlet structure. • **Riser**—Adds a new riser structure to the current composite outlet structure. • **Weir**—Adds a new weir structure to the current composite outlet structure.
	**Delete**	Deletes the currently highlighted composite outlet structure.
	**Rename**	Lets you rename the currently highlighted outlet structure.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the entry that is currently highlighted in the Outlet Structure list pane .
	**Graph**	Lets you generate a graph of the composite outlet structure after you compute your model.

When a composite control structure is highlighted in the list pane, controls become available in the right side of the dialog, allowing you to define values for attributes associated with the control structure. These controls are divided into two sections: the Tolerance Settings section and the ICPM Settings section.

The following controls are found in the Tolerance Settings section:

Tolerance Settings

Maximum Iterations	This value specifies the maximum number of iterations that the program will calculate. The higher the number entered here, the longer the calculation may potentially take.

Minimum Hw Tolerance	If the program is checking computed HW elevations during the iterative HW convergence computations, this tolerance value specifies the minimum target convergence between the computed value and the known headwater value.
Maximum Hw Tolerance	If the program is checking computed HW elevations during the iterative HW convergence computations, this tolerance value specifies the maximum allowable difference between the computed value and the known headwater value.
Minimum Tw Tolerance	If the program is checking computed TW elevations during the iterative TW convergence computations, this tolerance value specifies the minimum target convergence and the maximum allowable difference between the computed value and the known tailwater value.
Maximum Tw Tolerance	If the program is checking computed TW elevations during the iterative TW convergence computations, this tolerance value specifies the maximum allowable difference between the computed value and the known tailwater value.
Minimum Q Tolerance	If the program is checking computed flow during the iterative TW convergence computations, this tolerance value specifies the minimum target convergence between the computed value, and the known flow value.
Maximum Q Tolerance	If the program is checking computed flow during the iterative TW convergence computations, this tolerance value specifies the maximum allowable difference between the computed value, and the known flow value.

ICPM Settings

Override Hw Range	Checking this box causes the three headwater range attribute fields to become editable, allowing you to override the automatically calculated settings.

Headwater Step	Lets you define the interval at which headwater will be calculated. This field is only available for input when the Override Hw Range box is checked.
Minimum Headwater Elevation	Lets you define the minimum headwater elevation. This field is only available for input when the Override Hw Range box is checked.
Maximum Headwater Elevation	Lets you define the maximum headwater elevation. This field is only available for input when the Override Hw Range box is checked.
Override Tw Range	Checking this box causes the three tailwater range attribute fields to become editable, allowing you to override the automatically calculated settings.
Tailwater Step	Lets you define the interval at which tailwater will be calculated. This field is only available for input when the Override Tw Range box is checked.
Minimum Tailwater Elevation	Lets you define the minimum tailwater elevation. This field is only available for input when the Override Tw Range box is checked.
Elevation (Maximum Tailwater)	Lets you define the maximum tailwater elevation. This field is only available for input when the Override Tw Range box is checked.

When a component of the control structure is highlighted in the list pane, a number of controls become available that allow you to define values for the various attributes that are associated with the structural component.

The specific controls available to you varies somewhat, depending on the type of component being edited. The following controls are available in this section of the dialog box:

Orifice Common Settings

Flow Direction	Lets you specify the flow direction as bi-directional (forward or reverse), forward only, or reverse only.
Elevation	Lets you define the invert elevation, which is the elevation of the lowest point of the orifice opening or channel.

# of Openings	Lets you define the number of openings the orifice uses.
Orifice Coefficient	Lets you define the orifice coefficient. Although the orifice coefficients vary with shape, size, and head depth, an average C coefficient of 0.60 is often used for stormwater orifice openings. *Handbook of Hydraulics* (Brater and King 1976) lists orifice coefficients for various heads and sizes of circular, square, rectangular, and triangular shapes. *Engineering Field Manual for Conservation Services* (U.S. Soil Conservation Service 1986a) provides a chart of orifice coefficients for orifice plates placed over pipe openings.
Orifice Settings	The following data is required to define an Orifice outlet structure.
Orifice	Lets you choose between a circular orifice or one with a known area.
Diameter	Set the diameter of the clear opening for a circular orifice. This field is available only when the Orifice Type is Circular.
Orifice Area	This field allows you to set the orifice area, the clear opening area for each individual orifice opening. For example, if your outlet structure has 10 rectangular orifices that measure 2 ft. x 3 ft., you would enter an area of 6 sq. ft. not 60 sq. ft. This field is available only when the Orifice Type is Area.
Orifice Orientation	Lets you specify whether the orifice is parallel to or perpendicular to the direction of flow. This menu is available only when the Orifice Type is Area.
Datum Elevation	Lets you set the elevation from which to measure the orifice head (H) when a perpendicular orifice orientation is specified. This field is available only when the Orifice Type is Area and the Orifice Orientation is Perpendicular.

Elevation (Top)	Lets you define the highest elevation in the channel. Elevations above this value are considered to be overtopping the channel. For a perpendicular orifice, the top elevation is considered to be the highest point of the orifice opening. This field is available only when the Orifice Type is Area and the Orifice Orientation is Perpendicular.

Riser Common Settings

Flow Direction	Lets you specify the flow direction as bi-directional (forward or reverse), forward only, or reverse only.
Elevation	Lets you define the invert elevation, which is the elevation of the lowest point of the orifice opening or channel.
Weir Coefficient	This field allows you to define the weir coefficient, which is also known as the coefficient of flow. This value is used to account for variables that are not otherwise directly accounted for in the weir equation (e.g., vena contracta). **Note: Metric users must input the weir coefficient for U.S. customary units. Otherwise, a computational error results.**
Orifice Coefficient	Lets you define the orifice coefficient. Although the orifice coefficients vary with shape, size, and head depth, an average C coefficient of 0.60 is often used for stormwater orifice openings. *Handbook of Hydraulics* (Brater and King 1976) lists orifice coefficients for various heads and sizes of circular, square, rectangular, and triangular shapes. *Engineering Field Manual for Conservation Services* (U.S. Soil Conservation Service 1986a) provides a chart of orifice coefficients for orifice plates placed over pipe openings

Riser Settings	The following data is required to define a Riser outlet structure:
Riser	Lets you specify whether the riser is a stand pipe or an inlet box.
Diameter	Lets you set the diameter, which is used to compute the weir length (perimeter) and orifice area. This field is available only when the Riser Type is Stand Pipe.
Weir Length	Lets you set the weir length, the effective weir length for the inlet box. This value is used for computing flow when the inlet box is assumed to be operating under weir conditions. This field is available only when the Riser Type is Inlet Box.
Orifice Area	Lets you set the orifice area, the clear opening area for each individual orifice opening. For example, if your outlet structure has 10 rectangular orifices that measure 2 ft. x 3 ft., you would enter an area of 6 sq. ft. not 60 sq. ft. This field is available only when the Riser Type is Inlet Box.

Riser Advanced Options

Transition Elevation	Lets you define the elevation corresponding to the beginning of the transition zone. **Note: If the transition elevation is left blank, the value is automatically set such that the center of the transition zone is located at the elevation where weir flow equals orifice flow. In other words, half the transition height is below the equilibrium point and half the height is above.**
Transition Height	Lets you define the height of the transition zone.

K Reverse — The value specified in this field is used when the stand pipe or inlet box is under submerged or reverse flow conditions. The K coefficient is multiplied by the velocity head, (V2/2g), to determine the additional headloss due to entrance and exit losses.

Weir Common Settings

Flow Direction — Lets you specify the flow direction as bi-directional (forward or reverse), forward only, or reverse only.

Elevation — Lets you define the invert elevation, which is the elevation of the lowest point of the orifice opening or channel.

Weir Coefficient — This field allows you to define the weir coefficient, which is also known as the coefficient of flow. This value is used to account for variables that are not otherwise directly accounted for in the weir equation (e.g., vena contracta).

Note: **Metric users must input the weir coefficient for U.S. customary units. Otherwise, a computational error results.**

Weir Settings

The following data is required to define a Weir outlet structure:

Weir — Lets you specify whether the weir is Rectangular, Irregular, or V-Notch.

V-Notch Angle — Lets you define the angle of the weir notch in degrees. This input field is available only when the Weir Type is V-Notch.

Cross Section — Opens the **Irregular Weir Cross Section** dialog box. This button is available only when the Weir Type is Irregular.

Rectangular Weir — Lets you specify whether the rectangular weir is suppressed or contracted. This menu is available only when the Weir Type is Rectangular.

Weir Length	Lets you define the width of the clear opening space for a rectangular weir. This field is available only when the Weir Type is Rectangular.

Irregular Weir Cross Section Dialog Box

The Irregular Weir Cross Section dialog box allows you to define the cross-sectional shape of an irregular weir in a composite outlet structure by entering X (Station) vs. Y (Depth) data.

The Irregular Weir Cross Section dialog box is accessible only from within the Composite Outlet Structures dialog box when a weir component is selected in the list pane. For the selected weir, click the **Cross Section** button to display the Irregular Weir Cross Section dialog box.

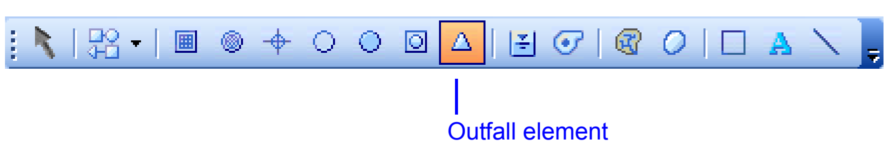
The dialog box contains the x vs. y table along with the following controls:

	**New**	This button creates a new row in the x-y table.
	**Delete**	This button deletes the currently highlighted row from the x-y table.

The table contains the following columns:

Column	Description
X	This field allows you to define the cross-sectional distance at the current point.
Y	This field allows you to define the depth (0 at the weir crest) for the current point.

Outfalls



Outfalls represent the ultimate termination points in a sanitary sewer network.

When you click the outfall element on the Layout toolbar, your mouse cursor changes into a outfall element symbol. Clicking in the drawing pane while this tool is active causes a outfall element to be placed at the location of the mouse cursor.

Adding Time vs. Elevation Data to an Outfall

You can define tidal curve tables for outfalls in SewerGEMS V8*i*. You define a tidal curve for an outfall as a Time vs. Elevation curve.

> **Note: You can also add a Cyclic Time vs. Elevation curve to an outfall. For more information, see** "Adding Cyclic Time vs. Elevation Data to an Outfall" on page 6-226**.**

To add a Time vs. Elevation curve to an outfall:

1. Click an outfall in your model to display the Property Editor, or right-click an outfall and select **Properties** from the shortcut menu.
2. In the Boundary Condition section of the Property Editor, select **Time-Elevation Curve** in the Boundary Condition Type field. The Time-Elevation Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Time-Elevation Curve field.
4. In the Time-Elevation Curve dialog box, each row in the table represents a point on the Time-Elevation curve. Type values for the Time and Elevation for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Time-Elevation Curve Dialog Box

This dialog box allows you to define tidal curve (time vs. elevation) tables for outfalls when the Boundary Condition is set to Time-Elevation Curve.

The dialog box contains the time-vs.-elevation table and the following buttons:

	**New**	This button creates a new row in the tidal curve table.
	**Delete**	This button deletes the currently highlighted row from the tidal curve table.
	**Report**	This button opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	This button opens a graph window plotting the time-elevation curve defined by the points in the table.

The table contains the following columns:

Column	Description
Time	This field allows you to define the hour of the tidal curve point.
Elevation	This field allows you to define the elevation for the tidal curve point.

Adding Elevation vs. Flow Data to an Outfall

You can add an Elevation-Flow (E-Q-T) curve to an outfall in SewerGEMS V8*i*. Each series of Elevation-Flow (E-Q-T) curves represents the performance curves for a pond outlet operating under various downstream tailwater (T) elevations.

To add an Elevation vs. Flow (E-Q-T) curve to an outfall:

1. Click an outfall in your model to display the Property Editor, or right-click an outfall and select **Properties** from the shortcut menu.
2. In the Boundary Condition section of the Property Editor, select **Elevation-Flow Curve** in the Boundary Condition Type field. The Elevation-Flow Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Elevation-Flow Curve field.

4. In the Elevation-Flow Curve dialog box, each row in the table represents a point on the Elevation-Flow curve. Type values for Elevation and Outlet Flow for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Elevation-Flow Curve Dialog Box

The dialog box contains the elevation vs. flow table and the following buttons:

	**New**	This button creates a new row in the E-Q-TW curve table.
	**Delete**	This button deletes the currently highlighted row from the E-Q-TW curve table.
	**Report**	This button opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	This button opens a graph window plotting the elevation-volume curve defined by the points in the table.

The table contains the following columns:

Column	Description
Outlet Elevation	This field allows you to define the elevation of the E-Q-TW curve point.
Outlet Flow	This field allows you to define the flow for the E-Q-TW curve point.

Adding Cyclic Time vs. Elevation Data to an Outfall

SewerGEMS V8*i* lets you add tidal curves to outfalls in your model. You define a tidal curve for an outfall as a Time vs. Elevation curve.

> **Note: You can also add a Time vs. Elevation curve to an outfall. For more information, see** “Adding Time vs. Elevation Data to an Outfall” on page 6-223**.**

To add a Cyclic Time vs. Elevation curve to an outfall:

1. Click an outfall in your model to display the Property Editor, or right-click an outfall and select **Properties** from the shortcut menu.
2. In the Boundary Condition section of the Property Editor, select **Tidal** in the Boundary Condition Type field. The Cyclic Time-Elevation Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Cyclic Time-Elevation Curve field.
4. In the Cyclic Time-Elevation Curve dialog box, each row in the table represents a point on the Cyclic Time-Elevation curve. Type values for Time and Elevation for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Cyclic Time-Elevation Curve Dialog Box

This dialog box allows you to define tidal curve (time vs. elevation) tables for outfalls when the Boundary Condition is set to Tidal.

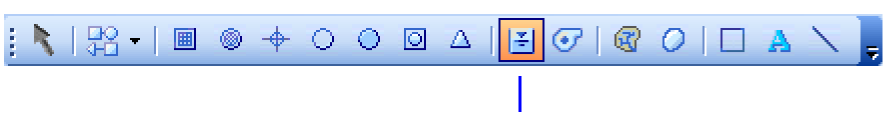
The dialog box contains the time-vs.-elevation table and the following buttons:

	**New**	This button creates a new row in the tidal curve table.
	**Delete**	This button deletes the currently highlighted row from the tidal curve table.
	**Report**	This button opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	This button opens a graph window plotting the time-elevation curve defined by the points in the table.

The table contains the following columns:

Column	Description
Time	This field allows you to define the hour of the tidal curve point.
Elevation	This field allows you to define the elevation for the tidal curve point.

Wet Wells



Wet wells are required at a pumping station to store wastewater before it is pumped. Wet wells represent boundary conditions between pressure and gravity portions of a sewer network. They serve as collection points for gravity systems, and as an HGL boundary node for the pressure system. Dry loads can also enter the sewer network at these locations.

When you click the wet well element on the Layout toolbar, your mouse cursor changes into a wet well element symbol. Clicking in the drawing pane while this tool is active causes a wet well element to be placed at the location of the mouse cursor.

Adding Depth vs. Area Data to a Wet Well

You can add Depth vs. Area data to a wet well in your model. You define the Depth vs. Area curve for a wet well in the Wet Well Depth-Area Curve dialog box.

To add a Depth vs. Area curve to wet well:

1. Click a wet well in your model to display the Property Editor, or right-click a wet well and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, select **Wet Well Depth-Area Curve** in the Wet Well Boundary Type field. The Wet Well Depth-Area Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Wet Well Depth-Area Curve field.
4. In the Wet Well Depth-Area Curve dialog box, each row in the table represents a point on the Wet Well Depth-Area curve. Type values for Depth and Area for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Wet Well Depth-Area Curve Dialog Box

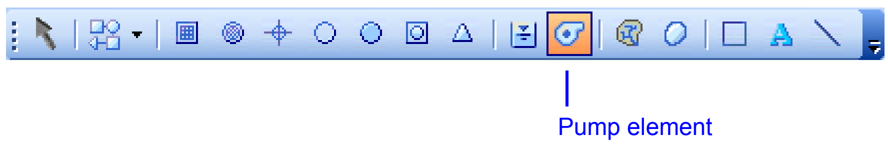
This dialog box allows you to define Depth vs. Area tables for wet wells. The dialog box contains the depth-area table and the following buttons:

	**New**	Creates a new row in the depth-area table.
	**Delete**	Deletes the currently highlighted row from the depth-area table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the surface depth-area curve defined by the points in the table.

The table contains the following columns:

Column	Description
Depth	Lets you enter the depth data for the curve.
Area	Lets you enter the area data for the curve.

Pumps



In a wastewater collection system, pumps are placed where the hydraulic grade line must be raised. Since sewage primarily flows by gravity, a pump transports sewage from a low elevation to a higher elevation. The sewage then flows again by gravity to the next pumping station or until it reaches its destination.

When you click the pump element on the Layout toolbar, your mouse cursor changes into a manhole pump symbol. Clicking in the drawing pane while this tool is active causes a manhole pump to be placed at the location of the mouse cursor.

Defining Pump Settings

You define the on/off settings for each pump in your model in the Pumps dialog box. You can define a collection of pump settings for each pump.

To define pump settings:

1. Click a pump in your model to display the Property Editor, or right-click a pump and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, click the **Ellipses (...)** button next to the Pumps field. The Pumps dialog box appears.
3. In the Pumps dialog box, each row of the table represents a separate set of pump settings. Click the **New** button to add a row to the table.
4. For each row in the table, perform these steps:
   a. Type a unique label for the pump settings.
   b. Select an existing pump curve from the Pump Definition field submenu, or click the **Ellipses** button to display the Pump Curve Definitions dialog box, where you can create new pump curve definitions or modify existing defini-

tions. Click **Close** to close the Pump Curve Definitions dialog box. New pump curve definitions automatically appear in the Pump Definitions field submenu.

c. Determine whether the pump is on or off by selecting On or Off in the Pump Setting field.

d. Type values for On and Off Elevations. If the suction element is a node, the elevation will be determined as the water surface elevation in that element; if it is a pressure pipe, then it is the water surface elevation in the node at the upstream end of the pressure pipe. The pump turns on when the calculated elevation exceeds the "on" setting and turns off then the calculated elevations drops below the "off" setting.

5. Click **OK** to close the Pumps dialog box and save your data in the Property Editor.

Pumps Dialog Box

This dialog allows you to define the pump settings for the currently highlighted pump. It consists of controls that allow you to add and delete pumps to the table, and to generate a report for the currently highlighted pump(s).

	**New**	Creates a new entry in the Pump Table.
	**Delete**	Deletes the entry that is currently highlighted in the Pump Table.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted pumps.

The table consists of the following columns:

Column	Description
Pump Definition	Lets you specify the pump curve definition to be associated with the current pump. The pulldown menu in this field lists all of the definitions that are associated with the current project. Clicking the ellipsis button in this field opens the Pump Curve Definitions dialog, allowing you to create new definitions.

Column	Description
Pump Setting	Lets you specify whether the pump is initially on or off.
On Elevation	Lets you define a control elevation for the pump. If the water surface elevation of the element upstream of the pump (typically a wet well or pond - defined in the Suction-side Node field) exceeds the value entered in this field, the pump will turn on.
Off Elevation	Lets you define a control elevation for the pump. If the water surface elevation of the element upstream of the pump (typically a wet well or pond - defined in the Suction-side Node field) exceeds the value entered in this field, the pump will turn off.

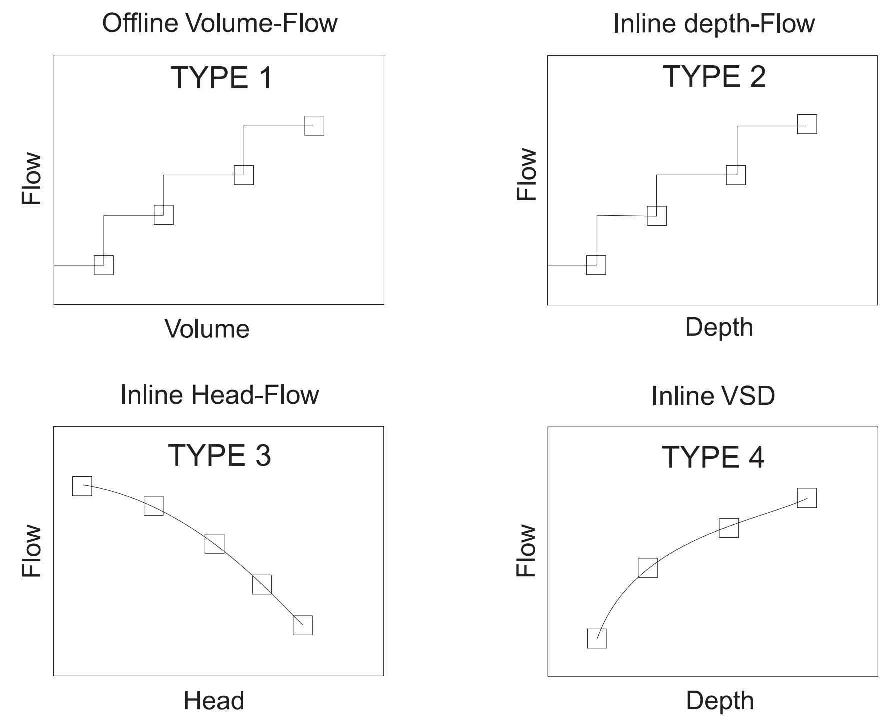
Creating Pump Curve Definitions

In SewerGEMS V8*i*, you can define pump curves of the following types:

- Offline Volume-Flow (Type I, step-wise curve)
- Inline Depth-Flow (Type-II, step-wise curve)
- Inline Head-Flow (Type-III, continuous curve)
- Inline Variable-Speed Pump (Type IV, continuous curve)

Note: These pump curve types are defined in the dialog box section of this help topic below.

The following illustration shows the four different pump curve types.



You define pump curves in the Pump Curve Definitions dialog box. You can also define pump curves in the Engineering Library.

To create a pump curve definition:

1. Select **Analysis** > **Pump Curve Definitions**.
2. Click **New** to create a new pump curve definition.
3. For each pump curve definition, perform these steps:
   a. Select the type of pump curve in the Pump Curve Type submenu (**Offline Volume-Flow Curve**, **Inline Depth-Flow**, **Inline Head-Flow**, or **Inline Variable Speed Pump**).
   b. Type values for volume vs. flow, head vs. flow, or depth vs. flow in the curve table. The columns change depending on which curve type you choose.
   c. Click the **New** button above the curve table to add a new row to the table, or press the **Tab** key to move to the next column in the table.

d. Click the **Delete** button above the curve table to delete the currently highlighted row from the table.

e. Click the **Graph** button above the curve table to view a plot of the curve.

4. You can save your new pump curve definition in Bentley SewerGEMS V8*i*' Engineering Libraries for future use. To do this, perform these steps:

   a. Click the **Synchronization Options** button, then select **Export to Library**. The Engineering Libraries dialog box appears.

   b. Use the plus and minus signs to expand and collapse the list of available libraries, then select the library into which you want to export your new unit sanitary load.

   c. Click **Close** to close the Engineering Libraries dialog box.

5. Perform the following optional steps:

   – To delete a pump curve definition, select the curve label then click **Delete**.

   – To rename a pump curve definition, select the label of the pump curve definition you want to rename, click **Rename**, then type the new name.

   – To view a report on a pump curve definition, select the label for the pump curve definition, then click **Report**.

6. Click **Close** to close the dialog box.

Pump Curve Definitions Dialog Box

This dialog box allows you to create pump curve definitions. There are two sections: the Pump Curve Definition Pane on the left and the tab section on the right. The Pump Curve Definition Pane lets you create, edit, and delete pump curve definitions.

The following controls are available in the Pump Curve Definitions dialog box:

	**New**	Creates a new entry in the Pump Curve Definition Pane.
	**Delete**	Deletes the currently highlighted entry in the Pump Curve Definition Pane.
	**Rename**	Lets you rename the currently highlighted entry in the Pump Curve Definition Pane.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted entry in the Pump Curve Definition Pane.
	**Synchronization Options**	Clicking this button opens a submenu containing the following commands: • **Browse Engineering Library**—Opens the Engineering Library manager dialog, allowing you to browse the Pump Curve Libraries. • **Synchronize From Library**—Lets you update a set of pump curve definition entries previously imported from a Pump Curve Engineering Library. The updates reflect changes that have been made to the library since it was imported. • **Synchronize To Library**—Lets you update an existing Pump Curve Engineering Library using current pump curve definition entries that were initially imported but have since been modified. • **Import From Library**—Lets you import pump curve definition entries from an existing Pump Curve Engineering Library. • **Export To Library**—Lets you export the current pump curve definition entries to an existing Pump Curve Engineering Library.

The tab section includes the following controls:

Pump Curve Data Tab

Pump Curve Type

This menu allows you to choose which type of pump curve to define. Note that the choice made here affects the columns in the pump curve points table.

- **Offline Volume-Flow Curve—**An off-line pump with a wet well where flow increases incrementally with wet well volume. The curve relates the volume of the source element to the outflow of the pump station. As the volume increases, the discharge increases.When this pump curve type is chosen, the table contains the following columns:
  - **Flow**—This column allows you to define the flow at the current curve point.
  - **Volume**—This column allows you to define the volume at the current curve point.
- **Inline Depth-Flow Curve—**An in-line pump where flow increases incrementally with node depth. This type of curve relates the depth of flow of the source element to the outflow of the pump. When this pump curve type is chosen, the table contains the following columns:
  - **Depth**—This column allows you to define the depth at the current curve point.
  - **Flow**—This column allows you to define the flow at the current curve point.
- **Inline Head-Flow Curve—**An in-line pump where flow varies continuously with head difference between the inlet and outlet nodes. When this pump curve type is chosen, the table contains the following columns:
  - **Head (Pump)**—This column allows you to define the pump head at the current curve point.
  - **Flow**—This column allows you to define the flow at the current curve point.
- **Inline Variable Speed Pump—**A variable speed in-line pump where flow varies continuously with node depth. When this pump curve type is chosen, the table contains the following columns:
  - **Depth**—This column allows you to define the depth at the current curve point.
  - **Flow**—This column allows you to define the flow at the current curve point.

Pump Curve Data Table	The Pump Curve Data Table, located on the bottom right of the Pump Curve Data tab, allows you to define the points that make up the pump curve for the definition that is currently highlighted in the Pump Curve Definition Pane on the left. The following controls are available: • **New**—Adds a new row to the current table. • **Delete**—Deletes the currently highlighted row from the table. • **Graph**—Displays a plot of the current pump curve.
Notes Tab	This tab contains a text field that allows you to enter descriptive notes that will be associated with the pump that is currently highlighted in the Pump Curves Definition Pane.
Library Tab	This tab displays information about the pump that is currently highlighted in the Pump Curves Definition Pane. If the pump is derived from an engineering library, the synchronization details can be found here. If the pump was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the pump was not derived from a library entry.

Pump Curve Dialog Box

This dialog allows you to define the points that make up the pump curve that is associated with the Pump Curve Library entry that is currently highlighted in the Engineering Library Manager explorer pane.

The columns that are available in this dialog will vary depending on the Pump Curve Type that is selected in the menu of the same name in the Engineering Library. The possible columns are as follows, sorted by pump type:

Column	Description
Offline Volume-Flow Curve	An off-line pump with a wet well where flow increases incrementally with wet well volume. When this pump curve type is chosen, the table contains the following columns:

Column	Description
Volume	This column allows you to define the volume at the current curve point.
Flow	This column allows you to define the flow at the current curve point.
Inline Depth-Flow Curve	An in-line pump where flow increases incrementally with node depth. When this pump curve type is chosen, the table contains the following columns:
Depth	This column allows you to define the depth at the current curve point.
Flow	This column allows you to define the flow at the current curve point.
Inline Head-Flow Curve	An in-line pump where flow varies continuously with head difference between the inlet and outlet nodes. When this pump curve type is chosen, the table contains the following columns:
Head (Pump)	This column allows you to define the pump head at the current curve point.
Flow	This column allows you to define the flow at the current curve point.
Inline VSD	A variable speed in-line pump where flow varies continuously with node depth. When this pump curve type is chosen, the table contains the following columns:
Depth	This column allows you to define the depth at the current curve point.
Flow	This column allows you to define the flow at the current curve point.

Catchments

When you click the catchment element on the Layout toolbar, your mouse cursor changes into a catchment element symbol. Clicking in the drawing pane while this tool is active causes a catchment element to be placed at the location of the mouse cursor.

Hydrograph Methods

With the exception of purely sanitary flow systems with no wet weather effects, Bentley SewerGEMS V8*i* starts its hydraulic calculations from a hydrograph for each catchment. There are numerous ways of generating those hydrographs. Most involve starting with storm events ("Adding Storm Events" on page 7-395) then calculating a runoff hydrograph using one of the following methods:

- SCS
- Unit Hydrograph
- EPA SWMM
- Modified Rational Method
- RTK Unit Hydrograph Method

Virtually any hydrograph and loss method can be used with any numerical engine with the following exceptions:

1. Modified rational method only works with Implicit engine and only with peak intensity (IDF) rainfall.
2. If you choose the SWMM engine and specify EPA-SWMM runoff method, then all catchments must use that runoff method and the loss method specified on the calculation options manager (only Green -Amt, Horton, or SCS).

However, it is also possible to directly enter a hydrograph by specifying a user-defined hydrograph ("Adding User Defined Hydrographs" on page 7-359) for any catchment.

Snowmelt

Bentley SewerGEMS V8*i* does not directly calculate snowmelt based on snow pack. If snow melt is significant for a given model, the user should calculate a snow melt hydrograph separately and enter it using a user-defined hydrograph (see “Adding User Defined Hydrographs” on page 7-359). For methods to determine snowmelt rates, the user is referred to the Haestad Press publication “Stormwater Conveyance Modeling and Design” (see “Bentley Institute Press” on page A-924).

Specifying a Time of Concentration (Tc) Method for a Catchment

You can add Time of Concentration (Tc) Methods to a catchment in your model. SewerGEMS V8*i* supports 13 different methods, which are listed below. You define the TC Method in the TC Data Collection dialog box. You can define both single and multiple flow segments for a catchment.

Some types of Tc equations can apply to flow segments within a multiple-segment Tc calculation. Other Tc methods are equations intended to model the entire average subarea flow distance and slope in one single flow segment. When combining multiple flow segments to compute Tc, it is up to you to only combine Tc methods that can be modeled in combination with multiple flow segments.

To define the Tc Method for a catchment:

1. Click a catchment in your model to display the Property Editor, or right-click a catchment and select **Properties** from the shortcut menu.
2. In the Runoff section of the Property Editor, select **Modified Rational Method** in the Runoff Method field. The Tc (hours) field becomes available.
3. Click the **Ellipses (...)** button next to the Tc (hours) field. The Tc Data Collection dialog box appears.
4. Click **New**, then select a Tc Method from the submenu.
5. Different fields become available depending on which Tc Method you select. For each Tc Method, type values in the appropriate fields.
6. Click **OK** to close the dialog box and save your Tc Collection data (time of concentration in hours) in the Property Editor.

Tc Data Collection Dialog Box

This dialog box allows you to define the Time of Concentration method. Both single and multiple flow segments can be modeled in this dialog box.

The dialog box contains the Tc Method display pane, which lists all of the methods currently assigned to the catchment, a control section that allows you to edit the attributes associated with the method currently highlighted in the table, and the following buttons:

	**New**	Displays a submenu that allows you to specify the Tc method to be created.
	**Delete**	Deletes the currently highlighted method from the table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.

SewerGEMS V8*i* supports the following 13 methods, which are listed along with the required input data for each:

- **User Defined Tc**—The user-defined time of concentration (Tc) is a method that allows the direct input of the Tc rather than using an equation to calculate it. This method would be used when the Tc needs to be calculated using a methodology that is not supported by Bentley SewerGEMS V8*i*, or when a quick estimate of Tc is sufficient for the analysis.
  - **User Defined Tc**—Lets you explicitly define the Tc, rather than have it calculated for you using one of the other methods.
- **Carter**—This method requires the following input data:
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section.
- **Eagleson**—This method requires the following input data:
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Manning’s n**—Lets you enter the Manning’s roughness value of the catchment section.
  - **Hydraulic Radius**—Lets you define the hydraulic radius of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section.
- **Espey/Winslow**—This method requires the following input data:

- **Channel Factor**—Lets you define the Espey channelization factor of the catchment section.
- **Hydraulic Length**—Lets you define the flow length of the catchment section.
- **Slope**—Lets you define the slope of the catchment section.
- **Impervious**—Lets you define the percentage of impervious area of the catchment section.

- **FAA Equation**—This method requires the following input data:
  - **Overland Flow Length**—Lets you define the length of the overland pipe flow of the catchment section.
  - **Rational Method C—**Lets you define the rational C coefficient of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section.
- **Kerby/Hathaway**—This method requires the following input data:
  - **Manning's n**—Lets you enter the Manning's roughness value of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section.
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
- **Kirpich PA**—This method requires the following input data:
  - **Tc Multiplier**—Lets you define the time-of-concentration adjustment multiplier.
  - **Slope**—Lets you define the slope of the catchment section.
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
- **Kirpich TN**—This method requires the following input data:
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section.
  - **Tc Multiplier—**Lets you define the time-of-concentration adjustment multiplier.
- **Length and Velocity**—This method requires the following input data:

  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Velocity**—Lets you define the velocity of flow in the catchment section.

- **SCS Lag**—This method requires the following input data:
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **CN**—Lets you define the SCS runoff curve number of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section of the catchment section.

- **TR-55 Sheet Flow**—This number represents the sheet flow time computed for each column of sheet flow data. This method requires the following input data:
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Manning's n**—Lets you enter the Manning's roughness value of the catchment section.
  - **Slope**—Lets you define the slope of the catchment section.
  - **2 yr. 24 hr. Depth**—Depth of 2 year 24 hour storm.

- **TR-55 Shallow Conc.**—This number represents the sheet flow time computed for each column of shallow concentrated flow data. This method requires the following input data:
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Is Paved**—Lets you specify whether the catchment section is paved or unpaved.
  - **Slope**—Lets you define the slope of the catchment section.

- **TR-55 Channel Flow**—This number represents the channel flow time computed for each column of channel flow data. This method requires the following input data:
  - **Flow Area**—Lets you define the flow area of the catchment section.
  - **Hydraulic Length**—Lets you define the flow length of the catchment section.
  - **Manning's n**—Lets you enter the Manning's roughness value of the catchment section.

– **Slope**—Lets you define the slope of the catchment section.

– **Wetted Perimeter**—Lets you define the wetted perimeter of the catchment section.

Defining the Geometry of a Catchment or a Pond

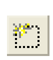
You define the geometry of a polygonal element, such as a catchment or a pond, by entering the location and angle of bends for the selected element. You enter X vs. Y points that plot the shape of the polygon that represents the element in the Polygon Vertices dialog box .

To define the geometry of a catchment or a pond:

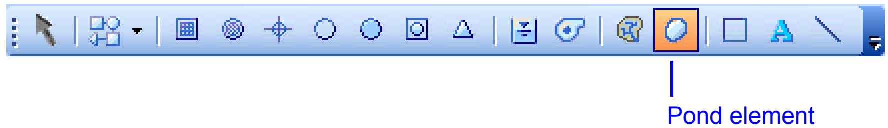
1. Click a catchment or pond in your model to display the Property Editor, or right-click a catchment or pond and select **Properties** from the shortcut menu.
2. In the Geometry section of the Property Editor, click the **Ellipses (...)** button next to the Geometry field.
3. In the Polygon Vertices dialog box, click the **New** button to add a new row to the table.
4. Type values for X and Y points for each row in the table.
5. To remove rows from the table, click the **Delete** button.
6. Click **OK**.

Polygon Vertices Dialog Box

This dialog box lets you define *X* vs. *Y* points that plot the shape of the polygon that represents the selected element. The dialog box contains the *X* vs. *Y* table that allows you to define any number of points and the following buttons:

	**New**	Creates a new row in the table.
	**Delete**	Deletes the currently highlighted row from the table.

Ponds



When you click the pond element on the Layout toolbar, your mouse cursor changes into a pond element symbol. Clicking in the drawing pane while this tool is active causes a pond element to be placed at the location of the mouse cursor.

Note: If there are multiple discharges locations serving a pond then they must all be modeled in the same manner; that is, they must either all be modeled with outlet control structures, or all modeled without outlet control structures.

Physical Characteristics of Ponds

There are categories of storage elements that can be modeled in Bentley SewerGEMS V8*i*:

- Outdoor ponds with a free surface
- Large underground pipes built solely to store water during storms
- Wet wells at pumping stations

The first two are modeled as “ponds” in Bentley SewerGEMS V8*i*. Any of these elements may be used as the suction side of a pump.

The following pond volume options are available:

- Elevation vs. Area
- Elevation vs. Volume
- Underground Pipe Volume
- Functional

Not all of the physical properties are needed for each method of describing pond (storage element) dimensions.

The type of attribute needed for each type of data entry is summarized in the table below:

Table 6-1: Required Pond Attributes

Attribute	Elevation -Volume Curve	Elevation -Area Curve	Volume Function	Pipe Volume
Volume Type	Required	Required	Required	Required
Elevation (Invert)	Required	Required	Required	Required
Depth (Maximum)	Required	Required	Required	
Elevation-Area Curve		Required		
Elevation-Volume Curve	Required			
Percent Void Space (%)	Required	Required		
Number of Barrels				Required
Length				Required
Invert (Start)				Required
Invert (Stop)				Required
Pipe Diameter				
Pond Coefficient A			Required	
Pond Exponent B			Required	
Pond Constant C			Required	

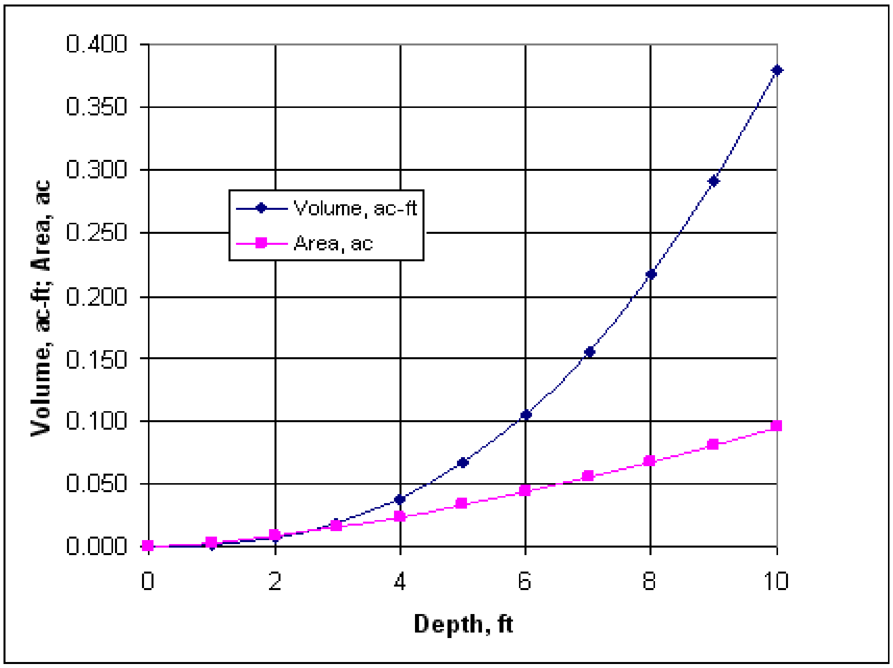
For more information on the physical characteristics of ponds, see the following help topics:

- "Outdoor Ponds" on page 6-246
- "Elevation vs. Area" on page 6-246

- "Elevation vs. Volume" on page 6-247
- "Percent Void Space (%)" on page 6-248
- "Pipe Volumes" on page 6-248
- "Functional (Equation)" on page 6-248

Outdoor Ponds

The physical size of outdoor graded ponds is usually described using a depth vs. area or depth vs. volume curve. A typical set of curves is shown below:



Elevation vs. Area

This approach is common for outdoor ponds. The Elevation vs. Area table represents the grading plan contour information for the pond. The area column represents the water surface area with respect to the corresponding water surface elevation on the same row in the table. Bentley SewerGEMS V8*i* will calculate the cumulative volume at each given elevation, based on the given Elevation vs. Area data.

Table 6-2: Pond Elevation vs. Area

Elevation (ft)	Area (ac)	Percent Void Space (%)
0	0	100
1	0.003	100
2	0.008	100
3	0.016	100
4	0.024	100
5	0.034	100
6	0.044	100
7	0.056	100
8	0.068	100
9	0.081	100
10	0.095	100

Elevation vs. Volume

This feature gives the user complete control over how to calculate pond volumes. The Elevation vs. Volume table is typically used in situations where the standard elevation vs. area or pipe volume calculations do not apply and the user desires to perform custom calculations and enter the results. The volume column represents the water volume being stored in the pond at the corresponding water surface elevation on the same row in the table.

Table 6-3: Pond Depth vs. Volume

Depth (ft)	Volume (ac-ft)	Percent Void Space (%)
0	0.000	100
1	0.001	100
2	0.007	100
3	0.019	100

Table 6-3: Pond Depth vs. Volume

4	0.038	100
5	0.067	100
6	0.106	100
7	0.156	100
8	0.217	100
9	0.292	100
10	0.379	100

Percent Void Space (%)

This option is available for both Elevation vs. Area and Elevation vs. Volume tables. The default value is 100%. The percent void represents the amount of real storage available at each elevation. For open ponds this value is 100%. This option is useful for situation where the pond storage area is filled with impervious material, such as stone or gravel. The percent void space represents the open pore areas that can store water.

Example: For 50% void space, only ½ of the calculated (or specified) total volume would be available as water storage.

Pipe Volumes

Another storage option models large, buried, sloped pipes. These are described to the model using the diameter, length and number of pipe barrels (assuming parallel buried pipes are of the same dimension). The stop invert elevation of the incoming pipe and the start invert elevation of the outgoing pipe must also be specified in order to determine the slope of the pipe, which will affect the volume calculation at each water surface elevation.

Functional (Equation)

Another approach for calculating pond volume is to enter the coefficients of the following polynomial equation:

$$Area = Coeff * DepthExp + Constant$$

Where:

- **Area** = Surface Area at given depth

- **Coeff** = User input value which is derived from existing volume data
- **Depth** = Distance from the invert of the pond
- **Exp** = User input value which is derived from existing volume data
- **Constant** = The area at the bottom of the pond and is a user input value

Note: The function parameters are based on depths in feet and areas in square feet.

Adding Elevation vs. Area Data to a Pond

You can add an elevation-area curve to a pond. The area-elevation points defined in the curve are used to calculate pond volumes. You define elevation-area curves in the Elevation-Area Curve dialog box.

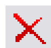
To add an Elevation vs. Area curve to a pond:

1. Click a pond in your model to display the Property Editor, or right-click a pond and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, select **Elevation-Area Curve** in the Volume Type field. The Elevation-Area Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Elevation-Area Curve field.
4. In the Elevation-Area Curve dialog box, each row in the table represents a point on the Elevation-Area curve. Type values for Elevation, Area, and Percent Void Space for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Elevation-Area Curve Dialog Box

This dialog box allows you to define elevation-area curves.

The dialog box contains the elevation-area table and the following buttons:

	**New**	Creates a new row in the elevation-area table.
	**Delete**	Deletes the currently highlighted row from the elevation-area table
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the area-elevation curve defined by the points in the table.

The table contains the following columns:

Column	Description
Elevation	Lets you enter the elevation data for the pond.
Area	Lets you enter the plan area at that elevation.
Percent Void Space	Lets you enter the void space volume data for the pond. Void space is used on any volumes option to adjust the effective storage volume for rock-filled or other porous media filled basins or vaults. Set this to 100% if there are no rocks or fill to reduce the available volume.

Adding Elevation vs. Volume Data to a Pond

You can add an elevation-volume curve to a pond. The area-volume points defined in the curve are used to calculate pond volumes. The elevation-volume points defined in the table are used to calculate total pond volume. It is important that this storage relation be single-valued. This means that any volumetric quantity occurs only once in the table only, as Bentley SewerGEMS V8*i* interpolates linearly between any two values in the table. You define elevation-area curves in the Elevation-Volume Curve dialog box.

To add an Elevation vs. Volume curve to a pond:

1. Click a pond in your model to display the Property Editor, or right-click a pond and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, select **Elevation-Volume Curve** in the Volume Type field. The Elevation-Volume Curve field becomes available.
3. Click the **Ellipses (...)** button next to the Elevation-Volume Curve field.
4. In the Elevation-Volume Curve dialog box, each row in the table represents a point on the Elevation-Volume curve. Type values for Elevation, Volume, and Percent Void Space for each row. Click the **New** button to add a row or press the **Tab** key to advance to the next field in the table.
5. Perform the following optional steps:
   – To delete a row from the table, select the row then click **Delete**.
   – To view a report on the curve, click **Report**.
   – To view a plot of the curve, click **Graph**.
6. Click **OK** to close the dialog box and save your curve data in the Property Editor.

Elevation-Volume Curve Dialog Box

This dialog box allows you to define elevation-volume tables. The dialog box contains the elevation-volume table and the following buttons:

	**New**	Creates a new row in the elevation-volume table.
	**Delete**	Deletes the currently highlighted row from the elevation-volume table
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the area-volume curve defined by the points in the table.

The table contains the following columns:

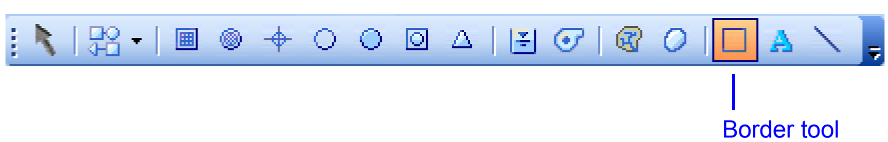
Column	Description
Elevation	Lets you enter the cumulative volume from the bottom of the pond to that elevation.
Volume	Lets you enter the volume data for the pond.
Void Space	Lets you enter the void space volume data for the pond. Void space is used on any volumes option to adjust the effective storage volume for rock-filled or other porous media filled basins or vaults. Set this to 100% if there are no rocks or fill to reduce the available volume.

Other Tools

Although Bentley SewerGEMS V8*i* is primarily a modeling application, some additional drafting tools can be helpful for intermediate calculations and drawing annotation. AutoCAD, of course, provides a tremendous number of drafting tools. Bentley SewerGEMS V8*i* provides the following tools:

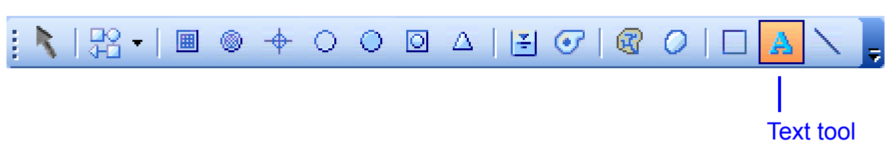
- Border tool
- Text tool
- Line tool

Border Tool



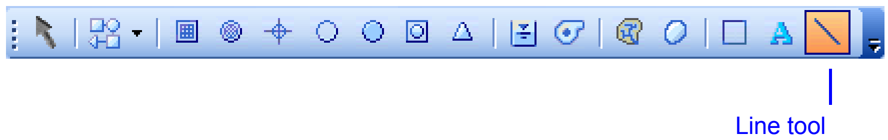
The Border tool lets you add rectangles to the drawing pane.

Text Tool



The Text tool lets you add text to the drawing pane.

Line Tool



The Line tool lets you add lines and polylines (multisegmented lines) to the drawing pane.

Adding Elements to Your Model

Bentley SewerGEMS V8*i* provides several ways to add elements to your model. They include:

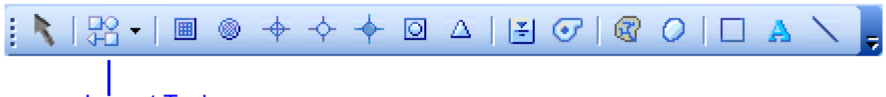
- Adding individual elements
- Adding elements using the layout tool
- Replacing an element with another element

To add individual elements to your model:

1. Click an element symbol on the Layout toolbar. The mouse cursor changes to the element symbol you selected.
2. Click in the drawing pane to add the element to your model.
3. Click again to add another element of the same type to your model.
4. To add a different element, click on the desired element symbol in the Layout toolbar, then click in the drawing pane.
5. To stop adding an element, right-click in the drawing pane to display a shortcut menu, then click **Done**.

To add elements using the layout tool:

The layout tool lets you quickly add new elements to your model without having to select a new element button on the Layout toolbar. When the layout tool is active, you can right-click in the drawing pane to select different elements and pipes to add to the model.



1. Click the Layout tool on the Layout toolbar. A shortcut menu appears.
2. Click the type of pipe you want to use to connect your elements in the model.
3. Right-click in the drawing pane, then select the type of element you want to add from the shortcut menu. The shortcut menu displays only those element types that are compatible with your pipe selection.
4. Click in the drawing pane to add the element.
5. Click again to add another of the same element type. The elements you add will automatically be connected by the type of pipe you selected earlier.
6. To change the type of pipe, right-click and select a different type from the shortcut menu.
7. To change the element, right-click and select a different element from the shortcut menu.
8. To stop adding elements using the Layout tool, right-click anywhere in the drawing pane and click **Done**.

Note: In AutoCAD, you must hold down the mouse button to keep the submenu open while selecting an element from the layout toolbar. Alternate layout methods include using the right-click menu to select elements or using the command line.

Modeling Curved Pipes

You can model curved pipes in SewerGEMS V8*i* by using the Bend command, which is available by right-clicking in the Drawing Pane when placing a link element.

Bentley SewerGEMS V8*i* does not account for any additional head loss due to the curvature because in most cases the increased head loss is negligible. If you feel the extra head loss is significant, it is possible to increase the Manning's n value to account for such losses.

To model a curved pipe:

1. Select the desired link element using the **Layout** button on the Layout toolbar.
2. Place the first segment of the curved pipe in your model, then right click and select **Bend** from the shortcut menu.
3. Repeat Step 2 for each segment in the curved pipe. Be sure to insert bends to clearly show the curved alignment.
4. When the curved pipe is complete, right click and select the next downstream element (for a conduit, this is usually a manhole).

Connecting Elements

When building your model, you must consider these rules of connectivity:

- A network needs at least one outfall or a pond to end the network. A lone outfall cannot be a boundary element type unless it’s draining into a pond.
- Cross section nodes need at least one channel connected to it and a channel needs at least one cross section.
- Gutters cannot be the only link exiting a catch basin, or the catch basin is considered hydraulically disconnected.

Table 6-4: Element Connectivity

Element	Permissible Upstream Elements	Permissible Downstream Elements
Catchment	None	Catch basin, manhole, cross section, junction chamber, outfall, wet well, pond
Manhole	Via a gutter: None Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure, catchment Via a pressure pipe: Pressure junction, pump	Via a gutter: None Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction, pump
Catch basin	Via a gutter: Catch basin, cross section, outfall Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure, catchment Via a pressure pipe: Pressure junction, pump	Via a gutter: Catch basin, cross section, outfall Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction, pump
Cross section	Via a gutter: Catch basin Via a channel: Manhole, catch basin, cross section, wet well, outfall, pond outlet structure Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure, catchment Via a pressure pipe: Pressure junction, pump	Via a gutter: Catch basin Via a channel: Manhole, catch basin, cross section, wet well, outfall, pond outlet structure Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction, pump

Table 6-4: Element Connectivity

Element	Permissible Upstream Elements	Permissible Downstream Elements
Junction chamber	Via a gutter: None Via a channel: None Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure, catchment Via a pressure pipe: Pressure junction, pump	Via a gutter: None Via a channel: None Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction, pump
Pressure junction	Via a gutter: None Via a channel: None Via a conduit: None Via a pressure pipe: Manhole, catch basin, cross section, junction chamber, pressure junction, wet well, pump, outfall, pond outlet structure	Via a gutter: None Via a channel: None Via a conduit: None Via a pressure pipe: Manhole, catch basin, cross section, junction chamber, pressure junction, wet well, pump, outfall, pond outlet structure
Wet well	Via a gutter: None Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure, catchment Via a pressure pipe: Pressure junction, pump	Via a gutter: None Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction, pump

Table 6-4: Element Connectivity

Element	Permissible Upstream Elements	Permissible Downstream Elements
Pump	Via a gutter: None Via a channel: None Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Manhole, catch basin, cross section, junction chamber, pressure junction, wet well, pump, outfall, pond outlet structure	Via a gutter: None Via a channel: None Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Manhole, catch basin, cross section, junction chamber, pressure junction, wet well, pump, outfall, pond outlet structure
Outfall	Via a gutter: Catch basin Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, pond outlet structure, catchment Via a pressure pipe: Pressure junction, pump	Via a gutter: Catch basin Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, pond outlet structure Via a pressure pipe: Pressure junction, pump When the Boundary Condition Type is set to Boundary Element, the flow coming into the outfall can be discharged to a pond, cross section, manhole, or catch basin
Pond outlet structure	Via a gutter: None Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction	Via a gutter: None Via a channel: Cross section Via a conduit: Manhole, catch basin, cross section, junction chamber, wet well, pump, outfall, pond outlet structure Via a pressure pipe: Pressure junction
Pond	Outfall, catchment	Pond outlet structures, pump. Only outfalls can drain to a pond. The only way to drain from a pond is via a pond outlet structure.

When To Use a Conduit vs. a Channel vs. a Gutter

Gutters are used in Bentley SewerGEMS V8*i* only to model the water which exceeds the capacity of in catch basin inlet and must flow through a surface gutter to the next catch basin. A Bentley SewerGEMS V8*i* gutter can only receive water from a catch basin.

A conduit can refer to any prismatic channel or pipe that conveys flow. The cross section of a conduit must remain constant from one end to the next.

A channel refers to a channel that changes geometry from the upstream cross section to the downstream cross section. Channels can be used to model natural streams or swales which are not prismatic in cross section. Channels must have a cross section element at either end and properties are interpolated along the channel.

A uniform trapezoidal channel can either be modeled as a conduit with the shape defined in a conduit element or a channel with the shape defined as a property of the cross section elements at each end.

What Is A Virtual Conduit?

Bentley SewerGEMS V8*i*'s network element layout is based on maintaining spatial fidelity between an element's model representation and it's actual physical location in the field. Similar to a pond outlet, a pump station is modeled as an element attached to a storage node or a manhole node and linked downstream to a force main pipe, and multiple pumps with different characteristics can be added into the element.

Virtual pipes are a special compatibility element included in Bentley SewerGEMS V8*i* to help modelers achieve fidelity between the Bentley SewerGEMS V8*i* model and other storm modeling solutions that model pumps and control structures such as weirs, orifices, and rating tables as network links. SWMM 5 models pumps, weirs, orifices, and rating tables as links. Virtual links are a useful means for mapping such elements into Bentley SewerGEMS V8*i* in a literal way. So, if you import a SWMM model containing pumps and/or control structures into Bentley SewerGEMS V8*i*, virtual links will be imported to provide equivalent hydraulic representations.

The virtual conduit has no physical properties such a length and diameter. It has no hydraulic effect in the model other than define hydraulic connectivity and preserve continuity relationships.

For example, in SWMM, it is possible to have the discharge side of a pump connected to a node thousands of feet away with no consideration of the interconnecting force main. However in Bentley SewerGEMS V8*i*, to model that connection, a virtual pipe, which shows up in the drawing pane as a dashed line, is used to show the connectivity.

For more information on virtual conduits, see "Virtual Link Types" on page 14-742.

How Do I Get Rainfall from a Catchment Into the Rest of My Model?

To get rainfall to move from a catchment into the rest of your model, you must specify an Outflow Node.

To set the outflow node for a catchment:

1. If the Property Editor is not open, click **View > Properties** (F4) to open it.
2. Click the catchment for which you want to set an outflow node.
3. In the Catchment section of the Property Editor, click the Outflow Node field to enable the selection drop-down.
4. Click **Select** if you want to select the outflow node from the model, or select the outflow node from the drop-down list.

Connecting a Pump to a Wet Well

Bentley SewerGEMS V8*i* assumes that either the pump is a submersible pump in the wet well or the pump is connected to the wet well through a short piece of pressure pipe with negligible head loss.

Do not connect a conduit or channel between the wet well and pump.

Pumps can also use ponds or manholes as suction elements. If a pressure pipe is used as the suction element, it is assumed that the distance is so short that the head loss is negligible. If head loss is significant in determining pump flow, then add pipe length or minor loss to the discharge pipe.

To specify the relationship between a wet well and a pump:

1. Double-click the pump in your model to open the Property Editor for the pump.
2. Select **Select Suction Element** as the Suction Side Node.

   SewerGEMS V8*i* prompts you to select the suction side node or link from your model (which can be a wet well, pond, pressure pipe, or manhole).
3. Select the appropriate node in your model. A dashed line appears to connect the two elements. If a manhole is used as a suction line, it cannot be the most down-stream end of a gravity network.

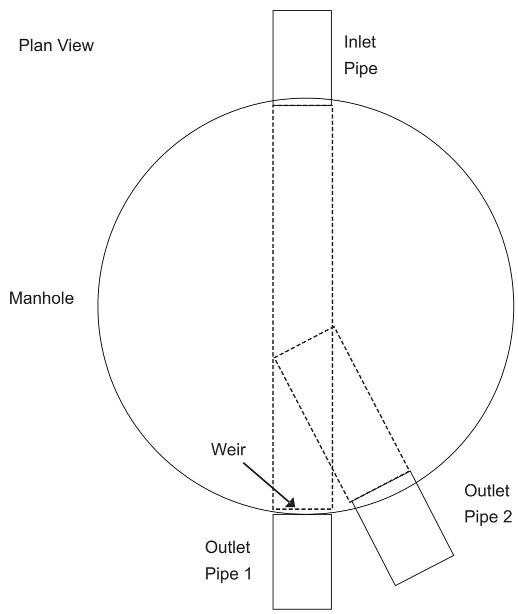
How Do I Model Weirs in Conduits?

Sharp crested weirs can be placed as control structures in conduits. Other control structures include orifices, functional structures and depth discharge curves.

To insert weirs into the start or stop ends of a conduit:

1. Set **Has Start Control Structure?** or **Has Stop Control Structure?** to **True** in the Physical:Has Control Structure section of the Property Editor for the conduit.
2. Click the **Ellipse** (…) button that appears next to the Start Control Structure or Stop Control Structure field. This opens the Conduit Control Structure dialog box.
3. Select **New > Weir**.

Weirs can be placed in any shape of conduit but the weir structure itself is treated as being in a rectangular section. The weir length is the distance across this section measured perpendicular to the flow (for all except side weirs).



Manipulating Elements

You can manipulate elements in your model in any one of the following ways:

- Select elements—manually select individual elements, manually select multiple elements, select all elements, or select all elements of a single element type
- Move elements
- Delete elements
- Split pipes

To manually select an element:

Click the element. Selected elements appear in red.

> **Note:** You can change the selection color in the Options dialog box, which is accessible by selecting Tools > Options.

To manually select multiple elements:

Click the first element, then click additional elements while holding down **Shift** or **Ctrl**.

To select all elements:

To select all of the elements in your model, select **Edit** > **Select All**.

To select all elements of the same type:

To select all elements of the same type (for example, all junction chambers), select **Edit** > **Select by Element**, then click the desired element type.

All elements of the selected type appear in red, including connecting pipes.

To clear selected elements:

Click the **Select** tool then click any blank space in the drawing pane.

or

Click **Edit** > **Clear Selection**.

or

Press the **Esc** key.

You can also clear a selected element by clicking a different element.

To move an element in the model:

1. Click the Select tool on the Layout toolbar.
2. Select the element(s) you want to move, then drag it to its new location. Pipe connections move with the element.

To delete an element:

Select the element, then press **Delete**.

or

Select **Edit** > **Delete**.

Splitting Pipes

You may encounter a situation in which you need to add a new element in the middle of an existing pipe. For example, you may want to insert a new manhole to maintain maximum access hole spacing.

To split an existing pipe:

1. Select the desired element symbol on the Layout toolbar.
2. In the drawing pane, place the cursor over the pipe you want to split and click.
3. You are prompted to confirm that you want to split the pipe.
   – If you choose to split the pipe, the element will be inserted and two new pipes will be created with the same characteristics as the original pipe (lengths are split proportionally).
   – If you choose not to split the pipe, the new element will be placed on top of the pipe without connecting to anything.

If you accidentally split a pipe, this action can be undone by selecting **Edit** > **Undo**.

You can also split an existing pipe with an existing element:

To do this in the Stand-Alone version, drag the element into position along the pipe to be split, then right-click the node and select **Split <Pipe Label>** from the shortcut menu (where **<Pipe Label>** is the name of the pipe to be split).

To do this in the MicroStation version, drag the element into position along the pipe to be split. Hold down the Shift key, then right-click the node and select **Split <Pipe Label>** from the shortcut menu (where **<Pipe Label>** is the name of the pipe to be split).

Disconnecting and Reconnecting Pipes

In certain circumstances, you may wish to disconnect a pipe from a node without deleting and redrawing the pipe in question. For example, if the model was built from a database and the Establish By Spatial Data option was used to determine pipe connectivity, pipes may have been connected to the wrong nodes.

To disconnect and reconnect a pipe:

1. Right-click the pipe to be disconnected.
2. A context menu will appear. Two reconnect options will be displayed, one for each of the end nodes of the pipe. Select the node from which you want to disconnect the pipe.
3. A broken line will appear, joining the node from which the pipe is being disconnected and your mouse cursor. Hover the mouse cursor over the junction to which you would like to connect the pipe and click the left mouse button. The pipe will now be connected to this junction.

How Do I Model a Split in a Channel?

If you have a channel with a control structure and you're trying to model a split in that channel, you should put the control structure on the upstream end of the branching links rather than the downstream end of the main link.

Editing Element Attributes

You edit element properties in the Property Editor, one of the dockable managers in Bentley SewerGEMS V8*i*.

To edit element properties:

Double-click the element in the drawing pane. The Property Editor displays the attributes of the selected element.

or

Select the element whose properties you want to edit, then select **View > Properties** or click the Properties button on the Analysis toolbar.

Property Editor

The Property Editor is a contextual dialog box that changes depending on the status of other dialog boxes. For example, when a network element is highlighted in the drawing pane, the Property Editor displays the attributes and values associated with that element. When one of the manager dialog boxes is active, the Property Editor displays the properties pertaining to the currently highlighted manager element.

Attributes displayed in the Property Editor are grouped into categories. An expanded category can be collapsed by clicking the plus (+) button next to the category heading. A collapsed category can be expanded by clicking the minus (-) button next to the category heading.

For the most efficient data entry in Text Box style fields, instead of clicking on the Field, click on the label to the left of the field you want to edit, and start typing. Press Enter to commit the value, then use the Up/Down keyboard arrows to navigate to the next field you want to edit. You can then edit the field data without clicking the label first; when you are finished editing the field data, press the Enter key, and proceed to the next field using the arrow keys, and so on.

Find Element

The top section of the Property Editor contains the Find Element tool. The Find Element tool lets you:

- Quickly find a recently-created or added element in your model. The Element menu contains a list of the most recently-created and added elements. Click an element in the Element menu to center the drawing pane around that element and highlight it.
- Find an element in your model by typing the element label or ID in the Element menu then clicking the Find button or pressing Enter. The drawing pane centers around the highlighted element.
- Find all elements of a certain type by using an asterisk (*) as a wild-card character. For example, if you want to find all of the conduits in your model, you type **co*** (this is not case-sensitive) then click the Find button. The drawing pane centers around and highlights the first instance of a conduit in your model, and lists all conduits in your model in the Element menu. Once the Element menu is populated with a list of elements, you can use the Find Next and Find Previous buttons to quickly navigate to the next or previous element in the list.

Note: See the ”Using the Like Operator’ **topic for more information about wildcard symbols.**

The following controls are included:

	Element	Type an element label or ID in this field then click the Find button to quickly locate it in your model. The element selected in this menu will be centered in the drawing pane when the Zoom To command is initiated, at the magnification level specified by the Zoom Level menu. The drop-down menu lists recently-created or added elements, elements that are part of a selection set, and that are part of the results from a recent Find operation.
	**Find Previous**	This button allows you to find the previous element in the list of results from a recent Find operation.
	**Find**	Zooms the drawing pane view to the element typed or selected in the Element menu at the magnification level specified in the Zoom Level menu.
	**Find Next**	This button allows you to find the next element in the list of results from a recent Find operation.
	**Help**	Displays online help for the Property Editor.
	Zoom Level	Allows you to specify the magnification level at which elements are displayed in the drawing pane when the Zoom To command is initiated.
	**Alphabetic**	Displays the attribute fields in the Property Editor in alphabetical order.
	**Categorized**	Displays the attribute fields in the Property Editor in categories. This is the default.

Relabeling Elements

You can relabel elements from within the Property Editor.

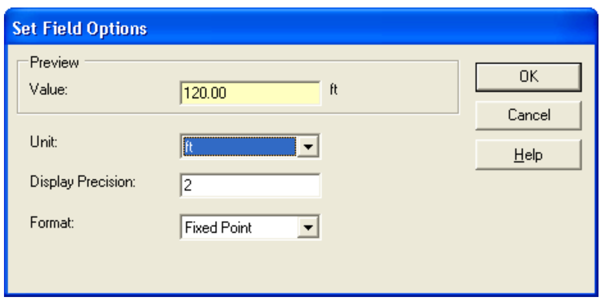
To relabel an element:

1. Select the element in the Drawing Pane then, if the Property Editor is not already displayed, select **View** > **Properties**.
2. In the General section of the Property Editor, click in the Label field, then type a new label for the element.

Set Field Options Dialog Box

The Set Field Options dialog box lets you set the units for a specific attribute without affecting the units used by other attributes or globally.

To use the Set Field Options dialog box, right-click any numerical field that has units, then select **Units and Formatting**.



Value	Displays the value of the currently selected item.
Unit	Displays the type of measurement. To change the unit, select the unit you want to use from the drop-down list. This option also lets you use both U.S. customary and S.I. units in the same worksheet.
Display Precision	Sets the rounding of numbers and number of digits displayed after the decimal point. Enter a negative number for rounding to the nearest power of 10: (-1) rounds to 10, (-2) rounds to 100, (-3) rounds to 1000, and so on. Enter a number from 0 to 15 to indicate the number of digits after the decimal point.

Format	Lets you select the display format used by the current field. Choices include: • **Scientific**—Converts the entered value to a string of the form "-d.ddd...E+ddd" or "-d.ddd...e+ddd", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. • **Fixed Point**—Abides by the display precision setting, and automatically enters zeros after the decimal place to do so. With a display precision of 3, an entered value of 3.5 displays as 3.500. • **General**—Truncates any zeros after the decimal point, regardless of the display precision value. With a display precision of 3, the value that would appear as 5.200 in Fixed Point format displays as 5.2 when using General format. The number is also rounded. So, an entered value of 5.35 displays as 5.4 regardless of the display precision. • **Number**—Converts the entered value to a string of the form "-d,ddd,ddd.ddd...", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. Thousand separators are inserted between each group of three digits to the left of the decimal point.

What Length is Used for Conduits, Channels, and Gutters When I Don't Enter a User-defined Length?

If you do not enter a user-defined length in the attributes for conduits, channels, and gutters, the length used in Bentley SewerGEMS V8*i* is the plan view distance between the coordinates at each end of the link element. This length is used as the actual length in hydraulic calculations. However, as the slope increases, the difference between the plan length and the actual length also increases as shown below.

The table below shows the difference between the actual and plan length as a function of slope. Note that for most reasonable slopes, the difference between the actual and plan view length is less than one percent. (100% slope is 1:1 slope.) As the slope approaches vertical, you must enter the actual length.

Table 6-5: Actual and Plan Length as a Function of Slope

Slope, % *	Actual/Plan Length
0	1.000
10	1.005
20	1.020
30	1.044

* The model's generalized friction formulation is only valid for slopes less than 10%.

If you are not satisfied with the plan view length, you can enter a user-defined length, which you can determine using the following equation:

$$Actual = \sqrt{Plan^2(1 + (\%slope/100)^2)}$$

What is the Difference Between a User Defined Unit Hydrograph and a Hydrograph Entered in the Inflow Collection Editor?

Within Bentley SewerGEMS V8*i*, you can enter a hydrograph (in flow units) at any node element (e.g. catch basin, manhole, catchment, cross section, wet well). You can also enter a unit hydrograph, but only at a catchment node.

To enter a hydrograph (in flow units) using the Inflow Collection Editor:

1. Select the element in your model.
2. In the Property Editor, click the **Ellipsis** button (...) in the Inflow Collection field.
3. In the Inflow Collection dialog box, click the **New** button and select Hydrograph Load.
4. Enter values into the table of flow vs. time, then click **OK**.

To enter a unit hydrograph at a catchment:

1. Select a catchment in your model.
2. In the Property Editor, select **Unit Hydrograph** as the Runoff Method.
3. In the Property Editor, select **Generic Unit Hydrograph** as the Unit Hydrograph Method.

4. Click the **Ellipsis** button (...) in the Unit Hydrograph Data field.
5. In the Unit Hydrograph Data dialog box, enter values into the table of flows (for a unit of rainfall) vs. time, then click **OK**.

Other hydrographs used by Bentley SewerGEMS V8*i* are calculated within Bentley SewerGEMS V8*i*.

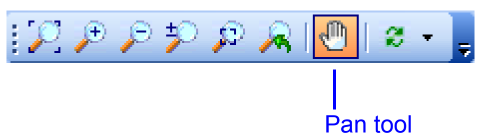
Changing the Drawing View

You change the drawing view of your model by using the pan tool or one of the zoom tools:

- "Panning" on page 6-270
- "Zooming" on page 6-271

Panning

You can change the position of your model in the drawing pane by using the Pan tool.



To use the Pan tool:

1. Click the **Pan** button on the Tools toolbar.

   The mouse cursor changes to the Pan icon.
2. Click anywhere in the drawing, hold down the mouse button and move the mouse to reposition the current view.

or

If your mouse is equipped with a mousewheel, you can pan by simply holding down the mousewheel and moving the mouse to reposition the current view.

or

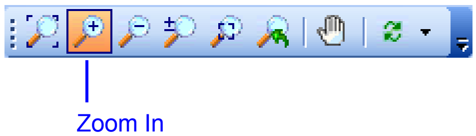
Select **View > Pan**, then click anywhere in the drawing, hold down the mouse button and move the mouse to reposition the current view

Zooming

You can enlarge or reduce your model in the drawing pane using one of the following zoom tools:

Zoom In and Out

The simple Zoom In and Zoom Out commands allow you to increase or decrease, respectively, the zoom level of the current view by one step per mouse click.



To use Zoom In or Zoom Out, click the desired button on the Tools toolbar, or select **View** > **Zoom** > **Zoom In** or **View** > **Zoom** > **Zoom In**.

If your mouse is equipped with a mousewheel, you zoom in or out by simply moving the mousewheel up or down respectively.

Zoom Window

The Zoom Window command lets you zoom in on an area of your model defined by a window that you draw in the drawing pane.

To use Zoom Window, select **View** > **Zoom** > **Zoom Window** button, then click and drag the mouse inside the drawing pane to draw a rectangle. The area of your model inside the rectangle will appear enlarged.

Note: If you use the Zoom Window command frequently, you might find it more convenient to add them to the Tools toolbar. See "Adding and Removing Toolbar Buttons" on page 2-38 **for more information.**

Zoom Extents

The Zoom Extents command automatically sets the zoom level such that the entire model is displayed in the drawing pane.

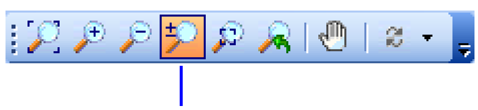
To use Zoom Extents, click the Zoom Extents button on the Tools toolbar. The entire model is displayed in the drawing pane.

or

Select **View** > **Zoom** > **Zoom Extents**.

Zoom Realtime

The Zoom Realtime command lets you dynamically scale up and down the zoom level. The zoom level is defined by the magnitude of mouse movement while the tool is active.



Zoom Previous and Zoom Next

Zoom Previous returns the zoom level to the most recent previous setting. To use Zoom Previous, click the Zoom Previous button on the Tools toolbar.

or

Select **View** > **Zoom** > **Zoom Previous**.

Zoom Next returns the zoom level to the setting that was active before a Zoom Previous command was executed. To use Zoom Previous, click **View** > **Zoom** > **Zoom Next**.

Note: If you use the Zoom Next command frequently, you might find it more convenient to add them to the Tools toolbar. See “Adding and Removing Toolbar Buttons” on page 2-38 **for more information.**

Using the Zoom Center Command

The Zoom Center command lets you enter drawing coordinates that will be centered in the drawing pane.

To use the Zoom Center command:

1. Select **View > Zoom > Zoom Center**. The Zoom Center dialog box appears.
2. Enter the X and Y coordinates.
3. Select the zoom factor from the Zoom drop-down, then click **OK**.

Zoom Center Dialog Box

The Zoom Center dialog box contains the following options:

X	Defines the X coordinate of the point at which the model will be centered.
Y	Defines the Y coordinate of the point at which the model will be centered.
Zoom Factor	Defines the zoom level that will be applied when the zoom center command is initiated. Available zoom levels are listed in percentages of 25, 50, 75, 100, 125, 150, 200 and 400.

Using Selection Sets

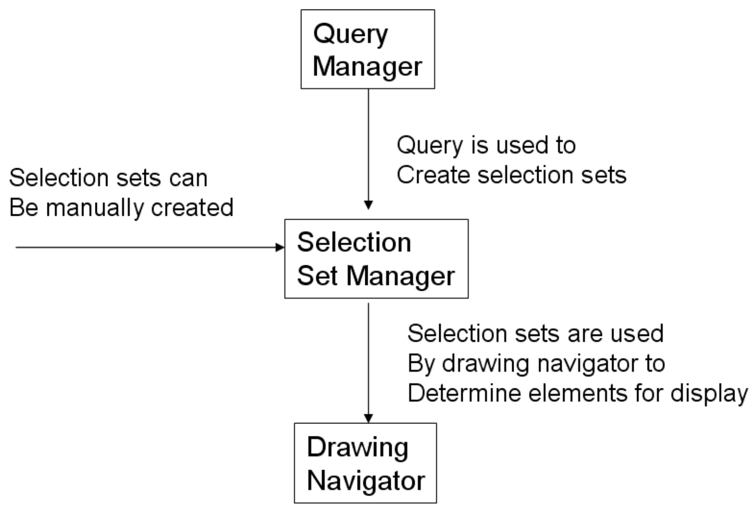
Selection sets are user-defined groups of network elements. They allow you to predefine a group of network elements that you want to manipulate together. You manage selection sets in the ”Selection Sets Manager’.

Bentley SewerGEMS V8*i* contains powerful features that let you view or analyze subsets of your entire model. You can find these elements using the Network Navigator (see “Using the Network Navigator” on page 6-281). The Network Navigator lets you choose a selection set, then view the list of elements in the selection set or find individual elements from the selection set in the drawing.

In order to use the Network Navigator, you must first create a selection set. There are two ways to create a selection set:

- From a selection of elements—You create a new selection set in the Selection Sets Manager, then use your mouse to select the desired elements in the drawing pane.
- From a query—Create a query in the Queries Manager, then use the named query to find elements in your model and place them in the selection set.

The following illustration shows the overall process.



You can perform the following operations with selection sets:

- "Viewing Elements in a Selection Sets" on page 6-276
- "Creating a Selection Set from a Selection" on page 6-277
- "Creating a Selection Set from a Query" on page 6-277
- "Adding Elements to a Selection Set" on page 6-279
- "Removing Elements from a Selection Set" on page 6-280

Selection Sets Manager

The Selection Sets Manager allows you to create, edit, and navigate to selection sets. The Selection Sets Manager consists of a toolbar and a list pane, which displays all of the selection sets that are associated with the current project.

The toolbar contains the following buttons:

	**New**	Contains the following commands: • **Create from Selection**—Creates a new static selection set from elements you select in your model. • **Create from Query**—Creates a new dynamic selection set from existing queries.
	**Delete**	Deletes the selection set that is currently highlighted in the list pane. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.

	**Edit**	• When a selection-based selection set is highlighted when you click this button, opens the Selection Set Element Removal dialog box, which lets you edit the selection set. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane. • When a query-based selection set is highlighted when you click this button, opens the Selection By Query dialog box, which lets you add or remove queries from the selection set. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.
	**Rename**	Lets you rename the selection set that is currently highlighted in the list pane. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.
	**Select In Drawing**	Lets you quickly select all the elements in the drawing pane that are part of the currently highlighted selection set. Once you have selected the elements in a selection set using Select In Drawing, you can delete them all at once or create a report on them. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.
	**Help**	Displays online help for the Selection Sets Manager.

You can view the properties of a selection in the Property Editor by right-clicking the selection set in the list pane and selecting **Properties** from the shortcut menu.

Viewing Elements in a Selection Sets

You use the Network Navigator to view the elements that make up a selection set.

To view the elements that make up a selection set:

1. Open the Network Navigator by selecting View > Network Navigator or clicking the Network Navigator button on the View toolbar.
2. Select a selection set from the Selection Set drop-down list. The elements in the selection set appear in the Network Navigator.

Tip: You can double-click an element in the Network Navigator to select and center it in the Drawing Pane.

Creating a Selection Set from a Selection

You create a new selection set by selecting elements in your model.

To create a new selection set from a selection:

1. Select all of the elements you want in the selection set by either drawing a selection box around them or by holding down the **Ctrl** key while clicking each one in turn.
2. When all of the desired elements are highlighted, right-click and select **Create Selection Set**.
3. Type the name of the selection set you want to create, then click **OK** to create the new selection set. Click **Cancel** to close the dialog box without creating the selection set.
4. Alternatively, you can open the Selection Set Manager and click the **New** button and select **Create from Selection**. Bentley SewerGEMS V8*i* prompts you to select one or more elements.

Create Selection Set Dialog Box

This dialog box appears when you create a new selection set. It contains the following field:

New selection set name	Lets you type the name of the new selection set.

Creating a Selection Set from a Query

You create a dynamic selection set by creating a query-based selection set. A query-based selection set can contain one or more queries, which are valid SQL expressions.

To create a new selection set from a query:

1. In the Selection Sets Manager, click the **New** button and select **Create from Query**. The Selection by Query dialog box appears.
2. Available queries appear in the list pane on the left; queries selected to be part of the selection set appear in the list pane on the right. Use the arrow buttons in the middle of the dialog to add one or all queries from the Available Queries list to the Selected Queries list, or to remove queries from the Selected list.
   – You can also double-click queries on either side of the dialog box to add them to or remove them from the selection set.

Selection by Query Dialog Box

The Selection by Query dialog box lets you create selection sets from available queries. The dialog box contains the following controls:

Available Queries	Contains all the queries that are available for your selection set. The Available Columns list is located on the left side of the dialog box.
Selected Queries	Contains queries that are part of the selection set. To add queries to the Selected Queries list, select one or more queries in the Available Queries list, then click the **Add** button [>].

Query Manipulation Buttons

Lets you select or clear queries to be used in the selection set:

- [>] Adds the selected items from the Available Queries list to the Selected Queries list.
- [>>] Adds all of the items in the Available Queries list to the Selected Queries list.
- [<] Removes the selected items from the Selected Queries list.
- [<<] Removes all items from the Selected Queries list.

Note: You can select multiple queries in the Available Queries list by holding down the Shift key or the Control key while clicking with the mouse. Holding down the Shift key provides group selection behavior. Holding down the Control key provides single element selection behavior.

Adding Elements to a Selection Set

You can add a single or multiple elements to a static selection set.

To add an element to a static selection set:

1. Right-click the element to be added, then select **Add to Selection Set** from the shortcut menu.
2. In the Add to Selection Set dialog box, select the selection set to which you want to add the element.
3. Click **OK** to close the dialog box and add the element to the selected selection set. Click **Cancel** to close the dialog box without creating the selection set.

To add a group of elements to a static selection set all at once:

1. Select all of the elements to be added by either drawing a selection box around them, or by holding down the **Ctrl** key while clicking each one in turn.
2. When all of the desired elements are highlighted, right-click and select **Add to Selection Set**.

3. In the Add to Selection Set dialog box, select the selection set to which you want to add the element.
4. Click **OK** to close the dialog box and add the element to the selected selection set. Click **Cancel** to close the dialog box without creating the selection set.

Add To Selection Set Dialog Box

This dialog box appears when you select the Add to Selection Set command. It contains the following field:

Add to:	Drop-down menu that lets you select the selection set to which the currently highlighted element or elements will be added.

Removing Elements from a Selection Set

You can easily remove elements from a static selection set in the Selection Set Element Removal dialog box.

To remove an element from a static selection set:

1. Display the Selection Sets Manager by selecting **View > Selection Sets** or clicking the **Selection Sets** button on the View toolbar.
2. In the Selection Sets Manager, select the desired selection set then click the **Edit** button.
3. In the Selection Set Element Removal dialog box, find the element you want to remove in the table. Select the element label or the entire table row, then click the **Delete** button.
4. Click **OK**.

Selection Set Element Removal Dialog Box

This dialog appears when you click the edit button from the Selection Set Manager. It allows you to remove elements from the selection set that is highlighted in the **Selection Sets Manager** when the **Edit** button is clicked.

Performing Group-Level Operations on Selection Sets

SewerGEMS V8*i* lets you perform group-level deletions on elements in a selection set using the Select In Drawing button in the Selection Sets Manager.

Note: While it is not possible to directly edit groups of elements in a selection set, you can use the Next button in the Network Navigator to quickly navigate through each element in the selection set and edit its properties in the Property Editor.

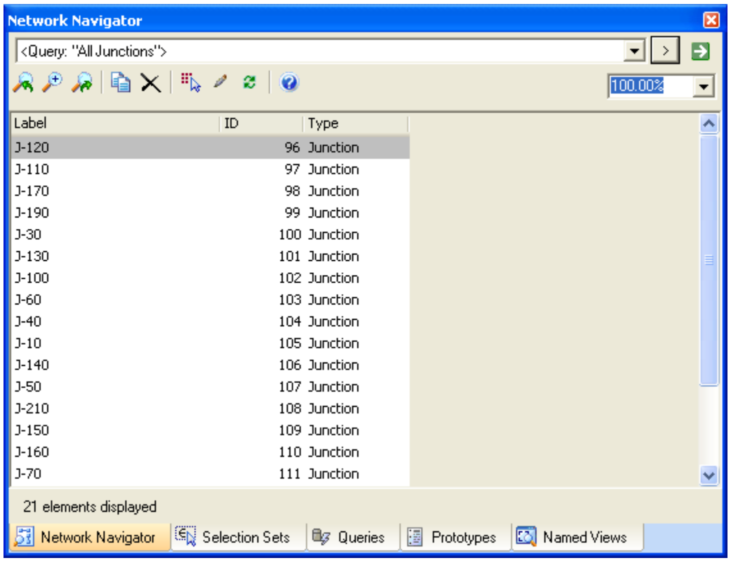
To delete multiple elements from a selection set:

1. Open the Selection Sets Manager by selecting **View > Selection Sets** or clicking the **Selection Sets** button on the View toolbar.
2. In the Selection Sets Manager, highlight the selection set that contains elements you want to delete.
3. Click the **Select In Drawing** button in the Selection Sets Manager to highlight all of the selection set's elements in the drawing pane.
   – If there is only one selection set listed in the Selection Set Manager, you don't have to highlight it before clicking the Select In Drawing button.
4. Shift-click (hold down the Shift key and click the left mouse button) any selected elements that you do not want to delete.
5. Right-click and select **Delete**. The highlighted elements in the selection set are deleted from your model.

Using the Network Navigator

The Network Navigator consists of a toolbar and a table that lists the Label and ID of each of the elements contained within the current selection. The selection can include elements highlighted manually in the drawing pane, elements contained within a selection set, or elements returned by a query.

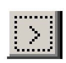
To open the Network Navigator, click the View menu and select the Network Navigator command, press <Ctrl+3>, or click the Network Navigator button  on the View toolbar.



The following controls are included in Network Navigator:

Query Selection List

Choose the element sets to use in the query. Once a query is selected, it can be executed when you click the > icon.



If there is already a Query listed in the list box, it can be run when the Execute icon is clicked.

Execute

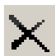
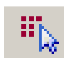
Click to run the selected query.

Previous

Zooms the drawing pane view to the element prior to the currently selected one in the list.

Zoom To

Zooms the drawing pane view to the selected element in the list.

Next 	Zooms the drawing pane view to the element below the currently selected element in the list.
Copy 	Copies the elements to the Windows clipboard.
Remove 	Removes the selected element from the list.
Select In Drawing 	Selects the elements in the drawing pane and performs a zoom extent based on the selection.
Highlight 	When this toggle button is on, elements returned by a query will be highlighted in the drawing pane to increase their visibility.
Refresh Drawing 	Refreshes the current selection.
Help 	Opens SewerGEMS V8*i* Help.

Predefined Queries

The Network Navigator provides access to a number of predefined queries grouped categorically, accessed by clicking the [>] button. Categories and the queries contained therein include:

Bolted Manholes

The Bolted Manholes query identifies manholes that have the Bolted Cover value set to True.

Control Structures

The Pond Outlet Control Structures query identifies pond outlet control structure elements.

Drawing Queries

Drawing Queries include the following:

- **Dead End Nodes** - Identifies nodes that are only connected to one link.
- **Duplicate Links** - Identifies instances in the model where a link shares both end nodes with another link.
- **Link Split Candidates**- Identifies nodes near a link that may be intended to be nodes along the link. The tolerance value can be set for the maximum distance from the link where the node should be considered as a link split candidate.
- **Links Missing Nodes** - Identifies which links are missing either one or both end nodes.
- **Nodes In Close Proximity** - Identifies nodes within a specific tolerance.
- **Orphaned Nodes** - Identifies nodes that are not connected to a link in the model.

Flooding

Flooding queries include the following:

- **Is Flooded? -** Identifies elements that are flooded during the current time step.
- **Is Flooded Ever? -** Identifies elements that are flooded at any point in time during an extended period simulation.

Hydrology

Runoff Methods - Identifies elements that use one of the runoff methods found in the submenu of this query.

Using Local Rainfall - Identifies catchments using the Local Rainfall hydrology method.

Inactive Elements

Identifies elements marked Inactive. The elements available under this query are divided into three subgroups: points (node elements), polygons (catchments and ponds), and polylines (link elements).

Inlet Types

- **Inflow Capture Curve Inlets** - Identifies nodes using inflow-capture curve inlets.
- **Maximum Capacity Inlets** - Identifies nodes using maximum capacity inlets.

Pipes

- **Culverts -** Identifies culvert link elements.
- **Dry Pipes -** Identifies link elements with no flow during the simulation.
- **Exceed Capacity** - Identifies link elements whose flow exceeds their capacity at some point during the simulation.
- **Shapes -** Identifes all link elements of the specified shape.

Pollutant Point Sources

Identifies elements that are pollutant point sources. This query is divided into subgroups for each element type.

Slopes

- **Horizontal Slopes -** Identifies link elements with horizontal slopes.
- **Near Horizontal** - Slopes Identifies link elements that have near-horizontal slopes.
- **Negative Slopes -** Identifies link elements with negative slopes (links whose upstream elevation is lower than their downstream elevation).

Surface Storages

- **Catch basins with Surface Storage -** Identifies catchbasin elements with surface storage capacity.
- **Manholes with Surface Storage -** Identifies manhole elements with surface storage capacity.

Using Prototypes

Prototypes allow you to enter default values for elements in your network. These values are used while laying out the network. Prototypes can reduce data entry requirements dramatically if a group of network elements share common data.

For example, if a section of the network contains all three foot-diameter manholes , use the manhole prototype to set the Diameter field to 3.00 ft. When you create a new manhole in your model, its diameter attribute will default to 3.00 ft.

Note: Changes to the prototypes are not retroactive and will not affect any elements created prior to the change.

If a section of your system has distinctly different characteristics than the rest of the system, adjust your prototypes before laying out that section. This will save time when you edit the properties later.

For instructions on how to create prototypes, see "Creating Prototypes" on page 6-286.

Creating Prototypes

Prototypes contain default values for Bentley SewerGEMS V8*i* elements. You create prototypes in the Prototypes Manager.

To create a prototype:

1. Open your Bentley SewerGEMS V8*i* project or start a new project.
2. Select **View > Prototypes** or press **Ctrl+6**.

   The Prototypes Manager opens. All Bentley SewerGEMS V8*i* element types are displayed in an expanding and collapsing list.
3. Select the element type for which you want to create a prototype, then click the **New** button.

   The element type in the list expands to display all the prototypes that exist for that element type.

   Each element type contains a default prototype, which is not editable, and any prototypes that you have created. The current set of default values for each element type is identified by the Make Current icon.
4. Double-click the prototype you just created. The Property Editor for the element type opens.
5. Edit the attribute values in the Property Editor as required.
6. To make the new prototype the default, click the **Make Current** button in the Prototypes Manager.

   The icon next to the prototype changes to indicate that the values in the prototype will be applied to all instances of that element type that you add to your current project.
7. Perform the following optional steps:
   - To rename a prototype, select the prototype in the list and click the **Rename** button.

– To delete a prototype, select the prototype in the list and click the **Delete** button.

– To view a report of the default values in the prototype, select the prototype in the list and click the **Report** button.

Prototypes Manager

The Prototypes Manager allows you to create prototypes, which contain default common data for each element type. The Prototypes Manager consists of a toolbar and a list pane, which displays all of the elements available in Bentley SewerGEMS V8*i*.

The list of elements in the Prototypes Manager list pane is expandable and collapsible. Click on the Plus sign to expand an element and see its associated prototypes. Click on the Minus sign to collapse the element.

Each element in the list pane contains a default prototype; you cannot edit this default prototype. The default prototypes contains common values for each element type; if you add elements to your model without creating new prototypes, the data values in the default prototypes appear in the Property Editor for that element type.

The toolbar contains the following buttons:

	**New**	Creates a new prototype of the selected element.
	**Delete**	Deletes the prototype that is currently highlighted in the list pane.

	**Rename**	Lets you rename the prototype that is currently highlighted in the list pane.
	**Make Current**	Lets you make the prototype that is currently highlighted in the list pane the default for that element type. When you make the current prototype the default, every element of that type that you add to your model in the current project will contain the same common data as the prototype.
	**Report**	Lets you view a report of the data associated with the prototype that is currently highlighted in the list pane.
	**Help**	Displays online help for the Prototypes Manager.

Engineering Libraries

Engineering Libraries are powerful and flexible tools that you use to manage specifications of common materials, objects, or components that are shared across projects. Some examples of objects that are specified through engineering libraries include pipe materials, storm events, and unit sanitary loads. You can modify engineering libraries and the items they contain by using the Engineering Libraries command in the Tools menu, or by clicking the ellipsis (…) buttons available next to the fields in dialog boxes that make use of engineering libraries.

Note: The data for each engineering library is stored in an XML file in your Bentley SewerGEMS V8*i* program directory. We strongly recommend that you edit these files only using the built-in tools available by selecting Tools > Engineering Libraries.

You work with engineering libraries and the items they contain in the Engineering Libraries dialog box, which contains all of the project's engineering libraries. Individual libraries are compilations of library entries, along with their attributes. For more information about working with engineering libraries, see "Working with Engineering Libraries" on page 6-289.

By default, each project you create in SewerGEMS V8*i* uses the items in the default libraries. In special circumstances, you may wish to create custom libraries to use with one or more projects. You can do this by copying a standard library or creating a new library.

When you change the properties for an item in an engineering library, those changes affect all projects that use that library item. At the time a project is loaded, all of its engineering library items are synchronized to the current library. Items are synchronized based on their label. If the label is the same, then the item's values will be made the same.

The default libraries that are installed with Bentley SewerGEMS V8*i* are editable. In addition, you can create a new library of any type, and can then create new entries of your own definition.

- Library types are displayed in the Engineering Library manager in an expanding/collapsing tree view.
- Library types can contain categories and subcategories, represented as folders in the tree view.
- Individual library entries are contained within the categories, subcategories, and folders in the tree view.
- Libraries, categories, folders, and library entries are displayed in the tree view with their own unique icons. You can right-click these icons to display submenus with different commands.

Working with Engineering Libraries

When you select a library entry in the tree view, the attributes and attribute values associated with the entry are displayed in the editor pane on the right side of the dialog box.

Working with Libraries

Right-clicking a Library icon in the tree view opens a shortcut menu containing the following commands:

Create Library	Creates a new engineering library of the currently highlighted type.
Add Existing Library	Lets you add an existing engineering library that has been stored on your hard drive as an .xml file to the current project.

Working with Categories

Right-clicking a Category icon in the tree view opens a shortcut menu containing the following commands:

Add Item	Creates a new entry within the current library.
Add Folder	Creates a new folder under the currently highlighted library.
Save As	Lets you save the currently highlighted category as an .xml file that can then be used in future projects.
Remove	Deletes the currently highlighted category from the library.

Working with Folders

Right-clicking a Folder icon in the tree view opens a shortcut menu containing the following commands:

Add Item	Creates a new entry within the current folder.
Add Folder	Creates a new folder under the currently highlighted folder.
Rename	Lets you rename the currently highlighted folder.
Delete	Deletes the currently highlighted folder and its contents.

Working with Library Entries

Right-clicking a Library Entry icon in the tree view opens a shortcut menu containing the following commands:

Rename	Lets you rename the currently highlighted entry.
Delete	Deletes the currently highlighted entry from the library.

Engineering Libraries Dialog Box

The Engineering Libraries dialog box contains an explorer tree-view pane on the left, a library entry editor pane on the right, and the following buttons above the explorer tree view pane:

	**New**	Opens a submenu containing the following commands: • **Create Library**—Creates a new engineering library. • **Add Existing Library**—Lets you add an existing engineering library that has been stored on your hard drive as an .xml file to the current project.
	**Delete**	Removes the currently highlighted engineering library from the current project.
	**Rename**	Lets you rename the currently highlighted engineering library.

Sharing Engineering Libraries On a Network

You can share engineering libraries with other SewerGEMS V8*i* users in your organization by storing the engineering libraries on a network drive. All users who will have access to the shared engineering library should have read-write access to the network folder in which the library is located.

To share an engineering library on a network, open the Engineering Libraries in SewerGEMS V8*i* and create a new library in a network folder to which all users have read-write access.

Pipe Catalog Dialog Box

This dialog box allows you to create, edit, and view catalog pipes. Catalog pipes are an efficient way to reuse common physical pipe definitions.

The dialog box contains a toolbar, a Pipe Catalog list pane, and two tabs. The toolbar contains the following buttons:

New — Creates a new entry in the Pipe Catalog List Pane.

Delete — Deletes the entry that is currently highlighted in the Pipe Catalog List Pane.

Rename — Lets you rename the entry that is currently highlighted in the Pipe Catalog List Pane.

Report — Lets you generate a preformatted report that contains the input data associated with the entry that is currently highlighted in the Pipe Catalog List Pane.

Synchronize — Clicking this button opens a submenu containing the following commands:

- **Browse Engineering Library**—This command opens the Engineering Library manager dialog, allowing you to browse the Pipe Catalog Library.
- **Synchronize From Library**—This command allows you to update a pipe catalog that was previously imported from a Pipe Catalog Engineering Library to reflect changes that have been made to the library since it was imported.
- **Synchronize To Library**—This command allows you to update an existing Pipe Catalog Engineering Library using current Pipe Catalog entries that were initially imported but have since been modified.
- **Import From Library**—This command allows you to import catalog entries from an existing Pipe Catalog Engineering Library.
- **Export To Library**—This command allows you to export the current catalog entries to an existing Pipe Catalog Engineering Library.

The following table describes the rest of the controls in the Pipe Catalog dialog box.

Pipe Catalog List Pane	Located on the left side of the dialog box, displays a list of all of the catalog pipes that have been defined in the current project. Highlighting a catalog pipe in this list causes the Cross Section Shape and Roughness Sections to display the associated information with the highlighted pipe.
Cross Section Shape	Located in the top-right corner of the Pipe Catalog tab, contains controls that allow you to define the size and shape of the catalog pipe currently highlighted in the List Pane. The controls that appear change according to the Cross Section Type that is selected.
Cross Section Type	Lets you define the type of cross section for the currently highlighted catalog pipe.
Diameter	Lets you define the diameter of the pipe. This field is only available for Circular catalog pipes.
<Section Type> Rise	Lets you define the rise (height) of the catalog pipe. This field is available for all cross section types except Circular.
<Section Type> Span	Lets you define the span (width) of the catalog pipe. This field is available for all cross section types except Circular.
Roughness	Located in the bottom-right corner of the Pipe Catalog tab, lets you define the roughness attributes of the catalog pipe currently highlighted in the List Pane. The controls that are available change depending on the Roughness Type selected.
Roughness Type	Lets you specify which of the available roughness methods to be applied to the catalog pipe currently highlighted in the List Pane. The other controls available in section are dependent on the selection made in this box.
Material	Lets you enter a material label. This field is informational only, and will not affect the roughness properties of the associated catalog entry.

Manning's n	Lets you define the roughness value for the catalog pipe. This field is available only when the Roughness Type is Single Manning's n.
Depth vs. Manning's Table	Lets you define a depth vs. roughness curve for the catalog pipe. This field is available only when the Roughness Type is Manning's n-Depth Curve.
Manning's vs. Discharge Table	Lets you define a flow vs. roughness curve for the catalog pipe. This field is available only when the Roughness Type is Manning's n-Flow.
Library Tab	Displays information pertaining to the catalog entry that is currently highlighted in the List Pane, including: • ID • Label • Modified Date • Library Source • Library Modified Date • Synchronization Status

Using the SWMM Water Quality Solver

Bentley SewerGEMS V8*i* can perform water quality modeling using the SWMM water quality solver. In order to make a water quality run choose the **SWMM5 Explicit Engine** from the **Analysis > Calculation Options > Engine Type** dialog.

You must then define the pollutant being modeled using the **Component > SWMM Extensions > Pollutants** dialog (see ”Pollutants Dialog Box’). In the Pollutants dialog, the enter a name for the pollutant and define the properties.

If water quality data is present in a scenario, water quality calculations will be performed.

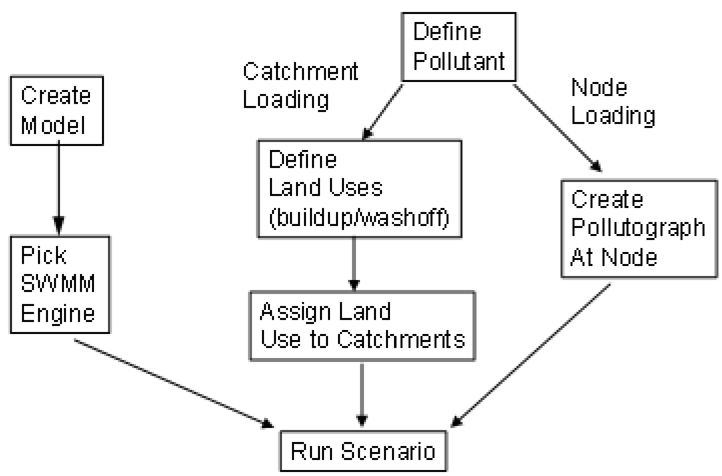
There are two methods for loading pollutants in a water quality simulation.

1. Point (node) load, which involves assigning a pollutograph (see ”Adding Pollutographs to a Node’) to a node element (such as a manhole or a cross section). This can be done in the element property grid or under the water quality alternative of the alternative manager for that element. This method is used for point loads such as industrial dischargers or normal domestic customers.

2. Catchment runoff, where the pollutant enters the system based on one of several different washoff functions for each land use assigned to a catchment. Land uses and washoff (and optional buildup/treatment) functions are defined under **Components** > **SWMM Extensions** > **Land Use**. Then under, **Alternatives** > **Water Quality** > **Catchments**, define what fraction of the catchment is associated with each land use (see "Land Uses Collection Dialog Box') and the initial buildup of pollutants (see "Initial Buildup Collection Dialog Box') at the start of the run. In long-term runs, you can specify the rate at which pollutants build up on the catchment surface (and are possibly removed by treatments such as street sweeping).

It is not possible to view the water quality results using the standard graphing. Instead, you must open the property grid for the element, scroll down to **Results** > **Pollutant** and open the collection by clicking on the ellipsis button. This will display a graph of concentration vs. time. Switching to the **Data** tab will give tabular results.

An overview of the water quality modeling process is shown below.



Water Quality Modeling Overview

Note: When SewerGEMS V8*i* is running calculations using the Implicit Engine Type (this setting is found in the Calculation Options Manager), SWMM attributes and their associated values are not considered. Only when the SWMM Engine Type is used will the data contained in these dialog boxes have any effect on the calculated results.

See also:

- "Evaporation Dialog Box" on page 6-297
- "Aquifers Dialog Box" on page 6-298

- "Control Sets Dialog Box" on page 6-299
- "Pollutants Dialog Box" on page 6-305
- "Adding Pollutographs to a Node" on page 6-307
- "Land Uses Dialog Box" on page 6-311
- "Adding Treatment to a Node" on page 6-321
- "Initial Buildup Collection Dialog Box" on page 6-323

SWMM Hydrology

You can use the Explicit (SWMM 5) engine in Bentley SewerGEMS V8*i* to route flows through the system. However, in order to use SWMM hydraulics, you must load the model through inflows at nodes, or you must select **EPA-SWMM Runoff** as the runoff method for a catchment. If you are not familiar with SWMM hydrology, we strongly recommend that you read SWMM 5 documentation for a detailed discussion of the topic.

The overall process through which precipitation is converted to flow in conduits and channels is summarized in the following figure.

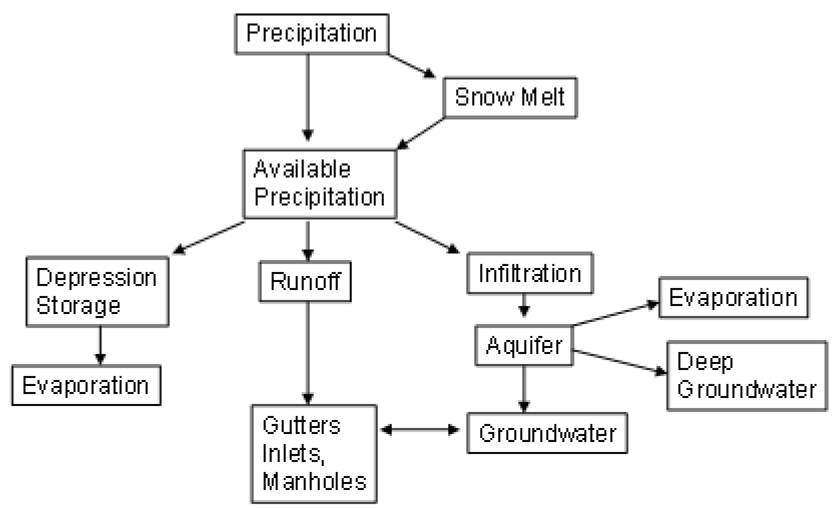


Figure 6-1: Conversion of Precipitation to Flow

The parameters used in determining runoff can be divided into two categories:

Parameters that are shared by the SWMM model and the Bentley SewerGEMS V8*i*/SewerGEMS native hydrology calculations. These include SCS runoff curve number, Horton max and min infiltration rates, etc. You can enter these parameters in the Property Editor or in the FlexTable for a specific catchment.

Parameters that are unique to the SWMM model. These include evaporation rates, aquifer conductivity, and wilting point. To enter these parameters, you must select **Analysis > SWMM Extensions**, then select the appropriate SWMM dialog.

Evaporation Dialog Box

This dialog box allows you to define the evaporation that can occur for standing water on subcatchment surfaces, for subsurface water in groundwater aquifers, and from water held in storage units.

Access this dialog box by selecting **Analysis** > **SWMM Extensions** > **Evaporation**.

The following controls are available in this dialog box are as follows:

Note: Available controls vary, depending on the Evaporation Type that is chosen.

Label	Displays the label of the current evaporation rate.
Evaporation Type	Allows you to specify the type of evaporation rate. The setting controls will vary according to the type chosen here, as follows: • **Constant Evaporation**—Use this choice if evaporation remains constant over time. Enter the rate of evaporation in the Constant Evaporation field that appears when this type is chosen. • **Monthly Evaporation**—Use this choice to supply an average rate for each month of the year. Enter the value for each month in the appropriate data fields that appear when this type is chosen. Note that rates remain constant within each month. • **Time-Series Evaporation**—Select this choice if evaporation rates will be specified in a time series. Enter the evaporation value for each of the time series points. Note that for each date specified in the time series, the evaporation rate remains constant at the value supplied for that date until the next date in the series is reached (i.e., interpolation is not used on the series).

Report	This button creates a report based on your evaporation settings.

Aquifers Dialog Box

Aquifers are sub-surface groundwater areas used to model the vertical movement of water infiltrating from the subcatchments which lie above them. They also permit the infiltration of groundwater into the conveyance system, or exfiltration of surface water from the conveyance system, depending on the hydraulic gradient that exists.

Access this dialog box by selecting **Analysis > SWMM Extensions > Aquifers**.

This dialog box contains a toolbar, a list pane on the left that displays all of the aquifers that have been defined in the current project, and the attribute fields on the right that permit the values to be defined for the aquifer that is currently highlighted in the list pane. The toolbar contains the following buttons:

	**New**	Creates a new entry in the Aquifer List Pane.
	**Delete**	Deletes the entry that is currently highlighted in the Aquifer List Pane.
	**Rename**	Lets you rename the entry that is currently highlighted in the Aquifer List Pane.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the entry that is currently highlighted in the Aquifer List Pane.

The attribute fields along the right side of the dialog box include:

Porosity	Lets you define the volume of voids / total soil volume for the currently highlighted aquifer.
Wilting Point	Lets you define soil moisture content at which plants cannot survive for the currently highlighted aquifer.

Field Capacity	Lets you define soil moisture content after all free water has drained off for the currently highlighted aquifer.
Aquifer Conductivity	Lets you define the soil's saturated hydraulic conductivity for the currently highlighted aquifer.
Conductivity Slope	Lets you define the slope of conductivity vs. soil moisture content curve for the currently highlighted aquifer.
Tension Slope	Lets you define the slope of soil tension vs. soil moisture content curve for the currently highlighted aquifer.
Upper Evaporation Fraction	Lets you define the fraction of total evaporation available for evapotranspiration in the upper unsaturated zone for the currently highlighted aquifer.
Lower Evaporation Depth	Lets you define the maximum depth into the lower saturated zone over which evapotranspiration can occur for the currently highlighted aquifer.
Lower Groundwater Loss Rate	Lets you define the rate of percolation from the saturated zone to deep groundwater when water table is at ground surface for the currently highlighted aquifer.
Elevation (Bottom)	Lets you define the elevation of the bottom of the aquifer for the currently highlighted aquifer.
Water Table Elevation	Lets you define the elevation of the water table in the aquifer at the start of the simulation for the currently highlighted aquifer.
Unsaturated Zone Moisture	Lets you define the moisture content of the unsaturated upper zone of the aquifer at the start of the simulation for the currently highlighted aquifer.

Control Sets Dialog Box

This dialog box allows you to create, view, and manage SWMM controls. Access it by selecting **Analysis > SWMM Extensions > Control Sets**.

This dialog box contains a toolbar, a list pane on the left that displays all of the controls that have been defined in the current project, and a control editor pane on the right that allows you to enter and edit the currently highlighted SWMM control definition. The toolbar contains the following buttons:

	**New**	Creates a new entry in the Control List Pane.
	**Delete**	Deletes the entry that is currently highlighted in the Control List Pane.
	**Rename**	Lets you rename the entry that is currently highlighted in the Control List Pane.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the entry that is currently highlighted in the Control List Pane.

The Control Editor Pane allows you to define SWMM controls. Each control rule is a series of statements of the form:

```
RULE   ruleID
IF     condition_1
AND    condition_2
OR     condition_3
AND    condition_4
Etc.
THEN   action_1
AND    action_2
Etc.
ELSE   action_3
```

```
AND   action_4
```

Etc.

```
PRIORITY value
```

where keywords are shown in boldface and ruleID is an ID label assigned to the rule, condition_n is a Condition Clause, action_n is an Action Clause, and value is a priority value (e.g., a number from 1 to 5).

Only the RULE, IF and THEN portions of a rule are required; the other portions are optional.

Blank lines between clauses are permitted and any text to the right of a semicolon is considered a comment.

When mixing AND and OR clauses, the OR operator has higher precedence than AND, i.e.,

```
IF A or B and C
```

is equivalent to

```
IF (A or B) and C
```

If the interpretation was meant to be

```
IF A or (B and C)
```

then this can be expressed using two rules as in

```
IF A THEN …

IF B and C THEN …
```

The PRIORITY value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

Note: Remember the pump icon represents a collection of pumps. When setting up controls be sure to reference the individual pumps in the pump collection, not the overall pump object. For example, PMP-1 will have pump in its collection PMP-1.1, PMP-1.2. Use the labels as defined in the pump collection dialog, not the overall pump station name.

Control Set Formats

Formats

Each control set consists of a series of statements of the form:

```
RULE   ruleID
IF     condition_1
AND    condition_2
OR     condition_3
AND    condition_4
```

Etc.

```
THEN   action_1
AND    action_2
```

Etc.

```
ELSE   action_3
AND    action_4
```

Etc.

```
PRIORITY value
```

where keywords are shown in boldface and ruleID is an ID label assigned to the rule, condition_n is a Condition Clause, action_n is an Action Clause, and value is a priority value (e.g., a number from 1 to 5).

Condition Clauses

A Condition Clause of a Control Rule has the following format:

```
object id attribute relation value
```

Where:

`object` = a category of object

`id` = the object's ID label

`attribute` = an attribute or property of the object

`relation` = a relational operator (=, <>, <, <=, >, >=)

`value` = an attribute value

Some examples of condition clauses are:

```
NODE N23 DEPTH > 10
PUMP P45 STATUS = OFF
SIMULATION CLOCKTIME = 22:45:00
```

The objects and attributes that can appear in a condition clause are as follows:

Table 6-6: Objects and Attributes in Condition Clauses

Object	Attribute	Value
NODE	DEPTH HEAD INFLOW	numerical value numerical value numerical value
LINK	FLOW DEPTH	numerical value numerical value
PUMP	STATUS FLOW	ON or OFF numerical value
ORIFICE WEIR	SETTING	fraction open
SIMULATION	TIME DATE CLOCKTIME	elapsed time in decimal hours or hr:min:sec month/day/year time of day in hr:min:sec

Action Clauses

An Action Clause of a Control Rule can have one of the following formats:

```
PUMP        id   STATUS    =    ON/OFF
ORIFICE     id   SETTING   =    value
WEIR        id   SETTING   =    value
```

Where `SETTING` is the fractional amount that an orifice is fully open or to the fractional amount of the original height between the crest and the top of a weir that remains (i.e., weir control is accomplished by moving the crest height up and down).

Some examples of action clauses are:

```
PUMP P67 STATUS = OFF
ORIFICE O212 SETTING = 0.5
```

Only the `RULE`, `IF` and `THEN` portions of a rule are required; the other portions are optional.

Blank lines between clauses are allowed, and any text to the right of a semicolon is considered a comment.

When mixing `AND` and `OR` clauses, the `OR` operator has higher precedence than `AND`, i.e.,

```
IF A or B and C
```

is equivalent to

```
IF (A or B) and C.
```

If the interpretation was meant to be

```
IF A or (B and C)
```

then this can be expressed using two rules as in

```
IF A THEN ...
IF B and C THEN ...
```

The PRIORITY value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

Examples

The following are examples of control rules.

Simple time-based pump control

```
RULE R1
IF SIMULATION TIME > 8
THEN PUMP 12 STATUS = ON
ELSE PUMP 12 STATUS = OFF ;
```

Multi-condition orifice gate control

```
RULE R2A
IF NODE 23 DEPTH > 12 AND LINK 165 FLOW > 100
THEN ORIFICE R55 SETTING = 0.5
RULE R2B
IF NODE 23 DEPTH > 12 AND LINK 165 FLOW > 200
THEN ORIFICE R55 SETTING = 1.0
RULE R2C
IF NODE 23 DEPTH <= 12 OR LINK 165 FLOW <= 100
THEN ORIFICE R55 SETTING = 0 ;
```

Pump station operation (as in a SWMM4 Type5 pump)

```
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
```

Pollutants Dialog Box

This dialog box allows you to track the generation, inflow and fate of any number of user-specified pollutants. In addition, 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 on subcatchment areas are determined by the Land Uses assigned to those areas.

Input loadings of pollutants from external and dry weather inflows are supplied through time series data associated with particular nodes of the collection system.

Access this dialog box by selecting **Analysis** > **SWMM Extensions** > **Pollutants**.

This dialog box contains a toolbar, a list pane on the left that displays all of the pollutants that have been defined in the current project, and the attribute fields on the right that permit the values to be defined for the pollutant that is currently highlighted in the list pane. The toolbar contains the following buttons:

New	Creates a new entry in the Pollutant List Pane.
Delete	Deletes the entry that is currently highlighted in the Pollutant List Pane.
Rename	Lets you rename the entry that is currently highlighted in the Pollutant List Pane.
Report	Lets you generate a preformatted report that contains the input data associated with the entry that is currently highlighted in the Pollutant List Pane.

The attribute fields along the right side of the dialog box include:

Rain Concentration	Lets you define the concentration of the pollutant in rain water for the currently highlighted pollutant.
Groundwater Concentration	Lets you define the concentration of the pollutant in groundwater for the currently highlighted pollutant.
Decay Coefficient	Lets you define the first-order decay coefficient for the currently highlighted pollutant.
Co-Pollutant	Lets you define the name of another pollutant runoff concentration the current pollutant is dependent on for the currently highlighted pollutant.
Co-Fraction	Lets you define the fraction of the co-pollutant's runoff concentration that contributes to the runoff concentration for the currently highlighted pollutant.

I & I Concentration	Lets you define the concentration of the pollutant in any Infiltration/Inflow.

Adding Pollutographs to a Node

Pollutographs are plots of time vs. mass rate or time vs. concentration, depending on which constituent inflow type you choose (mass or concentration). You create pollutographs in the Pollutographs dialog box and add them to individual manholes in your model using the node's Property Editor.

To add a pollutograph to a node:

1. Change the engine type to Explicit (SWMM 5) by performing these steps:
    a. Select **View > Calculation Options**.
    b. In the Calculation Options Manager, double-click **Base Calculation Options** or any other calculation options profile you have created.
    c. In Calculation Options section of the Property Editor, select **Explicit (SWMM 5)** as the Engine Type.
2. Define a pollutant by performing these steps:
    a. Select **Analysis > SWMM Extensions > Pollutants**.
    b. In the Pollutants dialog box, click the **New** button to create a new pollutant, then enter all the data that define the pollutant.
    c. Click **Close** to close the Pollutants dialog box and save your pollutant(s).
3. Create a pollutograph by performing these steps:
    a. Select **Analysis > SWMM Extensions > Pollutographs**.
    b. In the Pollutographs dialog box, click the **New** button, then click **Mass** or **Concentration** to create a new pollutograph.
    c. Enter all the data that define the pollutograph.

       For pollutographs based on mass, select a Pollutant, enter a Mass Conversion Factor, then enter Time vs. Mass Rate data points.

       For pollutographs based on concentration, select a Pollutant, then enter Time vs. Concentration data points.

Note: Bentley SewerGEMS V8*i* validates your data as you enter it and displays errors and warnings in the status bar at the bottom of the Pollutographs dialog box. Be sure to check this status bar for any errors or warnings as you enter data.

d. Optionally, view a report or plot of your pollutograph.

e. Click **Close** to save the pollutograph.

4. Click the desired manhole in your model. The Property Editor for the manhole appears. (If the Property Editor is not already displayed, you must double-click the node in your model.)
5. In the SWMM Extended Data section of the Property Editor, click the Pollutographs field, then click the **Ellipses (...)** button.
6. In the Pollutograph Collection dialog box, add pollutographs to the pollutograph collection by performing these steps:
   a. Click the **New** button to add a row to the Pollutograph table.
   b. Click the down arrow in the first row then select an existing pollutograph. If there are no pollutographs in your project, you can click the **Ellipses (...)** button to display the Pollutographs dialog box, where you can create new pollutographs.
   c. Repeat Steps a and b for every pollutograph you wish to add to the collection.
   d. Press **OK** to close the dialog box and add the collection to the node.
7. Complete your model, then click the **Compute** button on the Compute toolbar.
8. When the model has been successfully computed, open the Property Editor for the node the contains the Pollutograph Collection.
9. In the Results section of the Property Editor, click the **Ellipses (...)** button in the Pollutants field to view pollutant results.

Pollutograph Dialog Box

This dialog box contains a toolbar, a list pane on the left that displays all of the pollutographs that have been defined in the current project, and attribute fields on the right that let you enter values for the pollutograph that is currently highlighted in the list pane.

The toolbar contains the following buttons:

	**New**	Opens a submenu containing the following options: • **Mass**—Lets you create a plot of time vs. mass rate. • **Concentration**—Lets you create a plot of time vs. concentration.
	**Delete**	Deletes the currently-highlighted pollutograph from the list pane.
	**Rename**	Lets you rename the currently-highlighted pollutograph.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently-highlighted pollutograph.

The attribute fields along the right side of the dialog box include:

Pollutant	Lets you select the pollutant for the pollutograph. Select the pollutant from the drop-down menu or click the **Ellipses (...)** button to open the SWMM Pollutants dialog box, where you can define new pollutants.
New	This button adds a new row to the pollutograph table.
Delete	This button removes the current row from the pollutograph table.
Report	Displays a report of the data in the pollutograph table.
Graph	Displays a plot of the data in the pollutograph table.
Mass Conversion Factor	Lets you enter a mass conversion factor. . This field is available only for pollutographs using mass as the constituent inflow type.

Pollutograph Table	Lets you define the pollutograph by entering Time vs. Mass Rate points for pollutographs using mass as the constituent inflow type, or Time vs. Concentration points for pollutographs using concentration as the constituent inflow type.

There is also a status bar located at the bottom of the dialog box that displays any errors and warnings that may occur when you enter data.

Pollutograph Collection Dialog Box

The Pollutograph Collection dialog box lets you add multiple pollutographs to a node in your model. You access this dialog box from the SWMM Extended Data section of the Property Editor for the selected node.

The dialog box contains the following toolbar buttons:

	**New**	Adds a new row to the Pollutograph table. Click in the row to select a pollutograph from the drop-down menu, or click the **Ellipses (...)** button to create new pollutographs.
	**Delete**	Deletes the current row from the table.
	**Report**	Lets you view a report of the collection.

The dialog box also contains the following controls:

Pollutograph Table	Displays the pollutographs you have added to the collection. You add a pollutograph to the collection by clicking the **New** button, then selecting a pollutograph from the drop-down menu in the table row.

Pollutants Results Dialog Box

The Pollutants Results dialog box displays a plot for each pollutant after you successfully calculate your model. You access this dialog box by clicking the Ellipses (...) button in the Pollutants field in the Results section of the Property Editor.

The Pollutants Results dialog box contains a list pane on the left that displays all the pollutants assigned to the selected node, and a two tabs on the right:

Time vs. Concentration Tab	Displays a plot of Concentration over Time for each pollutant displayed in the list pane.
Data Tab	Displays the time (hours by default) vs. concentration (ppm by default) data points for each pollutant displayed in the list pane.

Land Uses Dialog Box

Land Uses are categories of activities or land surface characteristics that are assigned to catchment elements. Examples of land use activities are residential, commercial, industrial, and undeveloped. Land surface characteristics might include roof tops, lawns, paved roads, undisturbed soils, etc. Land uses are used solely to allow spatial variation in pollutant buildup and washoff rates.

You have complete freedom in defining land uses and assigning them to catchment elements. One approach is to assign a mix of land uses to each subcatchment, in which case all land uses in the subcatchment will have the same pervious/impervious characteristics. If this is not appropriate, the user can create subcatchments that have just a single land use classification along with a set of pervious/impervious characteristics that reflects the classification.

Access this dialog box by selecting **Analysis > SWMM Extensions > Land Uses**.

This dialog box contains a toolbar, a list pane on the left that displays all of the land uses that have been defined in the current project, and three tabs on the right that together permit the values to be defined for the land use that is currently highlighted in the list pane.

The toolbar contains the following buttons:

	**New**	Creates a new entry in the Land Use List Pane.
	**Delete**	Deletes the entry that is currently highlighted in the Land Use List Pane.
	**Rename**	Lets you rename the entry that is currently highlighted in the Land Use List Pane.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the entry that is currently highlighted in the Land Use List Pane.

The right side of the dialog contains the following tabs:

- ”Land Use General Tab’
- ”Land Use Buildup Tab’
- ”Land Use Washoff Tab’

Land Use General Tab

The General tab contains controls related to street cleaning. Street cleaning can be used on each land use category to periodically reduce the accumulated buildup of particular pollutants. This tab contains the following controls:

Street Cleaning Interval	Amount of time between cleaning.
Availability	The fraction of buildup of all pollutants that is available for removal by cleaning
Last Cleaned	Amount of time since land use category has last been cleaned.

Land Use Buildup Tab

Pollutant Buildup that accumulates over a category of land use is described by either a mass per unit of subcatchment area or per unit of curb length, as specified by the Normalizer field described below.

The buildup is usually given as mass of pollutant per unit area although they may be given as counts (in the case of coliforms) per unit curb length. Therefore B (and C1 in the functions below) can be given as lbs/acre, kg/hectare, coliform/acre, lbs/ft curb length, etc. Time in the functions refers to the number of antecedent dry days. Buildup is only used when the exponential washoff function is used to determine quality of runoff quality from catchments.

Three different functions can be used to describe buildup. The functions are all monotonically increasing. The figure below gives their relative shape.

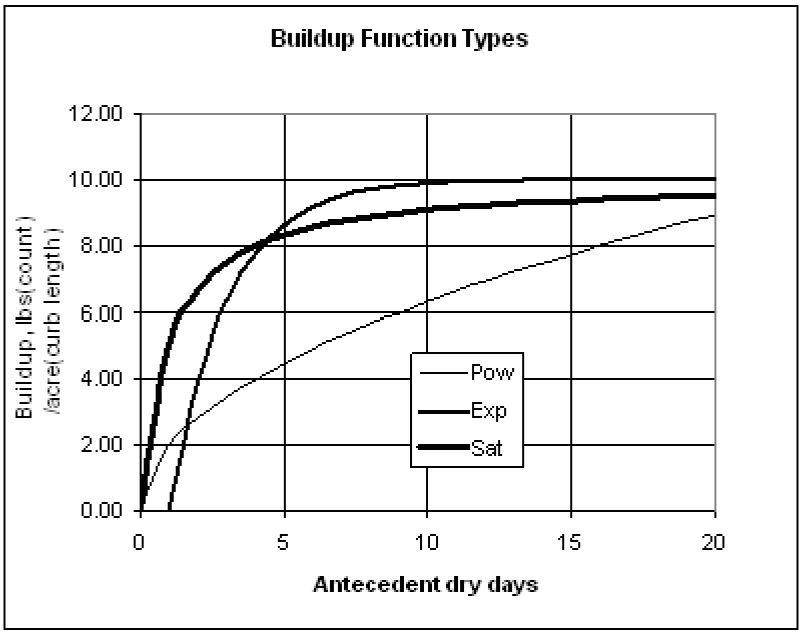


Figure 6-2: Buildup Function Types

The amount of buildup as a function of days of dry weather can be computed using one of the following functions:

- **Power Function**: Pollutant Buildup (B) accumulates proportional to time (t) raised to some power, until a maximum limit is achieved.

$$B = Min\left(C_1, C_2 t^{c_3}\right) \tag{6.1}$$

Where	B	=	buildup, mass(count/area(length)
	C_1	=	maximum possible build-up, mass (count)/ area(length)
	C_2	=	build-up rate
	C_3	=	time exponent

The effect of each coefficient is shown in the figure below.

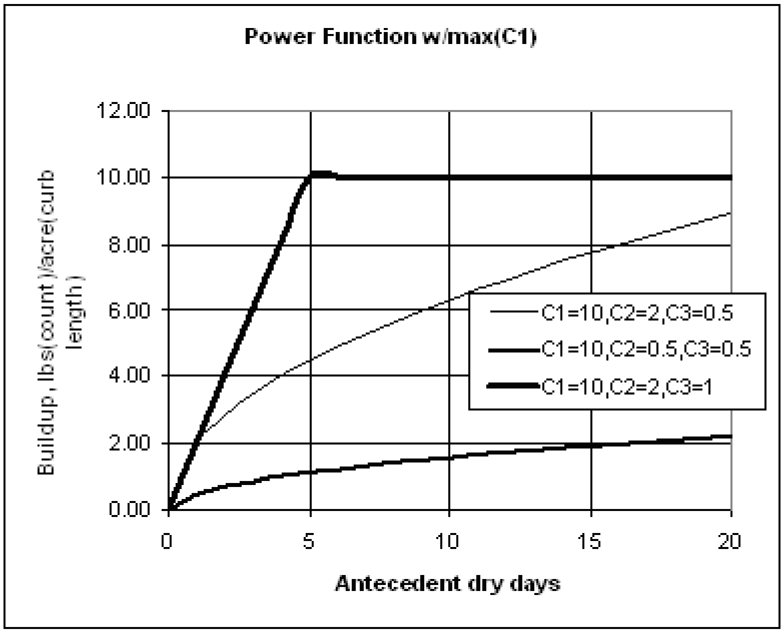


Figure 6-3: Effect of Power Function Coefficients

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

$$B = C_1 \left(1 - e^{-c_2 t}\right) \tag{6.2}$$

Where	B	=	buildup, mass(count)/area(length)
	C_1	=	maximum possible build-up, mass(count)/ area(length)
	C_2	=	build-up rate constant, 1/day

The effect of each coefficient is shown in the figure below.

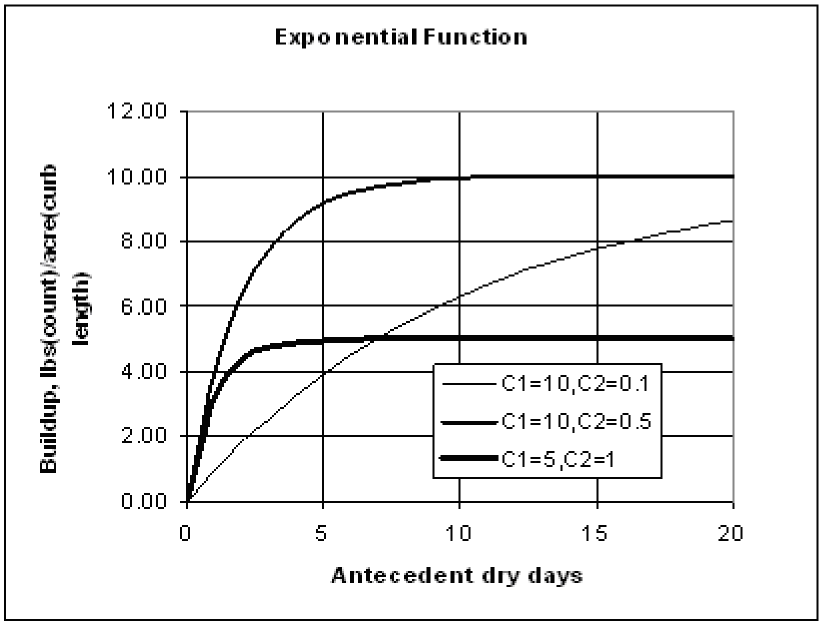


Figure 6-4: Effect of Exponential Function Coefficients

- **Saturation Function**: Buildup begins at a linear rate which proceeds to decline constantly over time until a saturation value is reached.

$$B = \frac{C_1 t}{C_2 + t} \tag{6.3}$$

Where	B	=	buildup, mass(count)/area(length)
	C_1	=	maximum possible build-up, mass(count)/ area(length)
	C_2	=	build-up rate constant, 1/day

The effect of each coefficient is shown in the figure below.

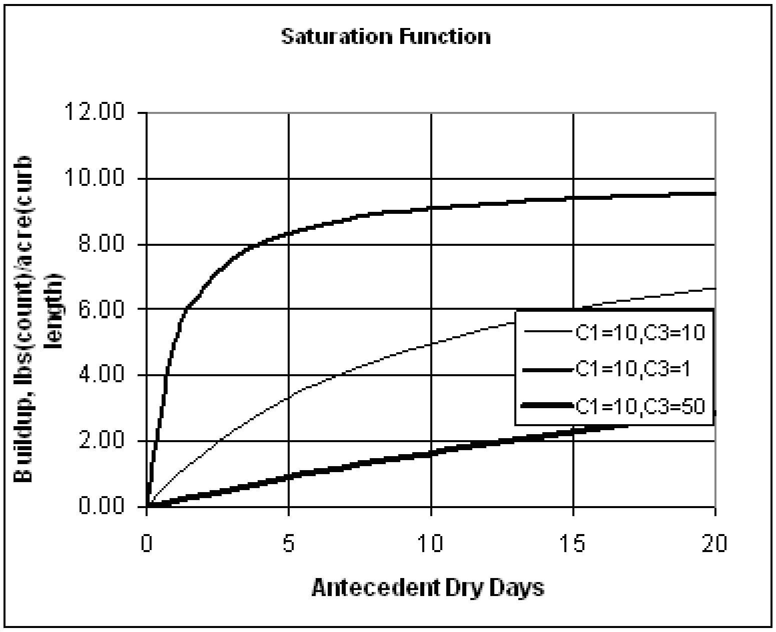


Figure 6-5: Effect of Saturation Function Coefficients

This tab contains a table with the following attribute columns:

Column	Description
Pollutant	This menu contains all of the pollutants that have been defined in the Pollutants dialog box for the current project. Select the one that should be used for the land use currently highlighted in the list pane.
Max. Buildup	This field allows you to define the value of C_1 in the above equations.
Rate Constant	This field allows you to define C_2 in the Power and Exponential equations described above.
Buildup Function	This menu allows you to specify which of the buildup functions described above will be used for the land use currently highlighted in the list pane.

Power Constant	This field allows you to define C_3 in the Power equation described above.
Half Saturation Constant	This field allows you to define C_2 in the Saturation equation described above.
Normalizer	Allows you to specify the method by which pollutant buildup is described for the land use currently highlighted in the list pane. Choices include Area (mass per unit of subcatchment area) or Curb (mass per unit of curb length).

Land Use Washoff Tab

Washoff refers to the amount of pollutants washed off from a catchment during a wet weather period. The user can select any of the three washoff functions described below:

- **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.

$$W = C_1 q^{c_2} B \tag{6.4}$$

Where	C_1	=	washoff coefficient
	C_2	=	washoff exponent
	q	=	runoff rate per unit area
	B	=	pollutant build-up in a mass per unit area or curb length

The effect of coefficients on washoff is shown in the following figure:

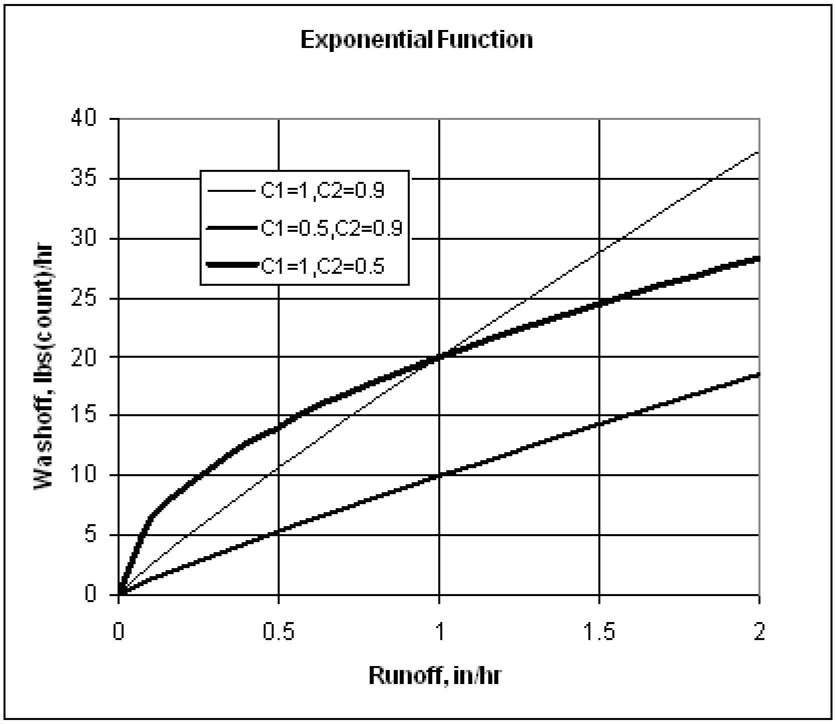


Figure 6-6: Effect of Exponential Function Coefficients on Washoff

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

$$W = C_1 Q^{C_2} \tag{6.5}$$

Where C_1 = washoff coefficient

C_2 = washoff exponent

Q = runoff rate, cfs (m^3/s)

- **Event Mean Concentration (EMC):** This is a special case of Rating Curve Washoff where the exponent (C_2) is 1.0 and the coefficient C_1 represents the concentration of any and all runoff in mass per liter.

The effect of coefficients on washoff is shown in the following figure:

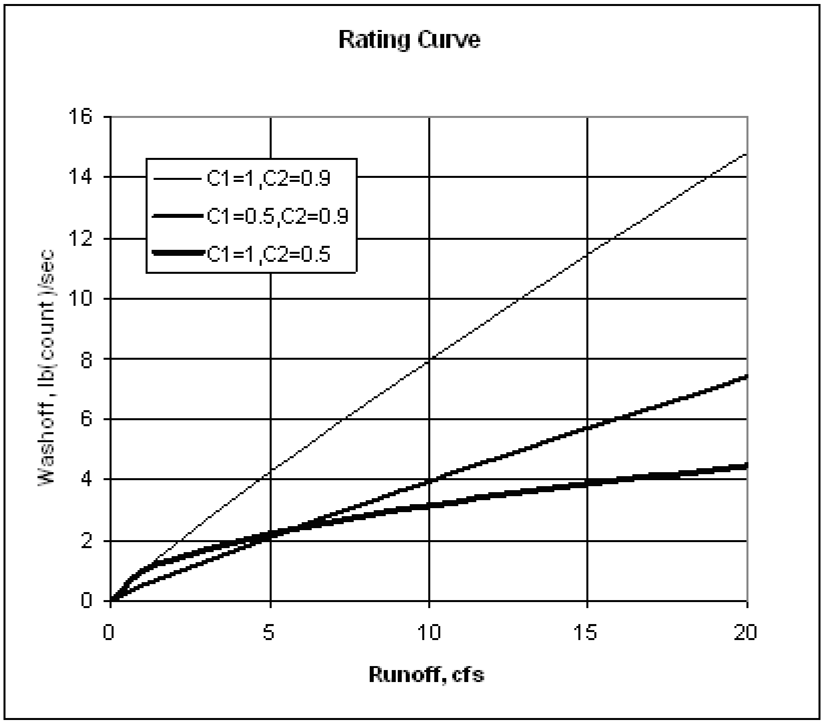


Figure 6-7: Effect of Rating Curve Coefficients on Washoff

This tab contains a table with the following attribute columns:

Column	Description
Pollutant	This menu contains all of the pollutants that have been defined in the Pollutants dialog box for the current project. Select the one that should be used for the land use currently highlighted in the list pane.
Washoff Function	This menu allows you to specify which of the buildup functions described above will be used for the land use currently highlighted in the list pane.
Washoff Coefficient	This field allows you to define C_1 in the equations described above.

Washoff Exponent	This field allows you to define C_2 in the Exponential and Rating Curve equations described above.
Cleaning Efficiency	The fraction of available buildup for each pollutant removed by cleaning.
Removal Efficiency	Washoff loads for a given pollutant and land use category can be reduced by a fixed percentage by specifying a Removal Efficiency in this field which reflects the effectiveness of any BMP controls associated with the land use.

Land Uses Collection Dialog Box

The Land Uses Collection dialog box lets you add multiple land uses as a collection to a catchment. You access the Land Uses Collection dialog box from the Property Editor for a catchment that uses EPA-SWMM Runoff as the runoff method.

The dialog box contains the following buttons:

	**New**	Adds a new row to the Land Uses table. Each row in the table contains a single land use entry.
	**Delete**	Deletes the current row from the Land Uses table.

The Land Uses Collection dialog box also contains a table with the following columns:

Column	Description
Catchment Land Use	Lets you select a land use entry to include in the collection. Click in the row to select a land use entry from the drop-down menu, or click the Ellipses (...) button to create new land uses.
Percent of Catchment Area (%)	Lets you specify the percent of the catchment area affected by the land.

Adding Treatment to a Node

You can model the removal of pollutants from the flow entering any drainage system by assigning treatment functions to the node. In Bentley SewerGEMS V8*i*, you assign treatment functions to a node by adding a treatment collection, which contains one or more pollutants and their associated treatment expressions.

A treatment expression involves the pollutant concentration of the mixture of all flow streams entering the node (use the pollutant name to represent a concentration), or the removal of other pollutants (use R_ prefixed to the pollutant name to represent removal).

Treatment expressions have the general form:

R = f(P, R_P, V)

or

C = f(P, R_P, V)

Where:

R = fractional removal

C = outlet concentration

P = one or more pollutant names

R_P = one or more pollutant removals, (add R_ to the front of the pollutant name)

V = one or more 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 (ft2 or m2)
- 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

To add a treatment collection to a node:

Note: Make sure you set the Engine Type to Explicit (SWMM 5) in the Calculation Options for your project before computing results.

1. Add a node to your model or select an existing node, then display the Property Editor for the node (double-click the node or press **F4**).
2. In the SWMM Extended Data section of the Property Editor, set Apply Treatment? to **True**. The Treatment field becomes available.
3. Click the **Ellipses (...)** button in the Treatment field (where "<Collection: 0 items>" is displayed) to display the Treatment Collection dialog box.
4. The Treatment Collection dialog box displays each pollutant and its associated treatment expression as a row in a table. Click the **New** button to add a row to the table.
5. Click in the Pollutant field, then select an existing pollutant from the drop-down menu, or click the **Ellipses (...)** button to display the Pollutants dialog box, where you define pollutants in your model.
6. Type a treatment expression in the Treatment column.
7. Repeat Steps 4 - 6 for each pollutant you wish to add to the treatment collection.
8. Click **OK** to close the dialog box and add the collection to the node.

Treatment Collection Dialog Box

The Treatment Collection dialog box lets you add multiple pollutants and their associated treatment expressions to a node for the purpose of removing pollutants in your model. You access this dialog box from the SWMM Extended Data section of the Property Editor for the selected node.

The dialog box contains the following toolbar buttons:

	**New**	Adds a new row to the Pollutants table. Click in the row to select an existing pollutant from the drop-down menu, or click the **Ellipses (...)** button to define new pollutants.
	**Delete**	Deletes the current row from the table.
	**Report**	Lets you view a report of the collection.

The dialog box also contains the following controls:

Column	Description
Pollutant	Lets you add a pollutant to the collection. Click the down-arrow to select an existing pollutant from the drop-down menu, or click the **Ellipses (...)** button to define new pollutants.
Treatment	Lets you type a valid treatment expression to the pollutant.

Initial Buildup Collection Dialog Box

The Initial Buildup Collection dialog box lets you specify the amount of pollutant buildup existing over a catchment at the start of the simulation.

You access this dialog box from the SWMM Extended Data section of the Property Editor for the selected catchment.

The dialog box contains the following toolbar buttons:

	**New**	Adds a new row to the Initial Buildup table. Click in the row to select an existing pollutant from the drop-down menu, or click the **Ellipses (...)** button to define new pollutants.
	**Delete**	Deletes the current row from the table.
	**Report**	Lets you view a report of the collection.

The dialog box also contains the following table:

Column	Description
Pollutant	Lets you add a pollutant to the collection. Click the down-arrow to select an existing pollutant from the drop-down menu, or click the **Ellipses (...)** button to define new pollutants.

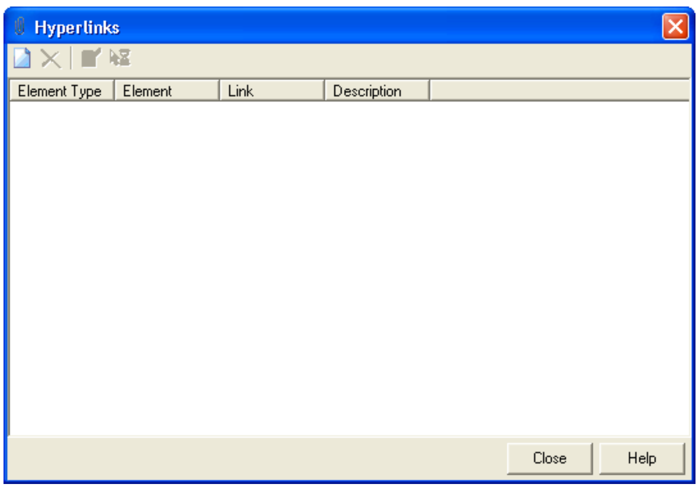
Initial Buildup	Lets you type an initial buildup value for the pollutant in the table row. If no buildup value is supplied for a pollutant, it is assumed to be 0.

Adding Hyperlinks to Elements

The Hyperlinks feature lets you associate external files, such as pictures or movie files, with elements. You can perform the following operations with hyperlinks:

- “Adding a Hyperlink” on page 6-326
- “Deleting a Hyperlink” on page 6-327

To use hyperlinks, select **Tools > Hyperlink**. The Hyperlink dialog box opens.



The hyperlink tool enables the user to associate a photo, word processing document, spreadsheet or other file with a given model element. Opening the hyperlink opens the file using its associated program (Picture Manager, Word, Excel, etc.).

The hyperlink can also be opened from the Property grid by picking the Hyperlink property from the grid and clicking the ellipse button which will open the hyperlink tool.

If a model file is moved to a different computer, the hyperlink will no longer work unless the associated file is moved to a comparable path on the same computer.

Hyperlinks Dialog Box

The Hyperlinks dialog contains a toolbar and a tabular view of all your existing hyperlinks.

The toolbar contains the following buttons:

	**New**	Lets you create a new hyperlink. Launches the Add Hyperlink dialog box.
	**Delete**	Deletes the currently highlight hyperlink.
	**Edit**	Lets you edit the currently highlighted hyperlink. Launches the Edit Hyperlink dialog box.
	**Launch**	Launches the external file associated with the currently highlighted hyperlink.

The table contains the following columns:

Column	Description
Element Type	Displays the element type of the element associated with the hyperlink.
Element	Displays the label of the element associated with the hyperlink.
Link	Displays the complete path of the hyperlink.
Description	Displays a description of the hyperlink, which you can optionally enter when you create or edit the hyperlink.

Adding a Hyperlink

To add a hyperlink:

Note: You can add more than one associated file to an element using the hyperlink feature, but you must add the associations one at a time.

1. Select **Tools** > **Hyperlink**. The Hyperlink dialog box opens.
2. Click **Add** to add a hyperlink. The Add Hyperlink dialog box opens.
3. Select the element to which you want to associate an external file.
4. Browse to the external file you want to use. This might be something like a picture of the element or a movie about the element.

Add Hyperlink Dialog Box

You create new hyperlinks in the Add Hyperlink dialog box. The dialog box contains the following controls:

Element Type	Lets you select an element type from the drop-down list.
Element	Lets you select an element from a drop-down list of specific elements from your model. Only those element types selected in the Element Type drop-down list are displayed.
Link	Lets you enter the complete path of the external file you want to associate with the selected element. You can type the path yourself or click the **Ellipsis (...)** button to search your computer for the file. Once you have selected the file, you can test the hyperlink by clicking the **Launch** button. 
Description	Lets you type a description of the hyperlink.

Editing a Hyperlink

You can edit existing hyperlinks using the Edit Hyperlink dialog box.

To edit a hyperlink:

1. Select **Tools** > **Hyperlink**. The Hyperlink dialog box opens.
2. Select the hyperlink you want to edit.
3. Click **Edit** to modify a hyperlink. The Edit Hyperlink dialog box opens.
4. Select the element you want to edit.
5. Edit the hyperlink by adding or deleting an associated file.

Edit Hyperlink Dialog Box

You edit existing hyperlinks in the Edit Hyperlink dialog box. The dialog box contains the following controls:

Link	Lets you edit the complete path of the external file associated with the selected hyperlink. You can type the path yourself or click the **Ellipsis (...)** button to search your computer for the file. Once you have selected the file, you can test the hyperlink by clicking the **Launch** button. 
Description	Lets you edit an existing description of the hyperlink or type a new description.

Deleting a Hyperlink

To delete a hyperlink:

1. Select **Tools** > **Hyperlink**. The Hyperlink dialog box opens.
2. Select the hyperlink you want to edit.
3. Click **Edit** to modify a hyperlink. The Edit Hyperlink dialog box opens.
4. Select the element you want to delete.
5. Click **Delete**.

Using Queries

A query in Bentley SewerGEMS V8*i* is a user-defined SQL expression that applies to a single element type. You use the Queries Manager to create and store queries; you use the Query Builder dialog box to construct the actual SQL expression.

You can create the following types of queries:

- Project queries—Queries you define that are available only in the Bentley SewerGEMS V8*i* project in which you define them.
- Shared queries—Queries you define that are available in all Bentley SewerGEMS V8*i* projects you create. You can edit shared queries.
- Predefined queries—Factory-defined queries included with Bentley SewerGEMS V8*i* that are available in all projects you create. You cannot edit predefined queries.

You can also use queries in the following ways:

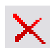
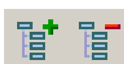
- Create dynamic selection sets based on one or more queries. For more information, see "Creating a Selection Set from a Query" on page 6-277.
- Filter the data in a FlexTable using a query. For more information, see "Sorting and Filtering FlexTable Data" on page 10-566.

For more information on how to construct queries, see "Creating Queries" on page 6-330.

Queries Manager

The Queries Manager is a docking manager that displays all queries in the current project, including predefined, shared, and project queries. You can create, edit, or delete shared and project queries from within the Queries Manager, as well as use it to select all elements in your model that are part of the selected query.

The Queries Manager consists of a toolbar and a tree view, which displays all of the queries that are associated with the current project. The toolbar contains the following buttons:

	**New**	Contains the following commands: • **Query**—Lets you create a new SQL expression as either a project or shared query, depending on which item is highlighted in the tree view. • **Folder**—Creates a folder in the tree view, allowing you to group queries. You can right-click a folder and create queries or folders in that folder.
	**Delete**	Deletes the currently-highlighted query or folder from the tree view. When you delete a folder, you also delete all of its contents (the queries it contains).
	**Rename**	Lets you rename the query or folder that is currently highlighted in the tree view.
	**Edit**	Opens the Query Builder dialog box, allowing you to edit the SQL expression that makes up the currently-highlighted query.
	**Expand All and Collapse All**	Expands or collapses the named views and folders.
	**Select in Drawing**	Lets you quickly select all the elements in the drawing pane that are part of the currently highlighted query. Once you have selected the elements in a selection set using Select In Drawing, you can delete them all at once.
	**Help**	Displays online help for the Queries Manager.

Query Parameters Dialog Box

Some predefined queries require that a parameter be defined. When one of these queries is selected, the Query Parameters dialog box will open, allowing you to type the parameter value that will be used in the query. For example, when the Pipe Split Candidates query is used the Query Parameters dialog will open, allowing the Tolerance parameter to be defined.

Creating Queries

A query is a valid SQL expression that you construct in the Query Builder dialog box. You create and manage queries in the Queries Manager. You also use queries to filter FlexTables and as the basis for a selection set.

To create a query from the Queries Manager:

1. Open the Queries Manager by selecting **View** > **Queries**, clicking the **Queries** button on the View toolbar, or by pressing **CTRL+5**.

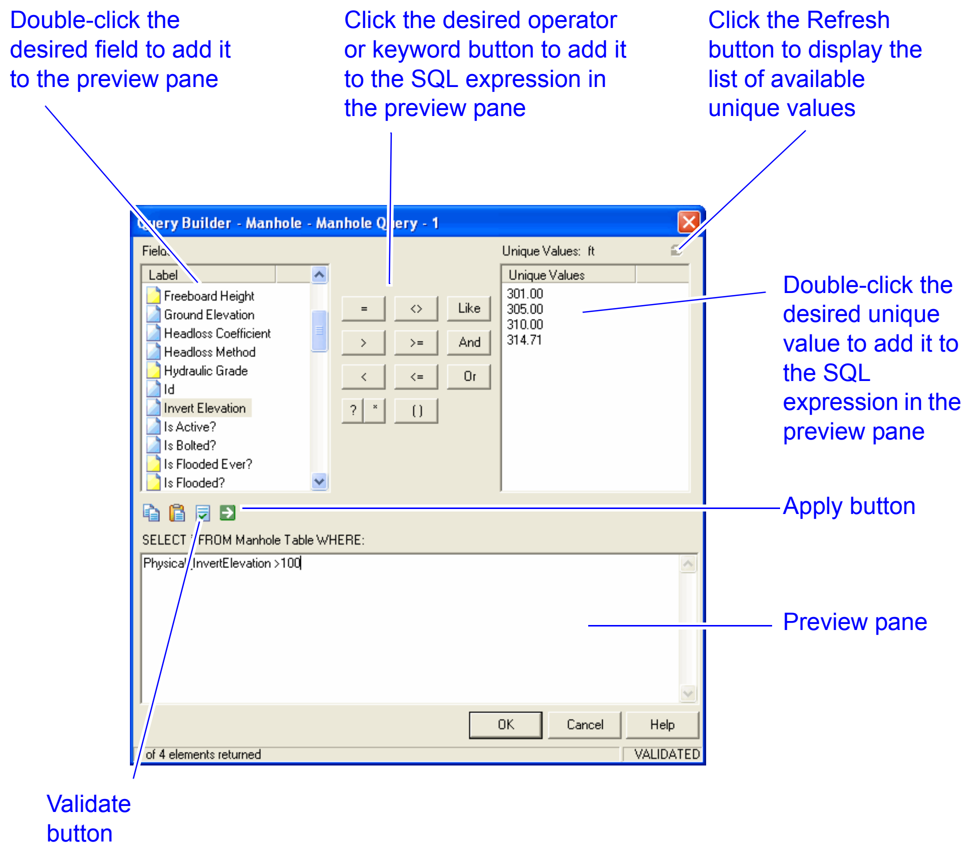
2. Perform one of the following steps:
   – To create a new project query, highlight **Queries - Project** in the list pane, then click the **New** button and select **Query**.
   – To create a new shared query, highlight **Queries - Shared** in the list pane, then click the **New** button and select **Query**.

Note: You can also right-click an existing item or folder in the list pane and select New > Query from the shortcut menu.

3. In the Select Element Type dialog box, select the desired element type from the drop-down menu. The Query Builder dialog box appears.
4. All input and results fields for the selected element type appear in the Fields list pane, available SQL operators and keywords are represented by buttons, and available values for the selected field are listed in the Unique Values list pane. Perform the following steps to construct your query:
   a. Double-click the field you wish to include in your query. The database column name of the selected field appears in the preview pane.
   b. Click the desired operator or keyword button. The SQL operator or keyword is added to the SQL expression in the preview pane.
   c. Click the **Refresh** button above the Unique Values list pane to see a list of unique values available for the selected field. Note that the Refresh button is disabled after you use it for a particular field (because the unique values do not change in a single query-building session).
   d. Double-click the unique value you want to add to the query. The value is added to the SQL expression in the preview pane.

Note: You can also manually edit the expression in the preview pane.

   e. Click the **Validate** button above the preview pane to validate your SQL expression. If the expression is valid, the word “VALIDATED” is displayed in the lower right corner of the dialog box.
   f. Click the **Apply** button above the preview pane to execute the query. If you didn’t validate the expression, the Apply button validates it before executing it.
   g. Click **OK**.



5. Perform these optional steps in the Queries Manager:
   - To create a new folder in the tree view, highlight the existing item or folder in which to place the new folder, then click the **New** button and select **Folder**. You can create queries and folders within folders.
   - To delete an existing query or folder, click the **Delete** button. When you delete a folder, you also delete all of its contents (the queries it contains).
   - To rename an existing query or folder, click the **Rename** button, then type a new name.
   - To edit the SQL expression in a query, select the query in the list pane, then click the **Edit** button. The Query Builder dialog box appears.
   - To quickly select all the elements in the drawing pane that are part of the currently highlighted query, click the **Select in Drawing** button.

Query Builder Dialog Box

You construct the SQL expression that makes up your query in the Query Builder dialog box. The Query Builder dialog box is accessible from the Queries Manager and from within a FlexTable.

The top part of the dialog box contains all the controls you need to construct your query: a list pane displaying all available attributes for the selected element type, a SQL control panel containing available SQL keywords and operators, and list view that displays all the available values for the selected attribute. The bottom part of the dialog box contains a preview pane that displays your SQL expression as you construct it.

All the dialog box controls are described in the following table.

	Fields	Lists all input and results fields applicable to the selected element type. This list displays the labels of the fields, while the underlying database column names of the fields become visible in the preview pane when you add them to the expression. Double-click a field to add it to your SQL expression.
	SQL Controls	These buttons represent all the SQL operators and controls that you can use in your query. They include =, >, <, _, **?**, *****, <>, >=, <=, **[ ]**, **Like**, **And**, and **Or**. Click the appropriate button to add the operator or keyword to the end of your SQL expression, which is displayed in the preview pane.
	Unique Values	When you click the **Refresh** button, this list displays all the available unique values selected field. Double-click a value in the list to add it to the end of your SQL expression, which is displayed in the preview pane. If you select a different field, you must click the **Refresh** button again to update the list of unique values for the selected field. When you first open the Query Builder dialog box, this list is empty.
	**Refresh**	Updates the list of unique values for the selected field. This button is disabled after you use it for a particular field.
	**Copy**	Copies the entire SQL expression displayed in the preview pane to the Windows clipboard.

	**Paste**	Pastes the contents of the Windows clipboard into the preview pane at the location of the text cursor. For example, if your cursor is at the end of the SQL expression in the preview pane and you click the **Paste** button, the contents of your clipboard will be added to the end of the expression.
	**Validate**	Validates the SQL expression in the preview pane. If the expression is not valid, a message appears. When you click this button and your SQL expression passes validation, the word “VALIDATED” appears in the lower right corner of the dialog box.
	**Apply**	Executes the query. The results of the query are displayed at bottom of the Query Builder dialog box in the form “*x* of *x* elements returned.”
	Preview Pane	Displays the SQL expression as you add fields, operators and/keywords, and values to it.

Using the Like Operator

The Like operator compares a string expression to a pattern in an SQL expression.

Syntax

expression Like “*pattern*”

The Like operator syntax has these parts:

Part	Description
expression	SQL expression used in a WHERE clause .
pattern	String or character string literal against which *expression* is compared.

You can use the Like operator to find values in a field that match the pattern you specify. For *pattern*, you can specify the complete value (for example, `Like “Smith”`), or you can use wildcard characters to find a range of values (for example, `Like “Sm*”`).

In an expression, you can use the Like operator to compare a field value to a string expression. For example, if you enter Like `"C*"` in an SQL query, the query returns all field values beginning with the letter C. In a parameter query , you can prompt the user for a pattern to search for.

The following example returns data that begins with the letter P followed by any letter between A and F and three digits:

```
Like "P[A-F]###"
```

The following table shows how you can use Like to test expressions for different patterns.

Kind of match	Pattern	Match (returns True)	No match (returns False)
Multiple characters	a*a	aa, aBa, aBBBa	aBC
	ab	abc, AABB, Xab	aZb, bac
Special character	a[*]a	a*a	aaa
Multiple characters	ab*	abcdefg, abc	cab, aab
Single character	a?a	aaa, a3a, aBa	aBBBa
Single digit	a#a	a0a, a1a, a2a	aaa, a10a
Range of characters	[a-z]	f, p, j	2, &
Outside a range	[!a-z]	9, &, %	b, a
Not a digit	[!0-9]	A, a, &, ~	0, 1, 9
Combined	a[!b-m]#	An9, az0, a99	abc, aj0

User Data Extensions

User data extensions are a set of one or more attribute fields that you can define to hold data to be stored in the model. User data extensions allow you to add your own data fields to your project. For example, you can add a field for keeping track of the date of installation for an element, or the type of area serviced by a particular element.

Note: The user data does not affect the hydraulic model calculations. However, their behavior concerning capabilities like editing, annotating, sorting and database connections is identical to any of the standard pre-defined attributes.

User data extensions exhibit the same characteristics as the predefined data used in and produced by the model calculations. This means that user data extensions can be imported or exported through database and shapefile connections, viewed and edited in the Property Editor or in FlexTables, included in tabular reports or element detailed reports, annotated in the drawing, color coded, and reported in the detailed element reports.

Note: The terms "user data extension" and "field" are used interchangeably here. In the context of the User Data Extension feature, these terms mean the same thing.

You define user data extensions in the User Data Extensions dialog box.

To define a user data extension:

1. Select **Tools > User Data Extensions**.
2. In the list pane on the left, select the element type for which you want to define a new attribute field.
3. Click the **New** button to create a new user data extension. A user data extension with a default name appears under the element type. You can rename the new field if you wish.
4. In the Property Editor for the new field, enter the following:
   – Type the name of the new field. This is the unique identifier for the field. The name field in the Property Editor is the name of the column in the data source.
   – Type the label for the new field. This is the label that will appear next to the field for the user data extension in the Property Editor for the selected element type. This is also the column heading if the data extension is selected to appear in a FlexTable.
   – Click the **Ellipses (...)** button in the Category field, then use the drop-down menu in the Select Category dialog box to select an existing category in which the new field will appear in the Property Editor. To create a new category, simply type the category name in the field.
   – Type a number in the Field Order Index field. This is the display order of fields within a particular category in the Property Editor. This order also controls the order of columns in Alternative tables. An entry of 0 means the new field will be displayed first within the specified category.
   – Type a description for the field. This description will appear at the bottom of the Property Editor when the field is selected for an element in your model. You can use this field as a reminder about the purpose of the field.

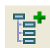
- Select an alternative from the drop-down menu in the Alternative field. This is the alternative that you want to extend with the new field.
- Select a data type from the drop-down menu in the Data Type field.
  - If you select **Enumerated**, an Ellipses (...) button appears in the Default Value field. Enumerated user data extensions are fields that present multiple choices.
- Enter the default value for the new field. If the data type is **Enumerated**, click the **Ellipses (...)** button to display the Enumeration Editor dialog box, where you define enumerated members.

5. Perform the following optional steps:
   - To import an existing User Data Extension XML File, click the **Import** button, then select the file you want to import. User Data Extension XML Files contain the file name extension .xml or .udx.xml.
   - To export existing user data extensions, click the **Export to XML** button, then type the name of the udx.xml file. All user data extensions for all element types defined in the current project are exported.
   - To share the new field among two or more element types, select the user data extension in the list pane, then click the **Sharing** button or right-click and select **Sharing**. In the Shared Field Specification dialog box, select the check box next to the element or elements that will share the user data extension. The icon next to the user data extension changes to indicate that it is a shared field. For more information, see "Sharing User Data Extensions Among Element Types" on page 6-342.
   - To delete an existing user data extension, select the user data extension you want to delete in the list pane, then click the **Delete** button, or right-click and select **Delete**.
   - To rename a the display label of an existing user data extension, select the user data extension in the list pane, click the **Rename** button or right-click and select **Rename**, then type the new display label.
   - To expand the list of elements and view all user data extensions, click the **Expand All** button.
   - To collapse the list of elements so that no user data extensions are displayed, click the **Collapse All** button.
6. Click **OK** to close the dialog box and save your user data extensions. The new field(s) you created will appear in the Property Editor for every instance of the specified element type in your model.

User Data Extensions Dialog Box

The User Data Extensions dialog box displays a summary of the user data extensions associated with the current project. The dialog box contains a toolbar, a list pane displaying all available Bentley SewerGEMS V8*i* element types, and a property editor. The toolbar contains the following controls:

	Import	Lets you merge the user data extensions in a saved User Data Extension XML file (.udx.xml or .xml) into the current project. Importing a User Data Extension XML file will not remove any of the other data extensions defined in your project. User data extensions that have the same name as those already defined in your project will not be imported.
	Export to XML	Lets you save existing user data extensions for all element types in your model to a User Data Extension XML file (.udx.xml) for use in a different project.
	New	Lets you create a new user data extension for the currently highlighted element type.
	Sharing	Lets you share the current user data extension with another element type. When you click this button, the Shared Field Specification dialog box opens. For more information, see “Sharing User Data Extensions Among Element Types” on page 6-342.

	**Delete**	Deletes the currently highlighted user data extension
	**Rename**	Lets you rename the display label of the currently highlighted user data extension.
	**Expand All**	Expands all of the branches in the hierarchy displayed in the list pane.
	**Collapse All**	Collapses all of the branches in the hierarchy displayed in the list pane.

The Property Editor contains the following controls:

	**Alphabetized**	Displays the attribute fields in the Property Editor in alphabetical order.
	**Categorized**	Displays the attribute fields in the Property Editor in categories. This is the default.

and the following fields, which define your new user data extension:

Table 6-7:

Attribute	Description
General	
Name	The unique identifier for the field. The name field in the Property Editor is the name of the column in the data source.
Label	The label that will appear next to the field for the user data extension in the Property Editor for the selected element type. This is also the column heading if the data extension is selected to appear in a FlexTable.
Category	The section in the Property Editor for the selected element type in which the new field will appear. You can create a new category or use an existing category. For example, you can create a new field for manholes and display it in the Physical section of that element's Property Editor.

Table 6-7:

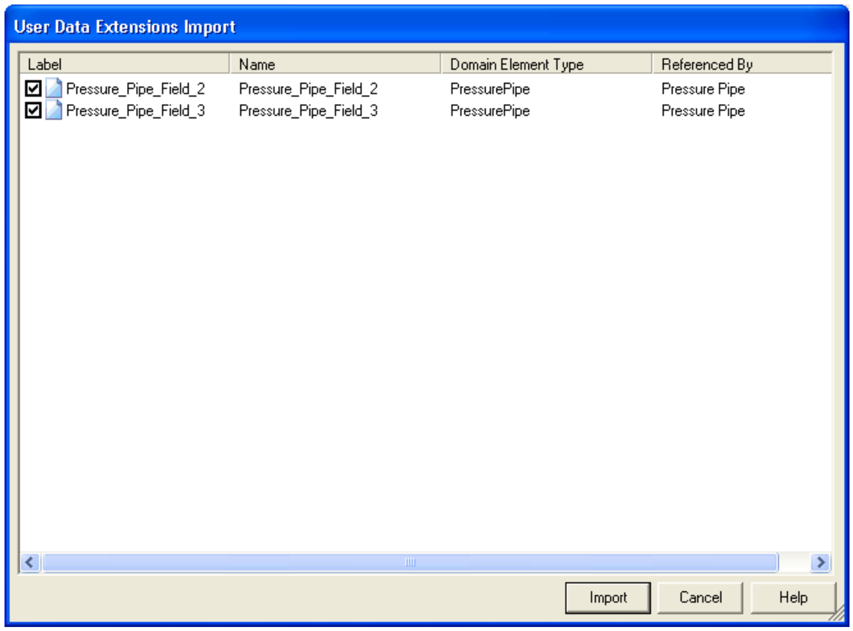
Attribute	Description
Field Order Index	The display order of fields within a particular category in the Property Editor. This order also controls the order of columns in Alternative tables. An entry of 0 means the new field will be displayed first within the specified category.
Field Description	The description of the field. This description will appear at the bottom of the Property Editor when the field is selected for an element in your model. You can use this field as a reminder about the purpose of the field.
Alternative	Lets you select an existing alternative to extend with the new field.
Referenced By	Displays all the element types that are using the field. For example, if you create a field called "Installation Date" and you set it up to be shared, this field will show the element types that share this field. So for example, if you set up a field to be shared by manholes and catch basins, the Referenced By field would show "Manhole, Catch Basin".
Units	
Data Type	Lets you specify the data type for the user data extension. Click the down arrow in the field then select one of the following data types from the drop-down menu: • **Integer**—Any positive or negative whole number. • **Real**—Any fractional decimal number (for example, 3.14). It can also be unitized with the provided options. • **Text**—Any string (text) value up to 255 characters long. • **Long Text**—Any string (text) up to 65,526 characters long. • **Date/Time**—The current date. The current date appears by default in the format month/day/year. Click the down arrow to change the default date. • **Boolean**—True or False. • **Enumerated**—When you select this data type, an Ellipses button appears in the Default Value field. Click the **Ellipses (...)** button to display the Enumeration Editor dialog box, where you can add enumerated members and their associated values. For more information, see "Enumeration Editor Dialog Box" on page 6-344.

Table 6-7:

Attribute	Description
Default Value	The default value for the user data extension. The default value must consistent with the selected data type. If you chose Enumerated as the data type, click the **Ellipses (...)** button to display the Enumeration Editor.
Dimension	Lets you specify the unit type. Click the drop-down arrow in the field to see a list of all available dimensions. This field is available only when you select **Real** as the Data Type.
Storage Unit	Lets you specify the storage units for the field. Click the drop-down arrow in the field to see a list of all available units; the units listed change depending on the Dimension you select. This field is available only when you select **Real** as the Data Type.
Numeric Formatter	Lets you select a number format for the field. Click the drop-down arrow in the field to see a list of all available number formats; the number formats listed change depending on the Dimension you select. For example, if you select **Flow** as the Dimension, you can select Flow, Flow - Pressurized Condition, Flow Tolerance, or Unit Load as the Numeric Formatter. This field is available only when you select **Real** as the Data Type.

User Data Extensions Import Dialog Box

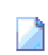
The Import dialog box opens after you initiate an Import command and choose the xml file to be imported. The Import dialog displays all of the domain elements contained within the selected xml file. Uncheck the boxes next to a domain element to ignore them during import.



Sharing User Data Extensions Among Element Types

You can share user data extensions across multiple element types in Bentley Sewer-GEMS V8*i*. Shared user data extensions are displayed in the Property Editor for all elements types that share that field.

The icons displayed next to the user data extensions in the User Data Extensions dialog box change depending on the status of the field:

- Indicates a new unsaved user data extension. 
- Indicates a user data extension that has been saved to the data source. 
- Indicates a user data extension that is shared among multiple element types but has not been applied to the data source. 
- Indicates a user data extension that is shared among multiple element types and that has been applied to the data source. Fields with this icon appear in the Property Editor for any elements of the associated element types that appear in your model. 

Observe the following rules when sharing user data extensions:

- You can select any number of element types with which to share the field. The list is limited to element types that support the Alternative defined for the Field. For example, the Physical Alternative may only apply to five of the element types. In this case, you will only see these five items listed in the Alternative drop-down menu.
- You cannot use the sharing feature to move a field from one element type to another. Validation is in place to ensure that only one item is selected and if it is the same as the original, default selection. If it is not, a message appears telling you that when sharing a field, you must select at least two element types, or select the original element type.
- To unshare a field that is shared among multiple element types, right-click the user data extension you want to keep in the list pane, then select **Sharing**. Clear all the element types that do not want to share the field with and click **OK**. If you leave only one element type checked in the Shared Field Specification dialog box, it must be the original element type for which you created the user data extension.
  – The fields that were located under the catch basin and conduit element type root nodes will be removed completely.
  – You can also unshare a field by using the **Delete** button or right-clicking and selecting **Delete**. This will unshare and delete the field.

To share a user data extension:

1. Open the User Data Extensions dialog box by selecting **Tools > User Data Extensions**.
2. In the list pane, create a new user data extension to share or select an existing user data extension you want to share, then click the **Sharing** button.
3. In the Shared Field Specification dialog box, select the check box next to each element type that will share the user data extension.
4. Click **OK**.
5. The icon next to the user data extension in the list pane changes to indicate that it is a shared field.

Shared Field Specification Dialog Box

You select element types to share a user data extension in the Shared Field Specification dialog box. The dialog box contains a list of all possible element types with check boxes.

Select element types to share the current user data extension by selecting the check box next to the element type. Clearing a selection if you no longer want that element type to share the current field.

Enumeration Editor Dialog Box

The Enumeration Editor dialog box appears when you select **Enumerated** as the Data Type for a user data extension, then click the **Ellipses (...)** button in the Default Value field. Enumerated fields are fields that contain multiple selections - you define these as members in the Enumeration Editor dialog box.

For example, suppose you want to identify conduits in a model of a new subdivision by one of the following states: Existing, Proposed, Abandoned, Removed, and Retired. You can define a new user data extension with the label "Pipe Status" for conduits, and select Enumerated as the data type. Click the **Ellipses (...)** button in the Default Value field in the Property Editor for the user data extension to display the Enumeration Editor dialog box. Then enter five members with unique labels (one member for each unique pipe status) and enumeration values in the table. After you close the User Data Extensions dialog box, the new field and its members will be available in the Property Editor for all conduits in your model. You will be able to select any of the statuses defined as members in the new Pipe Status field.

You can specify an unlimited number of members for each user data extension, but member labels and values must be unique. If they are not unique, an error message appears when you try to close the dialog box.

The dialog box contains a table and the following controls:

- **New—**Lets you add a new row to the table. Each row in the table represents a unique enumerated member of the current user data extension.
- **Delete—**Deletes the current row from the table. The enumerated member defined in that row is deleted from the user data extension.

You define enumerated members in the table, which contains the following columns:

- **Enumeration Member Display Label—**The label of the member. This is the label you will see in Bentley SewerGEMS V8*i* where ever the user data extension appears (Property Editor, FlexTables, etc.).
- **Enumeration Value—**A unique integer index associated with the member label. Bentley SewerGEMS V8*i* uses this number when it performs operations such as queries.

External Tools

Use the External Tool Manager to manage custom menu commands, which are then located in the Tools menu for quick accessibility.

Click Tools>External Tools to create a custom menu command from any executable file. Executable file types include:

- .exe
- .com
- .pif
- .bat
- .cmd

The External Tool Manager consists of the following elements:

- **External Tool List Pane**—This pane lists the external tools that have been created. All of the tools listed in this pane will be displayed in the Tools > External Tools menu.
- **New**—Creates a new external tool in the list pane.
- **Delete**—Deletes the currently highlighted tool.
- **Rename**—Allows you to rename the currently highlighted tool.
- **Command**—This field allows you to enter the full path to the executable file that the tool will initiate. Click the ellipsis button to open a Windows Open dialog to allow you to browse to the executable.
- **Arguments**—This optional field allows you to enter command line variables that are passed to the tool or command when it is activated. Click the > button to open a submenu containing predefined arguments. Arguments containing spaces must be enclosed in quotes. The available arguments are:
  – **Project Directory**—This argument passes the current project directory to the executable upon activation of the tool. The argument string is %(ProjDir).
  – **Project File Name**—This argument passes the current project file name to the executable upon activation of the tool. The argument string is %(ProjFileName).
  – **Project Store File Name**—This argument passes the current project datastore file name to the executable upon activation of the tool. The argument string is %(ProjStoreFileName).
  – **Working Directory**—This argument passes the current working directory to the executable upon activation of the tool. The argument string is %(ProjWorkDir).

- **Initial Directory**—Specifies the initial or working directory of the tool or command. Click the > button to open a submenu containing predefined directory variables. The available variables are:
  – **Project Directory**—This variable specifies the current project directory as the Initial Directory. The variable string is %(ProjDir).
  – **Working Directory**—This variable specifies the current working directory as the Initial Directory. The variable string is %(ProjWorkDir).

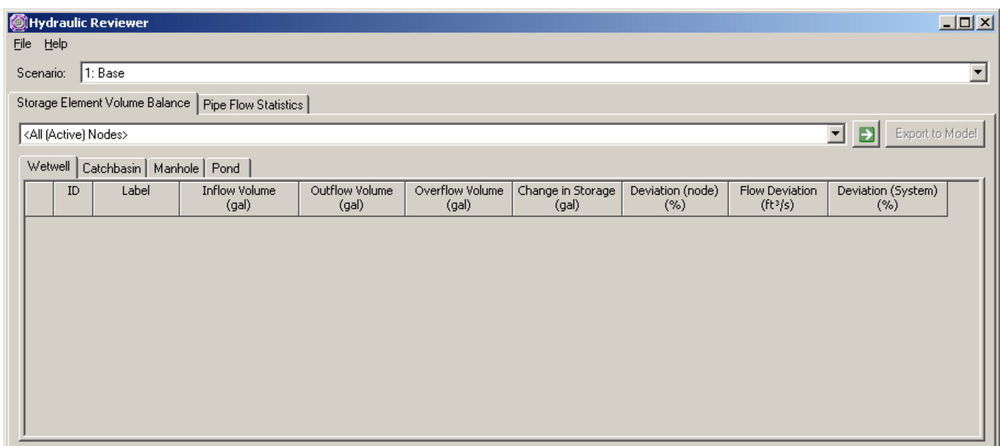
Test—This button executes the external tool using the specified settings.

Hydraulic Reviewer Tool

This version of SewerGEMS V8*i* includes a hydraulic reviewer tool to quickly assess the convergence and stability of the model. To begin using the tool click

Tools > Hydraulic Reviewer.

This opens a dialog which initially looks like this.



The user first selects the name of the scenario for which the hydraulic reviewer will be performed using a drop down list of scenarios. The calculation of the scenario must already have been run and the output file should not have been deleted.

Once the scenario has been selected, the user chooses between two tabs

Storage Element Volume Balance

The Storage Element Volume Balance tab determines the overall balance of flows at any node which can have storage which can include

- Wet Wells

- Catch basins
- Manholes
- Ponds

The user can also use a drop down list of any previously created selection set of node elements on which to perform the review. The default is All Active Nodes.

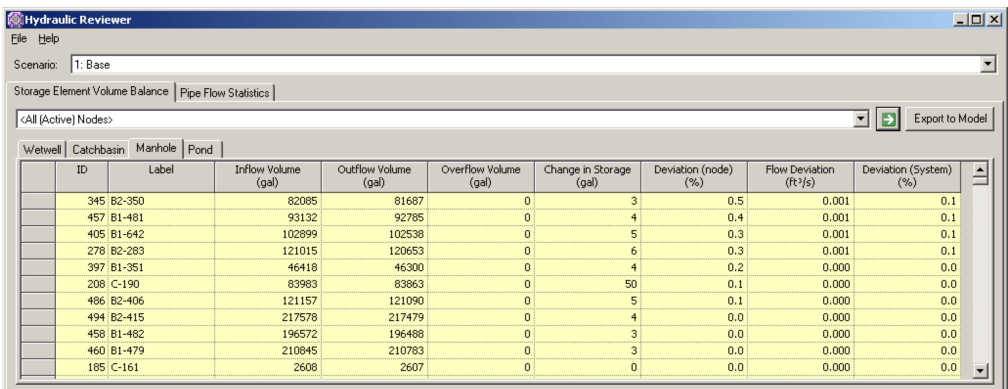
Picking the green Go arrow starts the calculation which for each node determines the inflow, outflow and overflow volumes over the course of the runs and the percent deviation from perfect flow balance as

Flow Deviation (node) = 100% (In - out - over - change in storage)/Duration

Deviation (node) = 100% (In - out - over - change in storage)/In

Deviation (system) = 100% (In - out - over - change in storage)/Total System Inflow

The results are presented in decreasing order based on Error. Any column can be sorted, filtered or have the display format changed as with any other flex table.



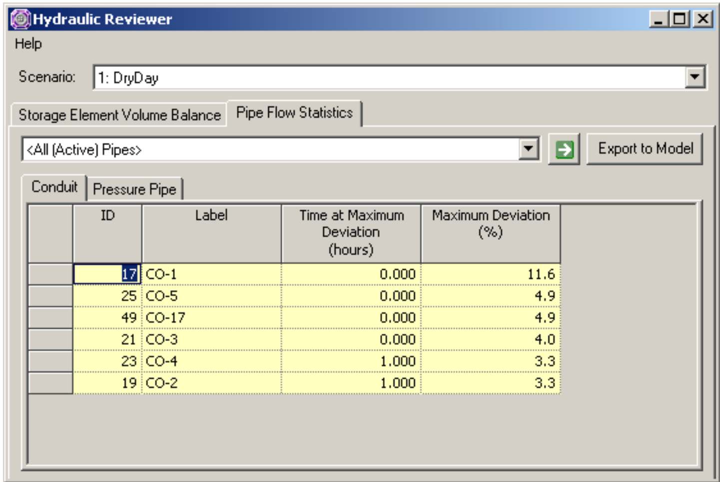
ID	Label	Inflow Volume (gal)	Outflow Volume (gal)	Overflow Volume (gal)	Change in Storage (gal)	Deviation (node) (%)	Flow Deviation (ft³/s)	Deviation (System) (%)
345	B2-350	82085	81687	0	3	0.5	0.001	0.1
457	B1-481	93132	92785	0	4	0.4	0.001	0.1
405	B1-642	102899	102538	0	5	0.3	0.001	0.1
278	B2-283	121015	120653	0	6	0.3	0.001	0.1
397	B1-351	46418	46300	0	4	0.2	0.000	0.0
208	C-190	83983	83863	0	50	0.1	0.000	0.0
486	B2-406	121157	121090	0	5	0.1	0.000	0.0
494	B2-415	217578	217479	0	4	0.0	0.000	0.0
458	B1-482	196572	196488	0	3	0.0	0.000	0.0
460	B1-479	210845	210783	0	3	0.0	0.000	0.0
185	C-161	2608	2607	0	0	0.0	0.000	0.0

On some occasions the Deviation (node) may appear large but this is primarily due to the inflow volume being very small. Users may want to discount the importance of any errors at nodes where the inflow rates (and deviations) are on the order of 0.1 cfs or less.

Pipe Flow Statistics

The Pipe flow statistics tab provides an indication of the change of flow rates from one time step to the next.

To use this tab, the user can chose All Pipes (the default) or any selection set of pipes. The calculation is run by picking the green Go arrow. It will display a table like the one below, sorted in order of decreasing Maximum Deviation.



High values for Maximum Deviation do not necessarily indicate that the model contains errors or is unstable. Some pipe links have very large changes in flow from one time step to another such as in the case of a pump cycling on an off. In some cases, numerical models can overshoot the calculated flow when the flow rate changes abruptly. This effect usually dies out after one or two time steps. Nevertheless, the Maximum Deviation can serve as an indicator of locations with possible stability issues.

Once the calculations have been performed, the user can export the values to a previously created User Defined Property so that the values can be used in color coding, flex tables, etc. Click on Export to Model to reach the dialog below and select Export to actually export the numerical values.

If the user defined property doesn't exist, the user can create one by picking the ellipse button and following the instructions for user data extensions.

TRex Wizard

The TRex Wizard steps you through the process of automatically assigning elevations to specified nodes based on data from a Digital Elevation Model or a Digital Terrain Model.

Step 1: File Selection

The DEM, DTM, DDF, or SHP (contour shapefile) file, the SewerGEMS V8*i* model, and the features to which elevations will be assigned are specified.

- **Data Source Type—**This menu allows you to choose the type of file that contains the input data you will use.
- **File**—This field displays the path where the DXF, XML, or SHP file is located. Use the browse button to find and select the desired file.
- **Spatial Reference (ArcGIS Mode Only)**—Click the Ellipsis (...) next to this field to open the Spatial Reference Properties dialog box, allowing you to specify the spatial reference being used by the elevation data file.
- **Select Elevation Field**—Select the elevation unit.
- **X-Y Units**—This menu allows the selection of the measurement unit type associated with the X and Y coordinates of the elevation data file.
- **Z Units**—This menu allows the selection of the measurement unit type associated with the Z coordinates of the elevation data file.
- **Clip Dataset to Model—**In some cases, the data source contains elevation data for an area that exceeds the dimensions of the area being modeled. When this box is checked, TRex will calculate the model's bounding box, find the larger dimension (width or height), calculate the Buffering Percentage of that dimension, and increase both the width and height of the model bounding box by that amount. Then any data point that falls outside of the new bounding box will not be used to generate the elevation mesh. If this box isn't checked, all the source data points are used to generate the elevation mesh. Checking this box should result in faster calculation speed and use less memory.
- **Buffering Percentage**—This field is only active when the Clip Dataset to Model box is checked. The percentage entered here is the percentage of the larger dimension (width or height) of the model's bounding box that will be added to both the bounding box width and height to find the area within which the source data points will be used to build the elevation mesh.
- **Spatial Reference (ArcGIS Mode Only)—**Click the Ellipsis (...) next to this field to open the Spatial Reference Properties dialog box, allowing you to specify the spatial reference being used by the SewerGEMS V8*i* model file.
- **Also update inactive elements**—Check this box to include inactive elements in the elevation assignment operation. When this box is unchecked, elements that are marked Inactive will be ignored by TRex.
- **All**—When this button is selected, TRex will attempt to assign elevations to all nodes within the SewerGEMS V8*i* model.
- **Selection**—When this button is selected, TRex will attempt to assign elevations to all currently highlighted nodes.

- **Selection Set**—When this is selected, the Selection Set menu is activated. When the Selection Set button is selected, TRex will assign elevations to all nodes within the selection set that is specified in this menu.

Note: If the SewerGEMS V8*i* model (which may or may not have a spatial reference explicitly associated with it) is in a different spatial reference than the DEM/DTM (which does have a spatial reference explicitly associated with it), then the features of the model will be projected from the model's spatial reference to the spatial reference used by the DEM/DTM.

Step 2: Completing the TRex Wizard

The results of the elevation extraction process are displayed and the results can be applied to a new or existing physical alternative.

- **Results Preview Pane**—This tabular pane displays the elevations that were calculated by TRex. The table can be sorted by label by clicking the Label column heading and by elevation by clicking the Elevation column heading. You can filter the table by right-clicking a column in the table and selecting the **Filter...Custom** command. You can also right-click any of the values in the elevation column to change the display options.
- **Use Existing Alternative**—When this is selected, the results will be applied to the physical alternative that is selected in the Use Existing Alternative menu. This menu allows the selection of the physical alternative to which the results will be applied.
- **New Alternative** —When this is selected, the results will be applied to a new physical alternative. First, the currently active physical alternative will be duplicated, then the results generated by TRex will be applied to the newly created alternative. The name of this new alternative must be supplied in the New Alternative text field.
- **Parent Alternative**—Select an alternative to duplicate from the menu, or select <None> to create a new Base alternative.
- **Export Results**—This exports the results generated by TRex to a tab or comma-delimited text file (.TXT). These files can then be re-used by SewerGEMS V8*i* or imported into other programs.

- Click Finish when complete, or Cancel to close without making any changes.

Chapter 7 Loading

Loading

The word "loading" is used in Bentley SewerGEMS V8*i* to describe flow entering the sewer system. Depending on the type of system, available data and level of detail, there are numerous ways of loading Bentley SewerGEMS V8*i* models. Some of the distinctions relate to whether the system is a combined or sanitary system, whether the loads are existing with flow data or proposed loads with only land use descriptions, whether the flow refers to dry weather sanitary flows or wet weather flow.

Note: For more information on loading, see Chapter 6 "Dry weather wastewater flows" and Chapter 7 "Wet weather wastewater flows" in the book Wastewater Collection System Modeling and Design" by Haestad Press.

With the exception of known fixed flows, the loading to the model consists of a table of flow or pattern values vs. time. The generic word "collection" is used to describe inputs to Bentley SewerGEMS V8*i* that are not a single value but are some type of table. For example, you will see Inflow Collections, which are simply a table of inflow vs. time.

Methods for Entering Loads

There are several methods for entering loads into Bentley SewerGEMS V8*i*. In general, most of the methods described can be applied to any node type element (i.e. manhole, cross section, catch basin, pond, pressure junction but not outfall). Some such as stormwater loading must be applied to catchments and conduit infiltration can only be applied to conduits. They are summarized below and described in more detail later in this chapter.

- Inflow consists of data which may be:

- Known constant flow
- Hydrograph - flow vs. time
- Pattern Load - baseline flow times multiplier

In general, the hydrograph input is used for wet weather events while pattern loads are used for sanitary flows which repeat from one day to the next. For more information, see “Inflows” on page 7-377.

- Sanitary loading consists of data which may be
  - Know constant flow
  - Hydrograph - flow vs. time
  - Pattern Load - baseline flow times multiplier
  - Unit loads - number of units (e.g. houses) times unit load (e.g. flow/house/day)

  Sanitary loads are generally used to describe dry weather contribution to flow from domestic, commercial and industrial customers. For more information, see “Sanitary (Dry Weather) Flow Collections” on page 7-390.

- Load Builder consists of using the LoadBuilder model to place loads on nodes using ArcGIS functions. Unlike the methods above which are applicable when the loads are already known for each node, LoadBuilder is used when the loading data is not yet associated with individual nodes. For example, the data can be in the form of:
  - Customer water use billing data
  - Area-wide flow measurement
  - Load (e.g. population or land use assigned to polygons) times unit loading factors (e.g. flow/day/area)

  This method must be started from ArcMap or ArcCatalog. For more information, see “LoadBuilder” on page 7-393.

- Rainfall Derived Infiltration and Inflow in Sanitary Systems (RDII) lets you load sanitary systems with I/I flow based on flow monitoring data. This usually involves entering a precipitation event and comparing the predicted catchment outflow with the model results. You can adjust the catchment parameters to match the observed outflow using generic unit hydrographs or RTK method. For more information, see “Rainfall Derived Infiltration and Inflow (RDII)” on page 7-394.
- Stormwater flow can be used to model inflow into a collection system based on rainfall events and any number of hydrologic models including:
  - Generic unit hydrographs

- SCS unit hydrograph
- Modified rational method
- SWMM runoff method

Losses (i.e. precipitation not entering runoff) can be modeled using several methods including:

- SCS
- Green Ampt
- Horton
- fLoss

These methods can be used for modeling stormwater collection systems and combined sewer systems but not sanitary systems because they do not account for the defects which allow wet weather flow to enter sanitary systems. These methods can only be applied to catchment elements, not other node elements. For more information, see "Stormwater Flow" on page 7-394.

- Conduit infiltration can be used to model infiltration into pipes along the length of the pipe. This can be specified as:
  - Known unit flow based on Length, Area, Diameter-length, or Count.
  - Hydrograph
  - Pattern

  For more information, see "Pipeline Infiltration" on page 7-427.

Summary

In general:

- Dry weather load can be entered using Inflow, Sanitary Loading and LoadBuilder.
- Wet weather flow in sanitary systems can be entered using inflow, RDII or conduit infiltration.
- Wet weather flow in stormwater and combined systems can be entered using Inflow, Stormwater flow or Conduit infiltration.

Types of Loads

Within each of the loading methods available in Bentley SewerGEMS V8*i*, there are several ways to enter (add) data. For example, under the method Inflow for loading the model, there are three types of inflow - fixed, hydrograph and pattern load. These loading types may be used by several methods. For example, pattern loading is used by the Inflow, Sanitary and Pipeline Infiltration methods. The dialogs for each of these types are the same regardless of the method being used.

The following table illustrates which types of loads are available in each method.

Table 7-1: Types of Loads

Method	**Fixed**	**Hydrograph**	**Pattern Load**	**Unit Load**	**Unit Hydrograph**	**RTK Method**	**Modified Rational**	**SWMM**	**SCS**
Inflow	X	X	X						
Sanitary Load	X	X	X	X					
LoadBuilder *									
RDII Inflow					X	X			
Stormwater					X		X	X	X
Conduit infiltration *	X	X	X						

*LoadBuilder used GIS based calculations described in [hot link].

*Conduit infiltrations uses fixed flows on a per length or area basis.

- “Hydrograph vs. Pattern Loads” on page 7-358
- “Pattern Loads” on page 7-360
- “Unit Sanitary Loading” on page 7-367

Adding Fixed Loads

A fixed load can be entered as a:

- Known flow under sanitary loading
- Pattern load with the multiplier set to 1
- Hydrograph inflow with "fixed" specified

- “Hydrograph vs. Pattern Loads” on page 7-358
- “Pattern Loads” on page 7-360

- "Unit Sanitary Loading" on page 7-367

Hydrograph vs. Pattern Loads

Hydrographs and pattern loads are two distinct ways to describe how flow varies over time. Ultimately, you can attain the same results using either type but there are some behavioral and semantic differences that should be noted.

Pattern loads consists of a single average base load and a series of dimensionless multipliers used to delineate how the load varies over time. A hydrograph, simply, is a time-discharge series.

Hydrographs are usually applied as wet weather loads, and are generated using hydrologic methods, while patterns are more typically applied to sanitary loads. Pattern multipliers are usually developed based on flow monitoring data for the system under consideration. The multipliers are used to account for time-of-day variations in sewer loads. Usually a handful of patterns are developed (e.g. residential area, commercial area, large industry) and these patterns are assigned to the appropriate nodes. These statements represent typical usage of both loading types; they do not represent hard and fast rules.

During an extended period simulation if the duration of the simulation exceeds the duration of a pattern then the pattern will repeat itself. If the duration of the simulation exceeds the duration of a hydrograph the last point of the hydrograph will remain constant for the extent of the remaining time.

The following figure shows the difference between a hydrograph and pattern load.

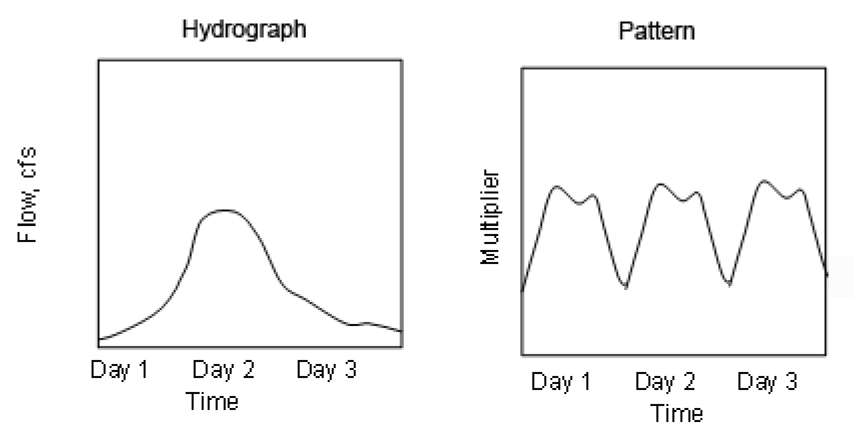


Figure 7-1: Difference between a Hydrograph and Pattern Load

- "Pattern Loads" on page 7-360
- "Unit Sanitary Loading" on page 7-367

Adding User Defined Hydrographs

You can directly associate a user-defined unit hydrograph to any node element (e.g. manhole, catchment, cross section, pressure junction) for runoff calculations.

To add a user defined hydrograph to a catchment:

1. Click a node in your model to display the Property Editor, or right-click a node and select **Properties** from the shortcut menu.
2. In the Runoff section of the Property Editor, select **User Defined Hydrograph** as the Runoff Method.
3. Click the **Ellipses (...)** button next to the Runoff Hydrograph field. The User Defined Hydrograph dialog box appears.
4. Click the **New** button to add a row to the Time vs. Flow table.
5. Enter Time vs. Flow data into the table. Press **Enter** after typing a value to add a new row to the table (or click the **New** button to add a new row).

   To insert a row at a time in between two other times, simply insert at the bottom of the table. When you close the table then reopen it, the row will be in the correct position.

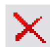
   Note: Time and flow units must be consistent with time and flow units used throughout the model.

6. Click the **Graph** button to view a plot of the Time vs. Flow data.
7. Click **OK** to close the dialog box and add the hydrograph to the Property Editor for the node.

User Defined Hydrograph Dialog Box

This dialog box allows you to define Time vs. Flow unit hydrographs.

The dialog box contains the time-vs.-flow table and the following buttons:

	**New**	Creates a new row in the hydrograph table.
	**Delete**	Deletes the currently highlighted row from the hydrograph table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the unit hydrograph defined by the points in the table.

The table contains the following columns:

Column	Description
Time	Lets you define the hour of the hydrograph point.
Flow	Lets you define the flow for the hydrograph point.

- “Hydrograph vs. Pattern Loads” on page 7-358
- “Pattern Loads” on page 7-360
- “Unit Sanitary Loading” on page 7-367

Pattern Loads

A pattern load consists of a base flow and a pattern, which is a set of multipliers used to adjust base flow over the course of a day (or some other period). Patterns can also be used with unit loads by assigning a pattern setup for a particular scenario.

For more information on building patterns, see “Defining Patterns” on page 7-362. For more information on pattern setups, see “Defining Pattern Setups” on page 7-365.

Related Topics

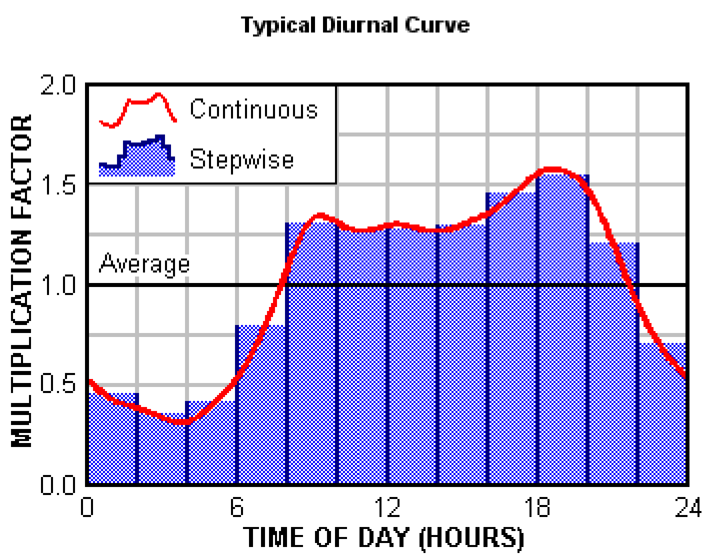
- “Adding Fixed Loads” on page 7-357
- “Hydrograph vs. Pattern Loads” on page 7-358

- "Adding User Defined Hydrographs" on page 7-359
- "Unit Sanitary Loading" on page 7-367

Working with Patterns

A dynamic analysis is actually a series of Steady State analyses run against time-variable loads such as sewer inflows, demands, or chemical constituents. Patterns allow you to apply automatic time-variable changes within the system. The most common application of patterns is for residential or industrial loads. Diurnal curves are patterns that relate to the changes in loads over the course of the day, reflecting times when people are using more or less water than average. Most patterns are based on a multiplication factor versus time relationship, whereby a multiplication factor of one represents the base value (which is often the average value).

Using a representative diurnal curve for a residence as illustrated below, we see that there is a peak in the diurnal curve in the morning as people take showers and prepare breakfast, another slight peak around noon, and a third peak in the evening as people arrive home from work and prepare dinner. Throughout the night, the pattern reflects the relative inactivity of the system, with very low flows compared to the average.



Note: This curve is conceptual and should not be construed as representative of any particular network.

There are two basic forms for representing a pattern: stepwise and continuous. A stepwise pattern is one that assumes a constant level of usage over a period of time, and then jumps instantaneously to another level where it remains steady until the next jump. A continuous pattern is one for which several points in the pattern are known and sections in between are transitional, resulting in a smoother pattern. For the continuous pattern in the figure above, the multiplication factor and slope at the start time and end times are the same. This is a continuity that is recommended for patterns that repeat.

Because of the finite time steps used for calculations, this software converts continuous patterns into stepwise patterns for use by the algorithms. In other words for a time step a multiplier is interpolated from the pattern curve. That multiplier is then used for the duration of the time step, until a new multiplier is selected for the next time step.

Patterns provide a convenient way to define the time variable aspects of system loads.

Click one of the following links to learn more about working with patterns:

- “Defining Patterns” on page 7-362
- “Defining Pattern Setups” on page 7-365

Defining Patterns

A pattern is a series of time step values, each having an associated multiplier value. During a dynamic analysis each time step of the simulation uses the multiplier from the pattern corresponding to that time. If the duration of the simulation is longer than the pattern, the pattern is repeated. The selected multiplier is applied to any baseline load that is associated with the pattern.

Patterns must begin and end with the same multiplier value. This is because patterns will be repeated if the duration of the dynamic analysis is longer than the pattern duration. In other words, the last point in the pattern is really the start point of the pattern's next cycle.

A dynamic analysis is actually a series of Steady State analyses for which the boundary conditions of the current time step are calculated from the conditions at the previous time step. This software will automatically convert a continuous pattern format to a stepwise format so that the demands and source concentrations remain constant during a time step.

An individual node can support multiple hydraulic demands. Furthermore, each load can be assigned any hydraulic load pattern. This powerful functionality makes it easy to combine two or more types of load patterns (such as residential and institutional) at a single loading node.

To define a pattern:

1. Select **Analysis > Patterns** or click the **Patterns** button on the Analysis toolbar.
2. In the Patterns dialog box, click the **New** button to create a new pattern.

3. On the right side of the dialog box, enter values for Start Time and Starting Multiplier.
4. Select **Stepwise** or **Continuous** in the Pattern Type field.
5. In the time step points table, enter values for Time from Start and Multiplier.
   – The Time from Start for row 1 in the table (the first point in the pattern) must be greater than zero.
   – The last point in the pattern must have the same multiplier as the starting multiplier. This is how the pattern duration is defined, and it ensures an infinitely repeating pattern.

Note: Bentley SewerGEMS V8*i* validates your data as you enter it and displays errors and warnings in the status bar at the bottom of the Patterns dialog box. Be sure to check this status bar for any errors or warnings as you enter data.

6. Perform the following optional steps:
   – To delete an existing pattern, select the pattern label in the list pane, then click the **Delete** button.
   – To rename an existing pattern, select the pattern label in the list pane, then click the **Rename** button and type the new name of the pattern.
   – To view a report on an existing pattern, select the pattern label in the list pane, then click the **Report** button.
   – To view a plot of a pattern, select the pattern label in the list pane, then click the **Graph** button above the time step points table.
   – To delete a row from the time step points table, select the row then click the **Delete** button above the table.
7. Click **Close**.

Patterns Dialog Box

You create, edit, and delete patterns in the Patterns dialog box. The dialog box contains a list pane on the left and several input fields and a table on the right:

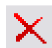
The dialog box contains the following controls above the list pane:

	**New**	Creates a new pattern in the list pane.
	**Delete**	Deletes the currently highlighted pattern.
	**Rename**	Lets you rename the currently highlighted pattern.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted pattern.

The following fields and controls appear on the right side of the dialog box:

Start Time	The first time step in the pattern.
Starting Multiplier	The multiplier value of the first time step point in your pattern. Any real number can be used for this multiplier (it does not have to be 1.0).
Pattern Type	Lets you select the type of pattern you want to create: • **Stepwise**—The multiplier values are considered to be the average value for the interval between the specified time and the next time. Patterns using this format will have a "staircase" appearance. Multipliers are set at the specified time and held constant until the next point in the pattern. • **Continuous**—The multipliers are considered to be the instantaneous values at a particular time. Patterns using this format will have a "curvilinear" appearance. Multipliers are set at the specified time, and are linearly increased or decreased to the next point in the pattern.

The following controls are located above the time step points table on the right:

	**New**	Creates a new row in the time step points table.
	**Delete**	Deletes the currently highlighted row from the time step points table.
	**Graph**	Displays a graph of a pattern that represents the multiplier variable of the pattern over time.

The time step points table contains the following columns:

Column	Description
Time from Start	Lets you specify the amount of time from the Start Time of the pattern to the time step point being defined.
Multiplier	Lets you specify the multiplier value associated with the time step point.

There is also a status bar located at the bottom of the dialog box that displays any errors and warnings that may occur when you enter data.

Related Topic

- “Working with Patterns” on page 7-361
- “Defining Pattern Setups” on page 7-365

Defining Pattern Setups

A pattern setup allows you to match unit sanitary (dry weather) loads with appropriate loading patterns. A pattern setup is associated with each scenario as specified in the Calculation Options Manager. Each scenario can use a different pattern setup, thus allowing you to model different loading alternatives for different extended period simulations.

Note: You must have at least one unit sanitary (dry weather) load set up in your model and at least one pattern defined before you can define a pattern setup.

To define a pattern setup:

1. Define at least one unit sanitary (dry weather) load in your current project. For more information, see “Adding Unit Sanitary (Dry Weather) Loads” on page 7-370.
2. Define at least one pattern in your current project. For more information, see “Defining Patterns” on page 7-362.
3. Select **Analysis** > **Pattern Setups** or click the **Pattern Setups** button on the Analysis toolbar.
4. In the Pattern Setups dialog box, click the **New** button to create a new pattern setup.
5. The table on the right side of the dialog box displays all of the unit sanitary (dry weather) loads currently associated with your current project. For each unit load in the table, select an existing pattern from the Setup Pattern submenu.
6. Perform the following optional steps:
   – To delete an existing pattern setup, select the pattern setup label in the list pane, then click the **Delete** button.
   – To rename an existing pattern setup, select the pattern setup label in the list pane, then click the **Rename** button and type the new name of the pattern setup.
   – To view a report on an existing pattern setup, select the pattern setup label in the list pane, then click the **Report** button.
7. Click **Close**.

Pattern Setups Dialog Box

The Pattern Setups dialog box lets you define a list of pattern setups. The dialog box contains a list pane on the left and a table on the right:

The dialog box contains the following controls above the list pane:

	**New**	Creates a new pattern setup in the list pane.
	**Delete**	Deletes the currently highlighted pattern setup.
	**Rename**	Lets you rename the currently highlighted pattern setup.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted pattern setup.

The right side of the dialog contains a table with the following fields:

Column	**Description**
Unit Load	Each row displays a unit sanitary (dry weather) load in the current project.
Set Pattern	Lets you select an existing pattern to apply to the unit load.

Related Topics

- “Adding Fixed Loads” on page 7-357
- “Hydrograph vs. Pattern Loads” on page 7-358
- “Adding User Defined Hydrographs” on page 7-359
- “Working with Patterns” on page 7-361
- “Defining Patterns” on page 7-362
- “Unit Sanitary Loading” on page 7-367

Unit Sanitary Loading

A unit loading consists of a unit (person, building, area) multiplied by a unit load (gal/capita/day, litres/sq m/day, cfs/acre). The units are assigned to individual nodes elements (manholes, pressure junctions) while the unit loads are created using the Unit Sanitary (Dry Weather) Loads dialog box.

Unit loads are calculated as: Load (flow Units) = Unit load (flow/number) x number of Loading units. For example: 100 gallons/capita/day x 40 people = 4000 gallons per day.

If the unit loads are not assigned to nodes but to polygons in a GIS, then it is best to use LoadBuilder to import the loads. For more information, see “LoadBuilder” on page 7-393.

For more information, see “Adding Unit Sanitary (Dry Weather) Loads” on page 7-370 and “Types of Unit Sanitary (Dry Weather) Loads” on page 7-368.

Unit loads correspond to a baseline load and time of day patterns can be assigned to scenarios. For more information, see “Defining Pattern Setups” on page 7-365.

Default unit load information is not stored with the project but with a library that can be shared between projects. Default values are provided in the library called "HMI Unit Sanitary (Dry Weather) Loads.xml"

Related Topics

- “Adding Fixed Loads” on page 7-357
- “Hydrograph vs. Pattern Loads” on page 7-358
- “Adding User Defined Hydrographs” on page 7-359
- “Pattern Loads” on page 7-360

Types of Unit Sanitary (Dry Weather) Loads

The following types of unit sanitary loads are supported in Bentley SewerGEMS V8*i*:

Population-based

The most common way of specifying sanitary loads to a sewer system is to make them proportional to the contributing population. Population-based unit sanitary loads define loads as a function of adjusted contributing population. You can select the population loading units that will be used and the unit load per population unit. For example, the unit sanitary load, Home (Average), specifies Resident as the population loading unit, and 280 l/d per Resident as the unit load per population unit.

Non-population-based

Non-population-based unit sanitary loads can be area-based (function of contributing area), discharge-based (function of direct discharge), or count-based (function of a user-defined count).

Area-based

Area-based unit sanitary loads are commonly used to specify industrial loads and steady inflows. Use these unit sanitary loads whenever your load is specified as a function of contributing area. For example, you may use "area residential" (in hectare) as a property of each node and 400 L/day/hectare as the unit loading.

Discharge-based

Discharge-based unit sanitary loads are used to directly specify loads without specifying them on the basis of some other count, such as population or area.

There are two general ways to use discharge based loads:

- Specify 1.0 discharge unit (e.g. l/day, gpd, cfs, etc.) as the unit load. Then, when using the load, specify the total desired load for the loading unit count. For example, you can create a load called Liter per Day whose loading unit type is Discharge, loading unit is l/day, and unit load is 1.0. When you use this load at a manhole, a wet well, or a pressure junction, you specify 50.0 as the loading unit count. This yields a base load of 50 l/day.
- Specify total desired load as the unit load. Then, when using the load, only specify 1.0 as the loading unit count. For example, you can create a load called Industry XYZ whose loading unit type is Discharge, loading unit is l/day, and unit load is 2000.0. When you use this load at the manhole, wet well, or pressure junction, you would specify 1.0 as the loading unit count. This yields a base load of 2000 l/day.

In other words, you can specify a unit load of 1.0 in the Unit Sanitary Load Library and determine the total load at each node through the loading unit count, or you can specify the total load in the Unit Sanitary Load Library and then have a loading unit count of 1.0.

Count-based

Count-based unit sanitary loads should be used for any load that is not area, population, nor discharge-based. These loads allow you to specify any loading unit such as loading per vehicle, machine, or anything else.

Loading units in user-defined counts are treated only as labels. Conversion between these units is always 1 to 1.

Related Topic

- "Unit Sanitary Loading" on page 7-367

Adding Unit Sanitary (Dry Weather) Loads

Bentley SewerGEMS V8*i* defines unit sanitary loads in editable Engineering Libraries, allowing you to edit predefined unit sanitary loads and insert new ones. A unit sanitary load is used to specify loads to a sewer system for a user-selected loading unit. Unit sanitary loads can be either population-based or non-population-based. Population-based unit sanitary loads specify a load to the sewer system as a function of the contributing population. Non-population-based loads specify loads based on service area, discharge, or user-defined counts.

You add unit sanitary loads using the Unit Sanitary (Dry Weather) Loads dialog box or Bentley SewerGEMS V8*i*' Engineering Libraries.

To add a unit sanitary (dry weather) load:

1. Select **Component > Unit Sanitary (Dry Weather) Loads**, or click the **Unit Sanitary (Dry Weather) Loads** button on the Analysis toolbar.
2. In the Unit Sanitary (Dry Weather) Loads dialog box, click the **New** button, then select the type of unit sanitary load you want to create from the submenu (Area, Count, Discharge, or Population).
3. On the Unit Sanitary Load tab, enter the following data:
   – For area-based loads, select the desired unit from the Area Unit drop-down menu.
   – For discharge-based loads, select the desired unit from the Discharge Units drop down menu.
   – For count-based loads, type the base unit used to define the count-based load in the Count Load Unit field. You can specify any unit you want, such as loading per vehicle, machine, or anything else.
   – For population-based loads, select the desired unit from the Population Units drop-down menu.
   – Type the amount of flow contributed per loading unit in the Unit Load field.

– Type the count of adjusted population per loading unit in the Population Equivalent field. For area based loads, this is essentially a population density, or population per unit area.

– Check the Report Adjusted Population check box to report the adjusted population with other populations. If you clear this check box, the adjusted population will be not be reported as part of the total population.

4. You can save your new load in Bentley SewerGEMS V8*i*’ Engineering Libraries for future use. To do this, perform these steps:

   a. Click the **Synchronization Options** button, then select **Export to Library**. The Engineering Libraries dialog box appears.

   b. Use the plus and minus signs to expand and collapse the list of available libraries, then select the library into which you want to export your new unit sanitary load.

   c. Click **Close** to close the Engineering Libraries dialog box.

5. Perform the following optional steps:

   – To delete a load, select the load label then click **Delete**.

   – To rename an load, select the load label you want to rename, click **Rename**, then type the new name for the load.

   – To view a report on a load, select the load label for which you want a report then click **Report**.

6. Click **Close** to close the Unit Sanitary (Dry Weather) Loads dialog box.

To add a unit sanitary load in the Engineering Library:

1. Select **Tools** > **Engineering Libraries** to display the Engineering Libraries dialog box.

2. Click the plus sign next to the Unit Sanitary (Dry Weather) Loads Library to expand the list of items (categories and folders) included in that library. This library includes a category entitled Unit Sanitary (Dry Weather) Loads.

   Note: You can add new items to a category or a folder, add new folders to categories, and add new categories to libraries. For more information, see “Engineering Libraries” on page 6-288**.**

3. Right-click the Unit Sanitary (Dry Weather) Loads category (or a different category or folder) and select **New Item**.

4. Define the new unit sanitary load in the Editor pane on the right as described in the following steps:
    a. Type the unit load in the Unit Load field.
    b. Select the load type from the Loading Unit Type drop-down (Area Based, Count Based, Discharge Based, or Population Based).
    c. Select the load units from the Sanitary Unit Load Units drop-down. For count-based loads, you can specify any unit you want, such as loading per vehicle or machine.
    d. For area-, count-, and discharge-based loads, type the count of adjusted population per loading unit in the Pop.Equivalent (Capita) field. For area based loads, this is essentially a population density, or population per unit area.
    e. For area-, count-, and discharge-based loads, select True from the Report Adjusted Population drop down to report the adjusted population with other populations. Select False if you don't want to report the adjusted population as part of total population.
5. Click **Close**. Your new unit sanitary load is now part of the Engineering Libraries and can be re-used any time.

Unit Sanitary (Dry Weather) Load Dialog Box

This dialog box allows you to create unit sanitary (dry weather) loads. There are two sections: the list pane on the left and the tab section on the right. The list pane lets you create, edit, and delete unit sanitary loads and the tab section contains entry fields for each type of unit sanitary load.

The following controls are available in the Unit Sanitary (Dry Weather) Load dialog box:

<table>
<tr>
<td></td>
<td>New</td>
<td>Creates a new unit sanitary load that uses an automatically created label.

Area—Adds a new area-based unit sanitary load to the list pane. An area-based unit sanitary load is a function of contributing service area.
Count—Adds a new count-based unit sanitary load. A count-based unit sanitary load is a function of a user-defined count. Count-based unit sanitary loads should be used for any load that is not area-, discharge-, or population-based.
Discharge—Adds a new discharge-based unit sanitary load. A discharge-based unit sanitary load is a function of direct discharge.
Population—Adds a new population-based unit sanitary load. A population-based unit sanitary load is a function of adjusted contributing population.

</td>
</tr>
</table>

	**Delete**	Deletes the currently highlighted unit sanitary load.
	**Rename**	Lets you rename the currently highlighted unit sanitary load.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted unit sanitary load.
	**Synchronization Options**	Clicking this button opens a submenu containing the following commands: • **Browse Engineering Library**—Opens the Engineering Library manager dialog, allowing you to browse the Unit Sanitary (Dry Weather) Load Engineering Libraries. • **Synchronize From Library**—Lets you update a set of unit sanitary loads previously imported from a Unit Sanitary (Dry Weather) Load Engineering Library. The updates reflect changes that have been made to the library since it was imported. • **Synchronize To Library**—Lets you update an existing Unit Sanitary (Dry Weather) Load Engineering Library using current unit sanitary loads that were initially imported but have since been modified. • **Import From Library**—Lets you import a unit sanitary load from an existing Unit Sanitary (Dry Weather) Load Engineering Library. • **Export To Library**—Lets you export the current unit sanitary load to an existing Unit Sanitary (Dry Weather) Load Engineering Library.

The tab section includes the following controls:

Unit Sanitary Load Tab

Area Unit	Lets you specify the base unit used to define the area-based load.
Unit Load	Lets you specify the amount of flow contributed per loading unit.
Population Equivalent	Lets you specify the count of adjusted population per loading unit. For area based loads, this is essentially a population density, or population per unit area. The Population equivalent field is optional and simply converts area or count into equivalent popultion, based on the Population equivalent value.
Report Adjusted Population	If this option is toggled ON, the adjusted population will be reported with other populations. If the option is OFF, adjusted population will be not be reported as part of the total population.
Count Load Unit	Lets you specify the base unit used to define the count-based load. You can specify any unit you want, such as loading per vehicle, machine, or anything else.
Discharge Units	Lets you specify the base unit used to define the discharge-based load.
Population Units	Lets you specify the base unit used to define the unit load.
Library Tab	This tab displays information about the unit sanitary load that is currently highlighted in the list pane. If the load is derived from an engineering library, the synchronization details can be found here. If the load was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the pump was not derived from a library entry.

Composite Hydrographs

A composite hydrograph graphs the total flow over time from multiple defined fixed/unit loads, hydrographs, and pattern loads.

You can access the composite hydrograph and its corresponding data table from the Inflow Collection dialog boxand the Sanitary (Dry Weather) Flow Collection Editor dialog box, both of which are available from the Property Editor for selected elements. For example, a manhole has properties for Inflow Collection and Sanitary Loading.

This graph is dynamic and is generated automatically each time it is requested.

The time step in a composite hydrograph is determined by going from time 0 to the Total Simulation Time divided by Calculation Time Step. You can define the Total Simulation Time and Calculation Time Step values in the Calculation Options Manager. For more information, see "Calculation Options Manager" on page 8-431.

Composite Hydrograph Window

This window displays the composite hydrograph from multiple fixed/unit loads, hydrographs, and pattern loads defined in either the Inflow Collection dialog box or Sanitary (Dry Weather) Flow Collection Editor dialog box.

You access the Composite Hydrograph window by clicking the Graph button in the Inflow Collection dialog box or Sanitary (Dry Weather) Flow Collection Editor dialog box. If you have only one load or hydrograph defined, the graph displays the data for that single load or hydrograph.

The Composite Hydrograph window contains the following button:

	**Chart Settings**	Opens the Chart Options dialog box, allowing you to change graph display settings.

Composite Hydrograph Data Table Window

This window displays a table of all the data points in a composite hydrograph from multiple fixed/unit loads, hydrographs, and pattern loads defined in either the Inflow Collection dialog box or Sanitary (Dry Weather) Flow Collection Editor dialog box.

The data table displays the same data used in the composite hydrograph in numerical form. The table contains two columns: the first column displays the time steps and the second column displays the flows. The values in this table can not be edited.

You access the Data Table window by clicking the Data Table button in the Inflow Collection dialog box or Sanitary (Dry Weather) Flow Collection Editor dialog box. If you have only one load or hydrograph defined, the Data Table displays the data for that single load or hydrograph.

The Composite Hydrograph Data Table window contains the following buttons:

	**Copy**	Copies the contents of the data table to the Windows clipboard. You can then paste the data into a different application or document.
	**Paste**	Pastes the contents of the Windows clipboard into the data table. This is useful if you've got the data defined in a different document or application and want to simply copy and paste it into a composite hydrograph.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.

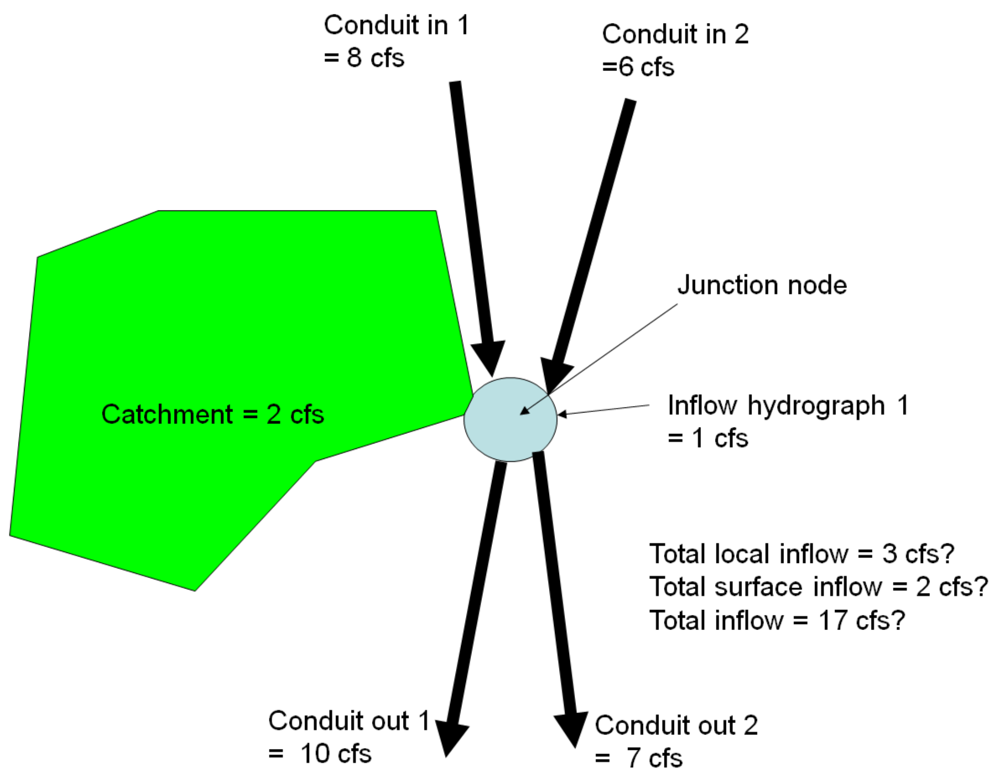
Inflows

The word "inflow" is used in two ways in sewer modeling. It is used first to describe wet weather flows to sewer systems that do not infiltrate through the ground [hot link - inflow and infiltration loading] and it is used in Bentley SewerGEMS V8*i* to describe any flow which enters a node element whether it is a fixed inflow, hydrograph or pattern load. The type of load available depends on the element type. The descriptions below, refer to the Bentley SewerGEMS V8*i* definition of inflow.

Inflows can be specified at any node element except a pond outlet and an outfall. Inflows are not a single value but are a collection (i.e. a table of flow vs. time) and as such must be specified from Property Editor for an element. For more information, see "Defining Inflow Collections" on page 7-378.

Inflow hydrographs are hydrographs with flow units and are not unit hydrographs. For more information, see "What is the Difference Between a User Defined Unit Hydrograph and a Hydrograph Entered in the Inflow Collection Editor?" on page 6-269.

The following diagram defines various flows.



Defining Inflow Collections

You can define an inflow collection for any node element in SewerGEMS V8*i*. An inflow collection can contain any combination of fixed, hydrograph, or pattern inflows.

To define an inflow collection:

1. Click a node element in your model to display the Property Editor, or right-click a node element and select **Properties** from the shortcut menu.
2. In the Inflow Collection section of the Property Editor, click the **Ellipses (...)** button. The Inflow Collection Editor appears.
3. Click the **New** button, then select the type of inflow you want to create from the submenu (Fixed Inflow, Hydrograph Inflow, Pattern Inflow).
4. For a Hydrograph Inflow, enter the data points in the hydrograph table. For Fixed and Pattern Inflows, enter the data in the appropriate fields.

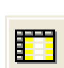
Note: For a hydrograph, if the last time in the table is less than the total simulation time, the simulation time and last flow will be appended to the hydrograph table.

5. Repeat steps 3 and 4 for each inflow you want to add to the collection.
6. Click the **Composite Hydrograph** button to see a graph of the composite hydrograph.
7. Click the **Composite Hydrograph Data Table** button to see a tabular view of all the data points in the composite hydrograph.
8. Click **OK** to close the dialog box and add the collection data to the Property Editor.

Inflow Collection Editor

This dialog box lets you define fixed loads, hydrograph loads, and pattern inflows for any node element. A node may have any combination of fixed loads, hydrograph loads, and pattern inflows.

The dialog box contains an Inflow list pane and the following controls:

	**New**	Opens a submenu containing the following options: • **Fixed Inflow**—Adds a new Fixed Load inflow to the Inflow list pane. • **Hydrograph Inflow**—Adds a new Hydrograph inflow to the Inflow list pane. •
	**Delete**	Lets you delete the currently highlighted load.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Composite Graph**	Opens a graph window plotting a curve of a selected single sanitary load or the composite hydrograph of all loads.
	**Composite Hydrograph**	Opens a window listing all the data points in the curve of a selected single sanitary load or in the composite hydrograph of all loads.

The Inflow Collection Editor also contains the following controls:

Hydrograph Table	Lets you define the hydrograph by entering Time vs. Flow points. This table is available only when a Hydrograph Load inflow is highlighted in the Inflow list pane. If the last time in the table is less than the total simulation time, the simulation time and last flow will be appended to the hydrograph table.
Fixed Load	Lets you define a fixed load value. This field is available only when a Fixed Load inflow is highlighted in the Inflow list pane. Set the fixed flow that affects the manhole.
Base Inflow	Lets you enter the average inflow over the duration of the simulation.
Inflow Pattern	Lets you select the pattern for the selected pattern inflow. Patterns are selected, edited, and created in the Pattern Manager, which you access by clicking the ellipsis (...) button next to this field.

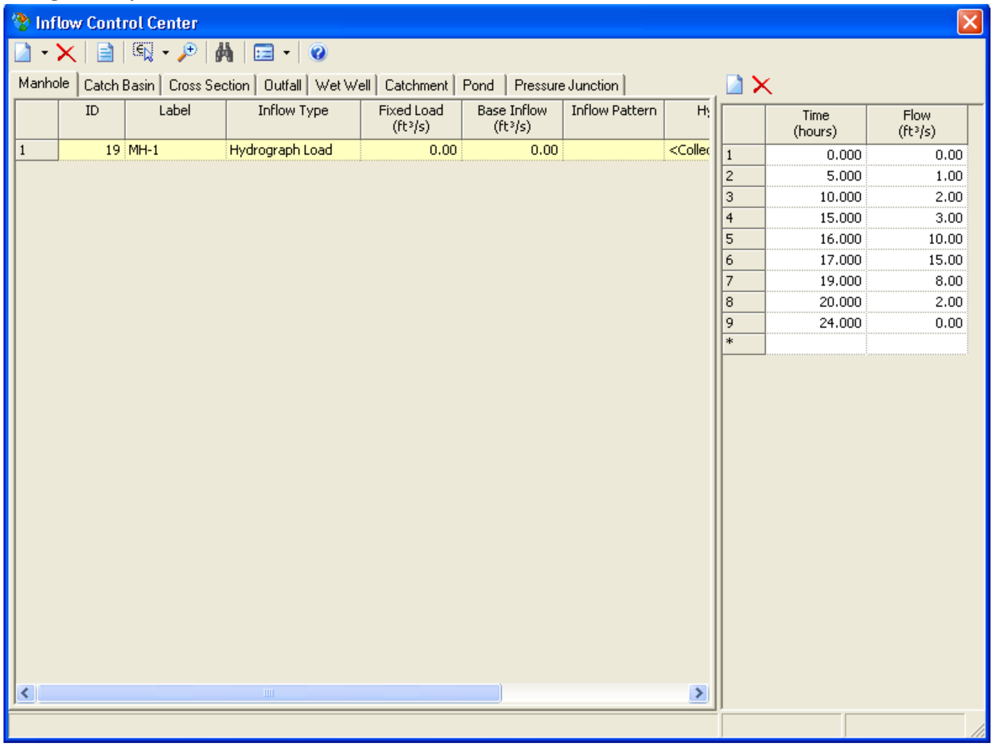
Inflow Control Center

The Inflow Control Center is an editor for manipulating all the inflows in your model. Using the Inflow Control Center, you can add new inflows, delete existing inflows, or modify the values for existing inflows using standard SQL select and update queries.

The Inflow Control Center provides demand editing capabilities which can:

- open on all inflow nodes, or subset of inflow nodes,
- sort and filter based on inflow criteria,

- add, edit, and delete individual inflows,
- globally edit inflows.



The Inflow Control Center consists of a pane consisting of tabs for each element type that list all of the inflows for all of the elements in the model and a pane that displays Hydrograph Load collections for the currently highlighted element.

It also contains the following controls:

New

Clicking the New button opens a submenu containing the following commands:

- Add Fixed Load to Element—Adds a new Fixed Load to the element currently selected in the list pane.
- Add Hydrograph Load to Element—Adds a new Hydrograph Load to the element currently selected in the list pane.
- Add Pattern Load to Element—Adds a new pattern load to the element currently selected in the list pane.
- Add Inflows—Return the view to the drawing pane, allowing you to select an element from the drawing. After an element has been selected, the Apply Inflow Type to Selection dialog opens, allowing you to enter a Fixed or Hydrograph Load to the element you selected.
- Initialize Fixed Loads for All Elements—Adds a Fixed Load to each element of the current type in the model that does not currently have an inflow defined. The loads added by this command have an initial value of 0.0.
- Initialize Hydrograph Loads for All Elements—Adds a Hydrograph Load to each element of the current type in the model that does not currently have an inflow defined. The hydrographs added by this command are initially blank.
- Initialize Pattern Load for All Elements—Adds a Pattern Load to each element of the current type in the model that does not currently have an inflow defined. The loads added by this command have an initial value of 0.0.
- Initialize Fixed Load for Selection—Adds a Fixed Load to each element that is currently selected in the list pane that does not currently have an inflow defined. The loads added by this command have an initial value of 0.0.
- Initialize Hydrograph Load for Selection—Adds a Hydrograph Load to each element that is currently selected in the list pane that does not currently have an inflow defined. The hydrographs added by this command are initially blank.
- Initialize Pattern Load for Selection—Adds a Pattern Load to each element that is currently selected in the list pane that does not currently have an inflow defined.The loads added by this command have an initial value of 0.0.

Note: **You can only use one type of Initialize operation in any given element. For example, if you perform an Initialize Fixed Loads for All Elements operation, you will not then be able to then perform an Initialize Hydrograph Loads for All Elements on the model.**

If you wish to add another inflow of a different type than the one that was Initialized, you must do so for each individual element.

Delete

Deletes the currently selected row from the list. Delete commands can not be undone.

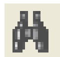
Report

Opens a report containing the inflow information displayed in each tab of the list pane.

Create or Add to a Selection Set

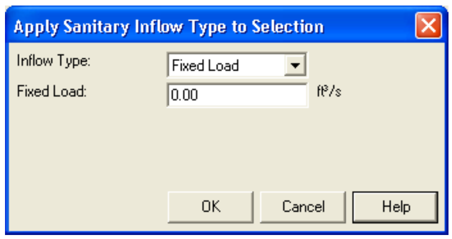
Clicking this button opens a submenu containing the following commands:

- Create Selection Set—Creates a new selection set consisting of the currently highlighted elements.
- Add to Selection Set—Adds the currently selected elements to an existing selection set.
- Remove from Selection Set—Removes the currently selected elements from an existing selection set.

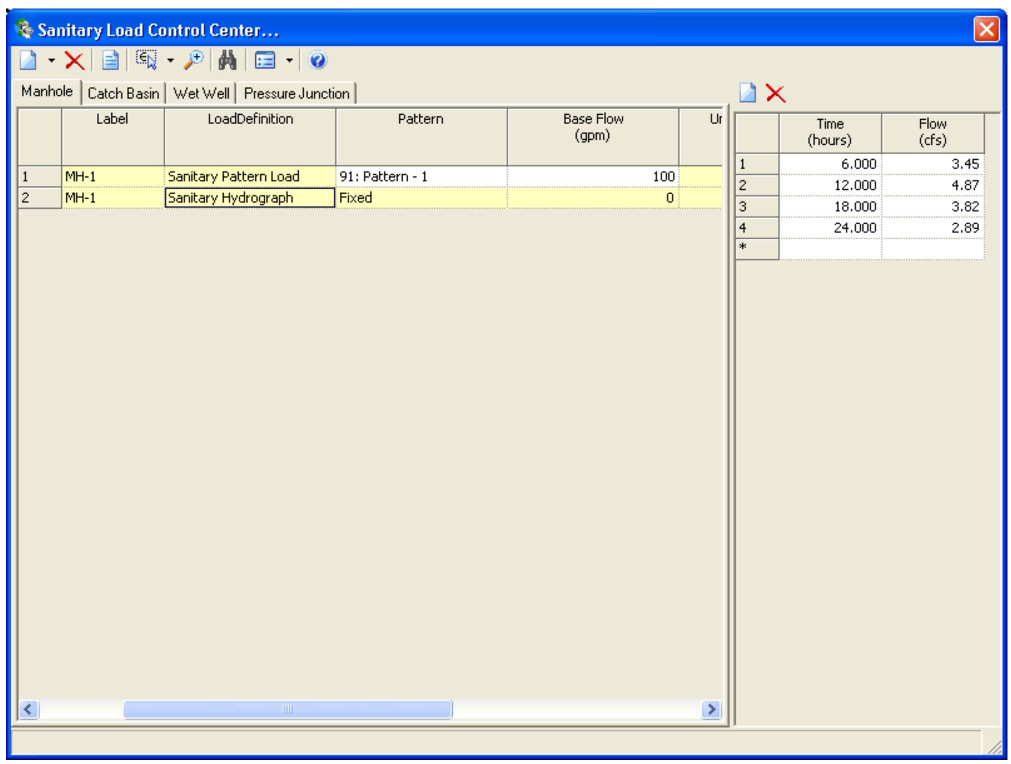
	**Zoom To**	Centers the drawing pane view on the currently selected element.
	**Find**	Opens the Find Element dialog, allowing you to search for a specific element.
	**Options**	Allows you to sort and/or filter the contents of the list pane.
	**Help**	Opens the online help.

Apply Sanitary Inflow Type to Selection Dialog

This dialog allows you to assign an inflow to the currently selected element or elements. The dialog appears after you have used the Add Inflows command in the Inflow Control Center. To add an inflow, choose the inflow type, then enter inflow data and click OK for a Fixed Inflow or just click OK for a Hydrograph Inflow. If a Hydrograph Inflow is selected, the hydrograph must then be defined in the hydrograph pane of the Inflow Control Center.



•



The Sanitary Load Control Center consists of a pane consisting of tabs for each element type that list all of the loads for all of the elements in the model and a pane that displays Hydrograph Load collections for the currently highlighted element, along with the following controls:

New

Clicking the New button opens a submenu containing the following commands:

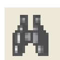
- Add Unit Load to Element—Adds a new Unit Load to the element currently selected in the list pane.
- Add Sanitary Hydrograph to Element—Adds a new sanitary hydrograph to the element currently selected in the list pane.
- Add Pattern Load to Element—Adds a new pattern load to the element currently selected in the list pane.
- Add Sanitary Loads—Return the view to the drawing pane, allowing you to select an element from the drawing. After an element has been selected, the Apply Sanitary Load to Selection dialog opens, allowing you to enter a Sanitary Hydrograph, Unit Load, or Pattern Load to the element you selected.
- Initialize Unit Loads for All Elements—Adds a Unit Load to each element of the current type in the model that does not currently have an inflow defined. The loads added by this command have an initial value of 0.0.
- Initialize Hydrograph Loads for All Elements—Adds a Hydrograph Load to each element of the current type in the model that does not currently have an inflow defined. The hydrographs added by this command are initially blank.
- Initialize Pattern Load for All Elements—Adds a Pattern Load to each element of the current type in the model that does not currently have an inflow defined. The loads added by this command have an initial value of 0.0.
- Initialize Unit Loads for Selection—Adds a Unit Load to each element that is currently selected in the list pane that does not currently have an inflow defined. The loads added by this command have an initial value of 0.0.
- Initialize Hydrograph Load for Selection—Adds a Hydrograph Load to each element that is currently selected in the list pane that does not currently have an inflow defined. The hydrographs added by this command are initially blank.
- Initialize Pattern Load for Selection—Adds a Pattern Load to each element that is currently selected in the list pane that does not currently have an inflow defined.The loads added by this command have an initial value of 0.0.

Note: **You can only use one type of Initialize operation for any given element. For example, if you perform an Initialize Hydrograph Loads for All Elements operation, you will not then be able to then perform an Initialize Pattern Load for All Elements on the model.**

If you wish to add another load of a different type than the one that was Initialized, you must do so for each individual element.

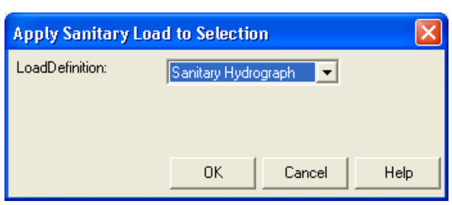
Delete

Deletes the currently selected row from the list. Delete commands can not be undone.

	**Report**	Opens a report containing the load information displayed in each tab of the list pane.
	**Create or Add to a Selection Set**	Clicking this button opens a submenu containing the following commands: • Create Selection Set—Creates a new selection set consisting of the currently highlighted elements. • Add to Selection Set—Adds the currently selected elements to an existing selection set. • Remove from Selection Set—Removes the currently selected elements from an existing selection set.
	**Zoom To**	Centers the drawing pane view on the currently selected element.
	**Find**	Opens the Find Element dialog, allowing you to search for a specific element.
	**Options**	Allows you to sort and/or filter the contents of the list pane.
	**Help**	Opens the online help.

Apply Sanitary Load to Selection Dialog

This dialog allows you to assign a sanitary load to the currently selected element or elements. The dialog appears after you have used the Add Sanitary Loads command in the Sanitary Load Control Center. To add a load, choose the load type, then enter load data and click OK for a Sanitary Unit Load or Sanitary Pattern Load. Click OK for a Sanitary Hydrograph. If a Sanitary Hydrograph is selected, the hydrograph must then be defined in the hydrograph pane of the Sanitary Load Control Center.



Defining CN Area Collections for Catchments

SewerGEMS V8*i* lets you define infiltration for a catchment based on CN (SCS Curve Number) and Area data that you specify. You define and save this data in a CN Area collection.

In the SCS and EPA-SWMM methods in SewerGEMS V8*i*, the sub-basin runoff is defined solely by the CN input for each watershed. The CN Area Collection dialog box automatically computes weighted CN values as a function of soil hydrologic class and cover characteristics, based on the CN Engineering Library entries that you select when you create a CN Area collection.

> **Note: The USDA has classified its soil types into four hydrologic soil groups. For a complete description of the CN values for various land uses and cover characteristics for each soil classification, see** "The Runoff Curve Number" on page 14-798**.**

To define a CN Area collection for a catchment:

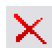
1. Click a catchment in your model to display the Property Editor, or right-click a catchment and select **Properties** from the shortcut menu.
2. In the Runoff section of the Property Editor, select **Unit Hydrograph** as the Runoff Method. The Loss Method field becomes available.
3. Select **SCS CN** as the Loss Method. the SCS CN field becomes available.
4. Click the **Ellipses (...)** button next to the SCS CN field. The CN Area Collection dialog box appears.
5. Type a description for the collection in the Description field, or click the **Ellipses (...)** button to display the CN Libraries in the Engineering Libraries.
6. Click the plus signs to expand the list of items in the CN Libraries until you find the CN Value for the soil hydrologic class and cover characteristics that you want to use.
7. Click **Select** to close the Engineering Libraries dialog box and add the CN Value to the table in the CN Area Collection dialog box.
8. SewerGEMS V8*i* automatically fills in the values for SCS CN and Area. Type values for Percent Connected Impervious Area and Percent Unconnected Impervious Area.

   > **Note: You can change the SCS CN value by clicking the Ellipses (...) button next to the SCS CN field, then selecting a different CN Value from the CN Engineering Libraries.**

9. Repeat Steps 5 - 8 for each item you want to add to the CN Area collection.
10. Click **OK**.

CN Area Collection Dialog Box

This dialog box lets you define infiltration based on Cn and Area data. The dialog box contains the Cn-Area table and the following buttons:

	**New**	Creates a new row in the cn-area table.
	**Delete**	Deletes the currently highlighted row from the cn-area table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.

The table contains the following columns:

Column	Description
Description	Lets you enter a description for the catchment.
CN	Lets you define the Cn value for the catchment.
Area	Lets you define the area for the catchment.
Percent Connected Impervious Area	This field allows you to enter the percent connected impervious area. This value represents the percentage of the area (for the current line of data) that contains directly connected impervious cover. Precipitation that falls on these types of impervious areas flow to the outfall point without ever flowing over pervious cover areas. An adjustment to the Cn for this line of data is made based on the % connected impervious areas (0% yields no adjustment). An example sequence of directly connected impervious areas are rainfall drops onto a concrete driveway, driveway drains to a paved gutter, gutter drains into a storm sewer, and storm sewer into a detention pond.

Column	Description
Percent Unconnected Impervious Area	Enter the percent unconnected impervious area. This value represents the percentage of the area (for the current line of data) that contains unconnected impervious cover. Precipitation falling on these types of impervious areas eventually flows over pervious areas before reaching the outfall. An adjustment to the Cn for this line of data is made based on the % unconnected impervious areas (0% yields no adjustment). An example sequence of directly unconnected impervious areas are rainfall drops onto a paved tennis court at a grassed park, water sheet flowing across a paved tennis court, water flowing off the tennis court onto a grass field in a park (there are no inlet drains on the court), and some of the water then being absorbed into the soil.

Sanitary (Dry Weather) Flow Collections

You can define a sanitary (dry weather) flow collection for any node element in your model. A sanitary flow collection can contain any combination of hydrograph, unit, or pattern loads.

To define a sanitary (dry weather) flow collection:

1. Click an element in your model to display the Property Editor, or right-click an element and select **Properties** from the shortcut menu.
2. In the Sanitary Loads section of the Property Editor, click the **Ellipses (...)** button. The Sanitary (Dry Weather) Flow Collection Editor appears.
3. Click the **New** button, then select the type of sanitary load you want to create from the submenu (Hydrograph - Flow vs. Time, Unit Load - Unit Type and Count, or Pattern Load - Base Flow and Pattern).
4. For a Hydrograph, enter the data points in the hydrograph table. For Unit and Pattern Loads, enter the data in the appropriate fields.

   Note: For a hydrograph, if the last time in the table is less than the total simulation time, the simulation time and last flow will be appended to the hydrograph table.

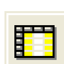
5. Repeat steps 3 and 4 for each load you want to add to the collection.

6. Click the **Composite Hydrograph** button to see a graph of the composite hydrograph.
7. Click the **Composite Hydrograph Data Table** button to see a tabular view of all the data points in the composite hydrograph.
8. Click **OK** to close the dialog box and add the collection data to the Property Editor.

Sanitary (Dry Weather) Flow Collection Editor

This dialog box lets you define collections of sanitary (dry weather) loads for the selected element in your model. Sanitary loads correspond to loads produced by residential, commercial, recreational, and industrial activity. A sanitary load represents the base load to the sewer system.

The dialog box contains a list pane and the following controls:

	**New**	Opens a submenu containing the following options: • **Hydrograph - Flow vs. Time**—Adds a new hydrograph load to the list pane. This is a flow vs. time distribution. • **Unit Load - Unit Type and Count**—Adds a new unit load to the list pane. Unit Type refers to the type of Unit Load and Count refers to the number of units associated with the unit load. For example, 5000 passengers at an airport. • **Pattern Load - Base Flow and Pattern**—Adds a new pattern load to the list pane. A pattern load is a direct, known sanitary load with a set pattern.
	**Delete**	Lets you delete the currently highlighted load.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Composite Graph**	Opens a graph window plotting a curve of a selected single sanitary load or the composite hydrograph of all loads.
	**Composite Hydrograph**	Opens a window listing all the data points in the curve of a selected single sanitary load or in the composite hydrograph of all loads.

Depending on the type of sanitary load you select in the list pane, the following controls appear:

Hydrograph Table	Lets you define the hydrograph by entering Time From Start vs. Sanitary Flow points. This table is available only when a hydrograph load is highlighted in the list pane. If the last time in the table is less than the total simulation time, the simulation time and last flow will be appended to the hydrograph table.

Unit Sanitary Load	Lets you select the type of the load (for example Apartment or Airport). Unit sanitary loads are selected, edited, and created in the Unit Sanitary (Dry Weather) Load dialog box, which you access by clicking the ellipsis (…) button next to this field.
Unit Sanitary Loading Unit	Lets you enter the local count of loading units for the selected unit sanitary load.
Unit Sanitary Base Flow	Lets you enter the average inflow over the duration of the simulation.
Pattern Load	Lets you select the pattern for the selected pattern load. Pattern loads are selected, edited, and created in the Pattern Manager, which you access by clicking the ellipsis (...) button next to this field.

Related Topics

- "Composite Hydrographs" on page 7-376
- "Working with Patterns" on page 7-361
- "Adding Unit Sanitary (Dry Weather) Loads" on page 7-370

LoadBuilder

LoadBuilder is a tool used to assign flows to elements in Bentley SewerGEMS V8*i*. If you already knows what flows to assign to an element, then you should use the other methods such as inflow, sanitary loads, or stormwater loading. The power of LoadBuilder is that it can take loading information from a variety of GIS based sources such as customer meter data, system flow meter or polygons with known population or land use and assign those flows to elements. LoadBuilder is oriented to the types of data available to describe dry weather flows and other methods in Bentley SewerGEMS V8*i* are more amenable to wet weather flows.

For more information about using LoadBuilder, see "Using LoadBuilder to Assign Loading Data" on page 11-679.

Rainfall Derived Infiltration and Inflow (RDII)

Rainfall derived infiltration and inflow (RDII) is the flow contribution to sanitary sewers from wet weather events. In Bentley SewerGEMS V8*i*, the most appropriate method for loading that flow is to derive unit hydrographs based on flow monitoring and enter them using the procedure described in "Adding Generic Unit Hydrographs" on page 7-411. Other methods may be added in the future.

Stormwater Flow

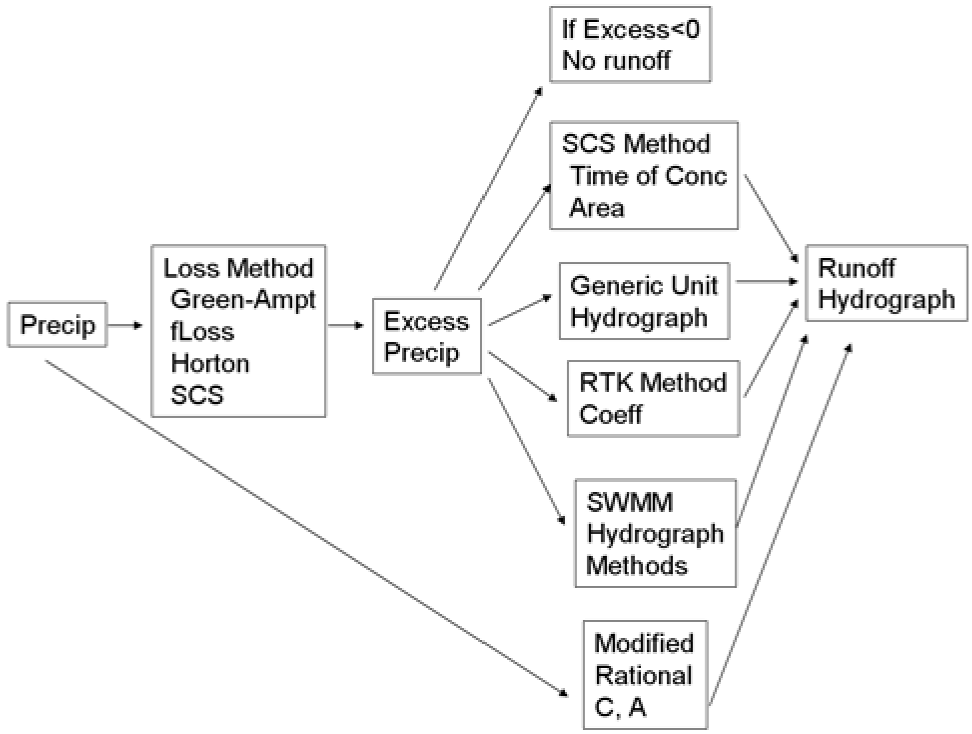
While it is possible to directly specify an inflow hydrograph at virtually any node element, users may wish to load models during wet weather with flow that are derived from precipitation. For this approach to be workable, you must specify:

- Storm event
- Catchment characteristics
  - Catchment size
  - Loss method
  - Hydrograph method

Flow calculated from stormwater runoff can only be placed on catchment elements. The methods described in this section are primarily intended for stormwater runoff or the wet weather contribution to combined sewer systems. For sanitary systems, "Rainfall Derived Infiltration and Inflow (RDII)" on page 7-394.

Snowmelt must be converted into equivalent precipitation to use the methods in this section.

The steps in using loss methods and hydrograph methods to generate a hydrograph from precipitation data are summarized in the following diagram.



Adding Storm Events

A storm event is a single rainfall curve that represents one rainfall event for a given recurrence interval. The rainfall curve can be represented in one of three ways.

- Intensity
- Incremental
- Cumulative

Once the storm event is created it can either be used locally at a catchment, or it can be used globally by selecting Analysis > Global Storm Events. This will apply the storm event to the current scenario. This storm event will then be applied to all catchments during analysis that do not have localized rainfall. Because Bentley SewerGEMS V8*i* dynamically routes flow through the system, it requires a complete storm hyetograph (i.e. a table with rainfall vs. time). There are many methods including actual historical storms and synthetic storms. These can be entered as cumulative precipitation or incremental precipitation.

For background on rainfall data, see the Modeling Rainfall chapter in *Stormwater Conveyance Modeling and Design* or the Wet weather flow chapter in *Wastewater Collection System Modeling and Design*. Both books are published by and available from Bentley Institute Press.

A storm event can be created one of two ways, both from within the Storm Events dialog box:

- You can manually create a storm event by selecting Cumulative, Incremental, Intensity, or IDF Storm Event when selecting **New** to create a new curve in the Storm Event dialog.
- You can construct a storm event from a dimensionless rainfall curve stored in the associated Dimensionless Rainfall Curve engineering library. To do so select **From Dimensionless Curve** when selecting **New** to create a new storm event in the Storm Event dialog box.

To add a storm event:

1. Select **Analysis > Storm Events** or click the Storm Events button on the Analysis toolbar.
   – You can define a local storm event for a specific catchment in the Rainfall section of the catchment's Property Editor. Set **Use Local Rainfall?** to True, then select the ellipses (<...>) from the Local Storm Event submenu.
2. In the Storm Events dialog box, click the **New** button, then select the type of storm event you want to create from the submenu (Cumulative, Incremental, Intensity, IDF Storm Event, or From Dimensionless Curve).
3. Define Cumulative, Incremental, Intensity, and IDF storm events on the Data tab by performing the following steps:
   a. For Cumulative, Incremental, and Intensity storm events, type the Start Time, Increment, and End Time in the Time Settings dialog box, then click **OK**. The Time Settings dialog box appears as soon as you select one of these storm events from the New submenu.
   b. For Cumulative, Incremental, Intensity, and IDF storm events, type the number of years in the Return Event field.
   c. For Cumulative, Incremental, and Intensity storm events, enter Depth or Intensity data values in the table. The Time values are automatically filled in based on the time settings you defined for the storm event. To edit the time settings, click the **Edit** button on the Data tab.
   d. For an IDF Storm Event, enter Time and Intensity data values into the table. Pressing **Tab** or **Enter** after typing a value creates a new row in the table.

4. To define a From Dimensionless Curve storm event, perform these steps:
    a. Select From Dimensionless Curve from the New submenu. The Engineering Libraries dialog box appears, showing the Dimensionless Rainfall Curve Library.
    b. Click the plus sign next to the library to expand it, select the dimensionless rainfall curve you want to use, then click **Select**.
    c. The Rainfall Curve Import Settings dialog box appears. Select the Storm Event Data Type (Intensity or Depth), the Storm Event Depth Type (Incremental or Cumulative), then type values for Depth and Duration. Click **OK** to close the Rainfall Curve Import Settings dialog box.
    d. The storm event appears in the Storm Events dialog box. Double-click the event to edit the import settings.

Note: In a Cumulative Dimensionless Depth rainfall curve, time is in hours and depth is dimensionless. In a Cumulative Dimensionless rainfall curve, both time and depth are dimensionless.

5. You can save your new storm events in Bentley SewerGEMS V8*i*' Engineering Libraries for future use. To do this, perform these steps:
    a. Select the storm event you want to save.
    b. Click the **Synchronization Options** button, then select **Export to Library**. The Engineering Libraries dialog box appears.
    c. Use the plus and minus signs to expand and collapse the list of available libraries, then select the library into which you want to export your new storm event.
    d. Click **Close**.

6. Perform the following optional steps:
    – To delete an event, select the event label then click **Delete**.
    – To rename an event, select the event label you want to rename, click **Rename**, then type the new name for the event.
    – To view a report on an event, select the event label for which you want a report then click **Report**.
    – To graph the rainfall for an event, select the event label for which you want a graph then click **Graph**.

7. Click **Close** to close the Storm Events dialog box.

To add a storm event in the Engineering Libraries:

1. Select **Tools** > **Engineering Libraries** to display the Engineering Libraries dialog box.
2. Click the plus sign next to the Storm Event Library to expand the list of items (categories and folders) included in that library.

   Note: You can add new items to a category or a folder, add new folders to categories, and add new categories to libraries. For more information, see “Engineering Libraries” on page 6-288**.**

3. Right-click a category or folder in the Storm Event Library and select **New Item**. The new item appears at the bottom of the list of items in the selected category or folder.
4. Define the new storm event in the Editor pane on the right as described in the following steps:
   a. Select the type of storm event from the Storm Event Type submenu (Time vs. Depth, Time vs. Intensity, or Rational Method IDF Curve).
   b. For Time vs. Depth or Time vs. Intensity storm events, type the start time, increment, and end time in the appropriate fields.
   c. For a Time vs. Depth storm event, select the depth type from the Depth Type submenu (Incremental or Cumulative).
   d. For a Time vs. Depth storm event, click the **Ellipses (...)** button next to the Depths field, then enter Time vs. Depth data in the Storm Event dialog box. Click **OK** to close the Storm Event dialog box when you are done.
   e. For a Time vs. Intensity storm event, click the **Ellipses (...)** button next to the Intensities field, then enter Time vs. Intensity data in the Storm Event dialog box. Click **OK** to close the Storm Event dialog box when you are done.
   f. For a Rational Method IDF Curve storm event, click the **Ellipses (...)** button next to the IDF Curve field, then enter data values in the Rational Method IDF Curve dialog box. Click **OK** to close the Ration Method IDF Curve dialog box when you are done.

5. To create a Dimensionless Rainfall Curve in the Engineering Libraries, perform these steps:

   a. Right-click the category or folder in the Dimensionless Rainfall Curves library in which you want to store your new dimensionless rainfall curve, then select **New Item**. The new item appears at the bottom of the list of items contained in the category or folder.
   b. Select the new item, then define the curve in the Editor pane. Select the Dimensionless Time Type (Time or Dimensionless Time), then type values for the Start Time, Increment, and End Time.

c. Click the **Ellipses (...)** button next to the Dimensionless Depth field. The Rainfall Curve Dictionary dialog box appears.

d. Time or Synthetic Time values are automatically inserted into the data table based on the start time, increment, and end times you entered. Type values for Depth, then click **OK**.

e. Click **Close** to close the Engineering Libraries.

Note: In a Cumulative Dimensionless Depth rainfall curve, time is in hours and depth is dimensionless. In a Cumulative Dimensionless rainfall curve, both time and depth are dimensionless.

6. Click **Close**. Your storm event is now part of the Engineering Libraries and can be re-used any time.

Storm Events Dialog Box

You create, edit, and delete storm events in the Storm Events dialog box.

The dialog box contains a list pane on the left and tabbed area on the right and includes the following controls:

New

Creates a new storm event that uses an automatically created label.

- **Cumulative**—Adds a new storm event to the list pane of the type Cumulative.
- **Incremental**—Adds a new storm event to the list pane of the type Incremental.
- **Intensity**—Adds a new storm event to the list pane of the type Intensity.
- **IDF Storm Event**—Adds a new IDF (intensity-duration-frequency) storm event to the list pane.
- **From Dimensionless Curve**—Opens the Engineering Library, where you can select an existing dimensionless rainfall curve to add as a new storm event.

Delete

Deletes the currently highlighted storm event.

	**Rename**	Lets you rename the currently highlighted storm event.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted storm event.
	**Graph**	Graphs a plot of the current storm event.
	**Synchronization Options**	Clicking this button opens a submenu containing the following commands: • **Browse Engineering Library**—Opens the Engineering Library manager dialog, allowing you to browse the Storm Event and Dimensionless Rainfall Curves Libraries. • **Synchronize From Library**—Lets you update a set of storm events previously imported from a Storm Event or Dimensionless Rainfall Curve Library. The updates reflect changes that have been made to the library since it was imported. • **Synchronize To Library**—Lets you update an existing Storm Event or Dimensionless Rainfall Curve Library Library using current storm events that were initially imported but have since been modified. • **Import From Library**—Lets you import a storm event from an existing Storm Event or Dimensionless Rainfall Curve Library. • **Export To Library**—Lets you export the current storm event to an existing Storm Event or Dimensionless Rainfall Curve Library.

The following fields and controls appear in the tabbed area:

Data Tab

Edit Button 

Displays the Time Settings dialog box, which lets you define the temporal attributes of the storm event.

Return Event

Lets you enter the return event value for the storm event.

Storm Event Data Table

Lets you define the points that make up the currently highlighted storm event (time vs. intensity or time vs. depth).

Notes Tab

This tab contains a text field that allows you to enter descriptive notes that will be associated with the currently highlighted storm event.

Library Tab

This tab displays information about the storm event that is currently highlighted in the list pane. If the storm event is derived from an engineering library, the synchronization details can be found here. If the storm event was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the storm event was not derived from a library entry.

Time Settings Dialog Box

You define the temporal attributes of a storm event in the Time Settings dialog box.

This dialog automatically appears when you select Cumulative, Incremental, or Intensity as a new storm event in the Storm Events dialog box. After you create one of these storm events, you can access the Time Setting dialog box by clicking the Edit button that appears above the table in the Storm Events dialog box.

This dialog box contains the following controls:

Start Time

Lets you define the time for the first ordinate in a storm event.

Increment

Lets you define the duration of the time step between ordinates in a storm event.

End Time	Lets you define the time for the last ordinate in a storm event.

Storm Event Dialog Box

This dialog box lets you define Time vs. Depth and Time vs. Intensity storm events from within Bentley SewerGEMS V8*i*' Engineering Libraries.

Access the Storm Event dialog box by clicking the Ellipsis (...) button next to the Depths or Intensities field when for a Storm Event entry in the Engineering Library explorer pane.

The dialog box includes the following controls:

	**Edit**	Opens the Time Settings dialog (see "Time Settings Dialog Box" on page 7-402 for more information), allowing you to define the start time, stop time, and increment for the current storm event.
	**Graph**	Graphs a plot of the current storm event.

The Storm Event dialog box also includes the following controls:

Return Event	Lets you enter the return event, a value that reflects the average time between similar storm events.
Time vs. Depth or Intensity table	Allows you to define time vs. depth or time vs. intensity points that describe the current storm event. Which table is displayed in the dialog depends on the Storm Event Type specified in the library entry.

Rainfall Curve Import Settings Dialog Box

This dialog box appears when you create a new From Dimensionless Curve storm event and after you select a dimensionless rainfall curve in the Engineering Libraries to import into the Storm Events dialog box. The Rainfall Curve Import Settings dialog box lets you define the data type, depth type, depth, and duration for a storm event created from a dimensionless curve.

The dialog box contains the following controls:

Storm Event Data Type	Lets you specify the data type as Depth or Intensity. If you select Intensity, the Storm Event Type field becomes inactive.
Storm Event Depth Type	Lets you specify the depth type as Cumulative or Incremental. This field is unavailable if the Storm Event Data Type is Intensity.
Depth	Lets you specify the depth of the storm event.
Start Time	Lets you specify the time at which the translated dimensionless curve begins. The default and suggested value is 0.00 hours.
Duration	Lets you specify the duration of the storm event.

Rainfall Curve Dictionary Dialog Box

This dialog lets you define dimensionless rainfall curves. These curves are stored in the Dimensionless Rainfall Curves Engineering Library, and can be used to define Global and local Storm Events (see “Adding Global Storm Events” on page 7-405 and “Adding Storm Events” on page 7-395 for more information).

The Rainfall Curve Dictionary dialog box is accessible by clicking the Ellipsis button of the Dimensionless Depths field when a Dimensionless Rainfall Curve entry is highlighted in the Engineering Library explorer pane.

The dialog contains the following controls:

	**Edit**	Opens the Time Settings dialog box (see “Time Settings Dialog Box” on page 7-402 for more information), allowing you to define the start time, stop time, and increment for the current rainfall curve.
	**Graph**	Graphs a plot of the current rainfall curve.

The Rainfall Curve Dictionary dialog box also contains the following table:

Time vs. Depth table	lets you define time vs. depth points that describe the current rainfall curve. The time column will be either Synthetic Time or Time, depending on whether the Dimensionless Time Type specified in the library entry is Dimensionless Time or Time, respectively.

Rational Method IDF Curve Dialog Box

This dialog allows you to define intensity-duration-frequency curves. These curves are stored in the Storm Events Engineering Library, and can be used to define Global and local Storm Events (see “Adding Global Storm Events” on page 7-405 and “Adding Storm Events” on page 7-395 for more information).

The Rational Method IDF Curve dialog box is accessible by clicking the Ellipsis button of the Depths field when a Rational Method IDF Curve entry is highlighted in the Engineering Library explorer pane.

The dialog contains the following controls:

Return Event	Lets you enter the return event, a value that reflects the average time between similar storm events.
Time vs. Intensity table	Allows you to define time vs. depth or time vs. intensity points that describe the current storm event. Which table is displayed in the dialog depends on the Storm Event Type specified in the library entry.

Adding Global Storm Events

Individual catchments in your model can have local storm events assigned to them. These are the storm events you create in the Storm Events dialog box or in the Storm Events Engineering Library. For catchments that do not use local rainfall (Use Local Rainfall setting in the Property Editor is set to False), project-wide global storm events apply. Project-wide global storm events are associated with Hydrology alternatives.

You define project-wide global storm events in the Global Storm Events dialog box.

To add a global storm event:

1. Select **Analysis > Global Storm Events** or click the **Global Storm Events** button on the Analysis toolbar.
2. In the Global Storm Events dialog box, each row in the table represents a Hydrology alternative. In the Global Storm Event column, select the storm event from the submenu or click the **Ellipses (...)** button to display the Storm Events dialog box, where you can create a new storm event.
3. The rest of the data is automatically added to the table based on the settings of the selected storm event. Click **Close**.

Global Storm Event Settings Dialog Box

This dialog box lets you define project-wide global storm event data. You access the Global Storm Events Settings dialog box by selecting **Analysis > Global Storm Events** or clicking the Global Storm Event button on the Analysis toolbar.

The dialog box a table with the following columns:

Alternative	Displays the name of the Hydrology alternative that is used by the current scenario.
Global Storm Event	Lists all of the rainfall curves that have been defined for the current project in the Storm Events dialog box, which is accessible by clicking the ellipsis button.
Source	Displays the location of the library file for storm events that are derived from an engineering library entry.
Return Event	Displays the return event that is associated with the storm event. The return event is a value that reflects the average time between similar storm events.
Depth	Displays the rainfall depth of the storm as defined by the currently selected storm event.
Duration	Displays the duration of the storm as defined by the currently selected storm event.

Catchment Characteristics

The precipitation that falls on a catchment can only be converted into a flow to a sewer if the user specifies the following:

- Area to determine magnitude of available flow
  (For more information, see "Entering Area" on page 7-407.)
- Runoff method which consists of:
  - Loss method to determine the amount of available flow that actually runs off
  - Hydrograph method to determine the shape of the runoff hydrograph

The modified rational method does not explicitly have a loss method associated with it.

Entering Area

There are two ways to specify area in a catchment.

The first involves directly entering the area in the Property Editor or FlexTable for the catchment.

The second is based on the scaled area from the drawing. This can only be used if the drawing is scaled and the units have been correctly identified. Bentley SewerGEMS V8*i* displays the scaled area in the Property Editor for the catchment and you must copy that value in the Area attribute. For more information, see "Catchment—Geometry" on page 15-890.

It is possible to have an area that is not uniform in terms of runoff. For more information, see "Defining CN Area Collections for Catchments" on page 7-387.

For more information, see "Catchment—Catchment" on page 15-890.

Defining CN Area Collections for Catchments

SewerGEMS V8*i* lets you define infiltration for a catchment based on CN (SCS Curve Number) and Area data that you specify. You define and save this data in a CN Area collection.

In SewerGEMS V8*i*, the sub-basin runoff is defined solely by the CNinput for each watershed. The CN Area Collection dialog box automatically computes weighted CN values as a function of soil hydrologic class and cover characteristics, based on the CN Engineering Library entries that you select when you create a CN Area collection.

Note: The USDA has classified its soil types into four hydrologic soil groups. For a complete description of the CN values for various land uses and cover characteristics for each soil classification, see “The Runoff Curve Number” on page 14-798**.**

To define a CN Area collection for a catchment:

1. Click a catchment in your model to display the Property Editor, or right-click a catchment and select **Properties** from the shortcut menu.
2. In the Runoff section of the Property Editor, select **Unit Hydrograph** as the Runoff Method. The Loss Method field becomes available.
3. Select **SCS CN** as the Loss Method. the SCS CN field becomes available.
4. Click the **Ellipses (...)** button next to the SCS CN field. The CN Area Collection dialog box appears.
5. Type a description for the collection in the Description field, or click the **Ellipses (...)** button to display the CN Libraries in the Engineering Libraries.
6. Click the plus signs to expand the list of items in the CN Libraries until you find the CN Value for the soil hydrologic class and cover characteristics that you want to use.
7. Click **Select** to close the Engineering Libraries dialog box and add the CN Value to the table in the CN Area Collection dialog box.
8. SewerGEMS V8*i* automatically fills in the values for SCS CN and Area. Type values for Percent Connected Impervious Area and Percent Unconnected Impervious Area.

   Note: You can change the SCS CN value by clicking the Ellipses (...) button next to the SCS CN field, then selecting a different CN Value from the CN Engineering Libraries.

9. Repeat Steps 5 - 8 for each item you want to add to the CN Area collection.
10. Click **OK**.

CN Area Collection Dialog Box

This dialog box lets you define infiltration based on Cn and Area data. The dialog box contains the Cn-Area table and the following buttons:

	New	Creates a new row in the cn-area table.
	Delete	Deletes the currently highlighted row from the cn-area table.
	Report	Opens a print preview window containing a report that details the input data for this dialog box.

The table contains the following columns:

Column	Description
Description	Lets you enter a description for the catchment.
CN	Lets you define the Cn value for the catchment.
Area	Lets you define the area for the catchment.
Percent Connected Impervious Area	This field allows you to enter the percent connected impervious area. This value represents the percentage of the area (for the current line of data) that contains directly connected impervious cover. Precipitation that falls on these types of impervious areas flow to the outfall point without ever flowing over pervious cover areas. An adjustment to the Cn for this line of data is made based on the % connected impervious areas (0% yields no adjustment). An example sequence of directly connected impervious areas are rainfall drops onto a concrete driveway, driveway drains to a paved gutter, gutter drains into a storm sewer, and storm sewer into a detention pond.

Column	Description
Percent Unconnected Impervious Area	Enter the percent unconnected impervious area. This value represents the percentage of the area (for the current line of data) that contains unconnected impervious cover. Precipitation falling on these types of impervious areas eventually flows over pervious areas before reaching the outfall. An adjustment to the Cn for this line of data is made based on the % unconnected impervious areas (0% yields no adjustment). An example sequence of directly unconnected impervious areas are rainfall drops onto a paved tennis court at a grassed park, water sheet flowing across a paved tennis court, water flowing off the tennis court onto a grass field in a park (there are no inlet drains on the court), and some of the water then being absorbed into the soil.

Runoff Method

Bentley SewerGEMS V8*i* supports the following runoff methods which can be selected by the user in the Property Editor or FlexTable under the attribute Runoff Method. They include:

- None - no runoff
- Modified rational method
- Unit hydrograph
- EPA SWMM
- User defined hydrograph

The data required varies for each method. The data requirements are summarized in “Catchment—Runoff” on page 15-891. The data needs for each method are described in their individual sections.

The input parameters for each method are listed below. Each of these methods is described in standard hydrology and stormwater references such as *Stormwater Modeling Conveyance and Design*, published Haestad Press.

Input Parameters for Unit Hydrograph Runoff Methods

- Loss method

- Green Ampt
  - Capillary suction
  - Ks
  - Moisture deficit
- SCS Curve Number
- Horton
  - fc
  - fo
  - Initial abstraction
  - K
  - Recovery constant
  - Maximum volume
- fLoss

- Time of Concentration
  (For more information, see "Specifying a Time of Concentration (Tc) Method for a Catchment" on page 6-239.)
- Unit hydrograph method
  (For more information, see "Unit Hydrograph Methodology" on page 14-807.)
  - Generic unit hydrograph
    (For more information, see "Generic Unit Hydrographs" on page 14-808.)
  - RTK Method
    (For more information, see "Adding Hydrographs Based On the RTK Method" on page 7-415.)
  - SCS
    (For more information, see "Using the SCS Unit Hydrograph Runoff Method" on page 7-421.)

For more information on unit hydrograph runoff methods, see "Unit Hydrograph Runoff Methods" on page 14-811.

Adding Generic Unit Hydrographs

You can directly associate a user-defined unit hydrograph to a subarea (catchment) for runoff calculations. In dynamic flow routing, the hydraulics of the system are modeled over time.

The coefficients for the generic unit hydrographs are usually developed based on a linear correlation analysis between precipitation data and flow measurement. Usually flow monitoring data are only available at a handful of locations in a collection system. Therefore, when using generic unit hydrograph, you only need to load a model at points corresponding to flow monitoring locations. This lumping of wet weather loads will model flows downstream of the monitoring point accurately but will underestimate flow immediately upstream of the monitoring location.

In this method, the rainfall is converted into catchment outflow in discrete time intervals, called convolution time steps. Usually, it is best to set the convolution time step to the same value as the increment in the unit hydrograph that you enter. Smaller intervals won't produce greater accuracy while larger intervals will lose resolution.

For more information on unit hydrographs, see "Generic Unit Hydrographs" on page 14-808 and "Soil Conservation Service (SCS)" on page 14-809.

Note: Implementation of generic unit hydrographs in PondPack is not the same as generic unit hydrographs in CivilStorm and SewerGEMS.

To add a unit hydrograph to a catchment:

1. Click a catchment in your model to display the Property Editor, or right-click a catchment and select **Properties** from the shortcut menu.
2. In the Runoff section of the Property Editor, select **Unit Hydrograph** as the Runoff Method.
3. Select **Generic Unit Hydrograph** in the Unit Hydrograph Method drop-down.
4. Click the **Ellipses (...)** button next to the Runoff Hydrograph field. The Unit Hydrograph Data dialog box appears.
5. Click the **New** button to add a row to the Time vs. Flow table.
6. Enter Time vs. Flow data into the table. Press **Enter** after typing a value to add a new row to the table (or click the **New** button to add a new row).
7. Click the **Graph** button to view a plot of the Time vs. Flow data.
8. Click **OK** to close the dialog box and add the hydrograph to the Property Editor for the catchment.

Unit Hydrograph Data Dialog Box

This dialog box allows you to define Time vs. Flow unit hydrographs. The dialog box contains the time-vs.-flow table and the following buttons:

	**New**	Creates a new row in the hydrograph table.
	**Delete**	Deletes the currently highlighted row from the hydrograph table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the unit hydrograph defined by the points in the table.

The table contains the following columns:

Column	Description
Time	Lets you define the hour of the hydrograph point.
Flow	Lets you define the flow for the hydrograph point.

EPA SWMM

The input parameters for each method are listed below.

- Loss method
  - Green Ampt
    - Capillary suction
    - Ks
    - Moisture deficit
  - SCS Curve Number
  - Horton
    - fc
    - fo
    - Initial abstraction
    - K
    - Recovery constant
    - Maximum volume
  - fLoss
- Storage (Impervious Depression)
- Storage (Pervious Depression)
- Impervious Manning's n
- Pervious Manning's n
- Percent impervious
- Slope
- Subarea routing
  - Impervious
  - Pervious

  – Outlet
- Percent routed

If you are using the EPA SWMM runoff method with the SWMM engine, you must:

1. Use the EPA-SWMM runoff for all catchments;

2. Use the same loss method for all catchments;

3. Set the Infiltration Method under Calculation Options to the loss method used by all catchments.

For more information on SWMM input data, see “Using the SWMM Water Quality Solver” on page 6-294.

Adding Hydrographs Based On the RTK Method

The RTK method is used to generate a hydrograph based on precipitation data. It forms the hydrograph by combining triangular hydrographs from three components of flow:

- Rapid inflow
- Moderate infiltration
- Slow infiltration

A typical RTK hydrograph is shown below. Q1, Q2 and Q3 refer to the three components of flow which must be summed to determine the flow.

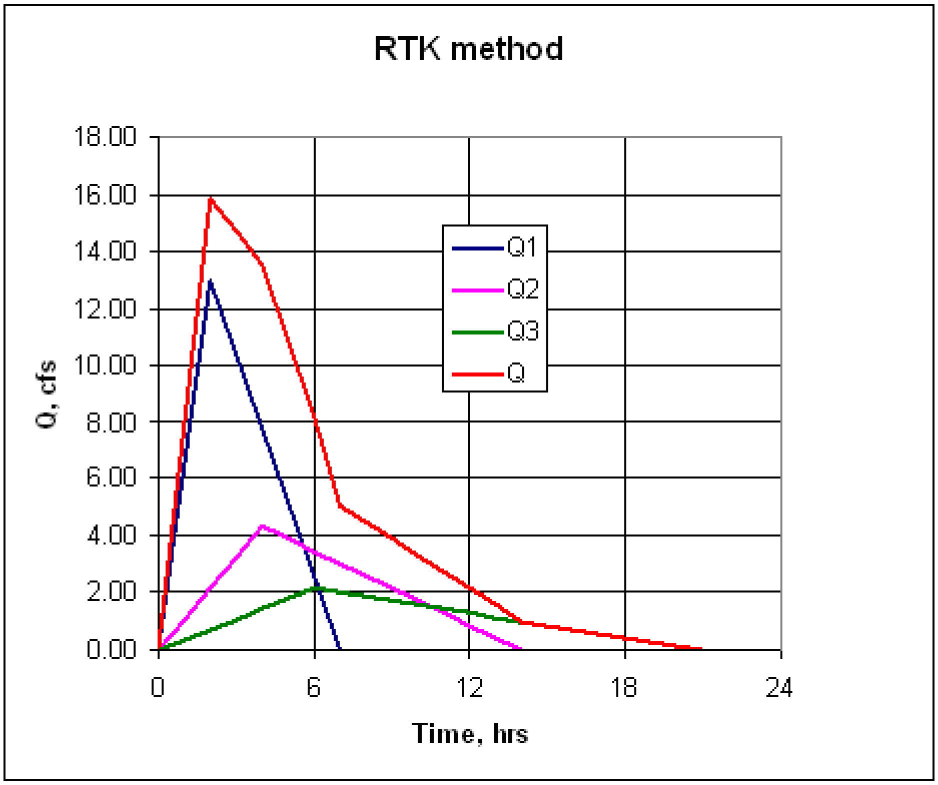


Figure 7-2: A Typical RTK Hydrograph

For information on the theory behind this method, see "RTK Methods" on page 14-815. RTK methods are described further in the wet weather flow chapter of *"Wastewater Collection System Modeling and Design,"* available from Bentley Institute Press.

The RTK method is most appropriate for determining RDII (Rainfall Derived Infiltration and Inflow) to sanitary sewers. It treats the system between the rainfall and flow in the sewer as a black box which can be represented by the three parameters R, T and K. In storm sewer systems, most of the flow moves over the surface and enters the sewer at known locations; a method such as the SCS hydrograph method is more appropriate in these systems.

The RTK method requires that the storm event be specified as a hyetograph, not simply a peak intensity. The resulting hydrograph from the catchment will have time steps equal to the time step size in the hyetograph. For example if precipitation is specified in 0.25 hr increments, flow will be calculated in those increments.

For more information about RTK methods, see:

- "Assembling RTK Parameters" on page 7-417
- "Creating an RTK Table and Assigning it to a Catchment" on page 7-418
- "RTK Tables Dialog Box" on page 7-420
- "RTK Methods" on page 14-815

Assembling RTK Parameters

The RTK parameters are:

- **R** - fraction of precipitation that enters the collection system for that component
- **T** - the time from the precipitation pulse to the peak of that component of the hydrograph
- **K**- the ratio of the time to peak to time to end of hydrograph for that component.

There is no theoretical method to determine R, T and K. They must be determined empirically for each system based on a comparison of a measured rainfall hyetograph with measured wet weather sewer flow.

In general, R will be higher for systems that have significant infiltration and inflow problems than tight systems. T will be larger for larger catchments and will be larger for slow infiltration than rapid inflow. K is usually on the order of 1.5 to 2.5. The sum of the R values for all components should be positive but less than 1. The default units for T is hours while R and K are dimensionless.

Determining R, T and K for a particular catchment (or group of catchments) and event involve constructing a model, then trying different values of R, T and K that, when combined with sanitary dry weather flow, match the measured system hydrograph.

In some cases, the RTK parameters may have been determined for a large area, say several square kilometers (square miles) but the model is being loaded based on catchments on the order of a few hectares (acres). In this case, the R and K values are likely to be valid but the T values may need to be decreased to reflect the shorter time of concentration of these smaller catchments.

It is best to calibrate the RTK method using several storm events.

Note: The word "infiltration" is used in two different ways in wet weather flow monitoring:

1. It is the precipitation that seeps into the ground.

2. It is the precipitation that seeps through the groundwater into collection systems.

In Bentley SewerGEMS V8*i*, generally the first definition is used. In the RTK method, the second definition is typically used.

Creating an RTK Table and Assigning it to a Catchment

The RTK parameters are a property of each catchment. However, it is not uncommon for many catchments with similar characteristics to share the same RTK parameters. Therefore, the RTK parameters are entered in a named RTK table and that table can be shared among many catchments.

The RTK table is a set of RTK parameters that is available to be assigned to catchments. The following table shows some example ranges of RTK values.

Table 7-2: Example RTK Values

Component	R	T	K
Rapid Inflow	0.08	2	1.5
Moderate Infiltration	0.04	3.5	1.75
Slow Infiltration	0.06	7	1.67

In most cases, the RTK hydrograph implicitly accounts for infiltration so when using RTK, you should set this type of infiltration to zero. This can be done by setting the infiltration rate to zero in the fLoss method or setting fc and fo to zero in the Horton (generic) method.

To create a RTK table:

1. Open your project in Bentley SewerGEMS V8*i*, then do one of the following:
   - Select **Analysis > RTK Tables**.
   - Double-click a catchment in your model, then in the Property Editor for the catchment, select **Unit Hydrograph** as the Runoff Method, **RTK Unit Hydrograph** as the Unit Hydrograph Method, then **New** as the RTK Table.
   - Open a catchment FlexTable, then click the **Ellipse (...)** button in the RTK Table cell.

2. Click the **New** button to create a new RTK table, then type a new name for the RTK table.
3. Enter RTK values for Rapid Inflow, Moderate Infiltration, and Slow Infiltration. You can use the Tab key to quickly move from one field to the next.

   Note: If you do not use one of the flow components, set the R value of that component to zero.

   Bentley SewerGEMS V8*i* validates your RTK data as you enter it and displays errors and warnings in the status bar at the bottom of the RTK Tables dialog box. Be sure to check this status bar for any errors or warnings as you enter data.

4. Click **Close** to close the dialog box and save the RTK table.
5. In the Property Editor for the catchment, select the new RTK table from the drop-down menu in the RTK Table field.

To assign an RTK table to a catchment:

The advantage of using FlexTables to assign RTK tables to catchments is that the hydrograph types can be assigned globally.

1. To assign an RTK table to a catchment using the Property Editor, perform these steps:
   a. Double-click a catchment in your model.
   b. In the Property Editor, select **Unit Hydrograph** as the Runoff Method.
   c. Select **RTK Unit Hydrograph** as the Unit Hydrograph Method.
   d. In the RTK Table field, select the desired RTK table from the drop-down menu, or select **New** to create a new RTK table.

2. To assign an RTK table to a catchment using a FlexTable, perform these steps:
   a. Open or create a catchment FlexTable.

      For information on creating FlexTables, see "Creating a New FlexTable" on page 10-563. For information on adding columns to an existing FlexTable, see "Editing FlexTables" on page 10-564.
   b. Add the following columns to the FlexTable:
      - Runoff Method
      - Unit Hydrograph Method
      - RTK Table
   c. In the Runoff Method cell in the FlexTable, select **Unit Hydrograph**.

d. In the Unit Hydrograph Method cell, select **RTK Unit Hydrograph**.

e. In the RTK Table cell for the catchment, select the appropriate RTK table or click the **Ellipses (...)** button to create a new table.

RTK Tables Dialog Box

You create RTK tables in the RTK Tables dialog box. The dialog box contains a list pane with the following controls:

	**New**	Creates a new RTK table that uses an automatically created label.
	**Delete**	Deletes the currently highlighted RTK table.
	**Rename**	Lets you rename the currently highlighted RTK table.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted RTK table.
	**Duplicate**	Lets you create an exact copy of the selected RTK table with a new name.

The RTK Tables dialog box also contains the following controls:

Rapid Inflow

Lets you enter RTK values for the rapid inflow component of flow:

- **R—**Fraction of precipitation that enters the collection system for rapid inflow.
- **T—**The time from the precipitation pulse to the peak of rapid inflow of the hydrograph.
- **K—**The ratio of the time to peak to time to end of hydrograph for rapid inflow.

Moderate Infiltration	Lets you enter RTK values for the moderate infiltration component of flow: • **R—**Fraction of precipitation that enters the collection system for moderate infiltration. • **T—**The time from the precipitation pulse to the peak of moderate infiltration of the hydrograph. • **K—**The ratio of the time to peak to time to end of hydrograph for moderate infiltration.
Slow Infiltration	Lets you enter RTK values for the slow infiltration component of flow: • **R—**Fraction of precipitation that enters the collection system for slow infiltration. • **T—**The time from the precipitation pulse to the peak of slow infiltration of the hydrograph. • **K—**The ratio of the time to peak to time to end of hydrograph for slow infiltration.

There is also a status bar located at the bottom of the dialog box that displays any errors and warnings that may occur when you enter data.

Using the SCS Unit Hydrograph Runoff Method

When the SCS Unit Hydrograph Runoff Method is assigned to a catchment, Bentley SewerGEMS V8*i* allows you to choose between the following three dimensionless methods to further customize the hydrograph:

- **Default Curvilinear—**This method uses the standard dimensionless curvilinear unit hydrograph ordinates, as defined in “Chapter 4” of the National Engineering Handbook, Section 4.
- **Triangular Unit Hydrograph—**This method uses a triangular unit hydrograph shape to define the coordinates for the unit hydrograph. The default shape factor is 484 (37.5% of the volume on the rising limb of the unit hydrograph).
- **Dimensionless Unit Hydrograph—**This method allows you to customize the Q/Q_p-T/T_p unit hydrograph that is used with the SCS Unit Hydrograph method. See ““ Soil Conservation Service (SCS)”on page 14-811Adjusting the Q/Qp-T/Tp Unit Hydrograph” on page 7-422 for more information.

“ Soil Conservation Service (SCS)”on page 14-809

Adjusting the Q/Q_p-T/T_p Unit Hydrograph

Q/Q_p-T/T_p is a dimensionless unit hydrograph that is used with the SCS Unit Hydrograph method. The Q_p and T_p are determined based on subarea characteristics. Typically, the SCS method uses a default curvilinear unit hydrograph. SewerGEMS V8*i* allows you to assign your own Q/Q_p-T/T_p hydrograph to better suit your local conditions.

To create a new Q/Qp-T/Tp Unit Hydrograph:

1. Highlight a catchment in the model.
2. In the Property Editor, change the value in the Runoff Method field to Unit Hydrograph.
3. Change the Unit Hydrograph Method to SCS Unit Hydrograph.
4. Change the Unit Hydrograph Method Type to Dimensionless Unit Hydrograph.
5. Click the Dimensionless Unit Hydrograph field and select the Edit Dimensionless Unit Hydrograph command.
6. In the Dimensionless Unit Hydrographs dialog that appears, click the New button.

To import a Q/Qp-T/Tp Unit Hydrograph from an Engineering Library:

1. Highlight a catchment in the model.
2. In the Property Editor, change the value in the Runoff Method field to Unit Hydrograph.
3. Change the Unit Hydrograph Method to SCS Unit Hydrograph.
4. Change the Unit Hydrograph Method Type to Dimensionless Unit Hydrograph.
5. Click the Dimensionless Unit Hydrograph field and select the Edit Dimensionless Unit Hydrograph command.
6. In the Dimensionless Unit Hydrographs dialog that appears, click the Synchronization Options button and select the Import From Library command from the menu that appears.
7. In the Engineering Libraries dialog that appears, find the desired library, highlight it, and click the Select button to import it.

Dimensionless Unit Hydrograph Dialog

This dialog allows you to create the Q/Q_p-T/T_p unit hydrographs that are used with the SCS Unit Hydrograph method.

The following controls are available in the Dimensionless Unit Hydrograph dialog box:

	**New**	Creates a new unit hydrograph that uses an automatically created label.
	**Delete**	Deletes the currently highlighted unit hydrograph.

Rename

Lets you rename the currently highlighted unit hydrograph.

Report

Lets you generate a preformatted report that contains the input data associated with the currently highlighted unit hydrograph.

Graph

Lets you plot a graph based on the input data associated with the currently highlighted unit hydrograph.

Synchronization Options

Clicking this button opens a submenu containing the following commands:

- **Browse Engineering Library**—Opens the Engineering Library manager dialog, allowing you to browse the Dimensionless Unit Hydrograph Engineering Libraries.
- **Synchronize From Library**—Lets you update a set of unit sanitary loads previously imported from a Dimensionless Unit Hydrograph Engineering Library. The updates reflect changes that have been made to the library since it was imported.
- **Synchronize To Library**—Lets you update an existing Dimensionless Unit Hydrograph Engineering Library using current unit hydrograph ordinates that were initially imported but have since been modified.
- **Import From Library**—Lets you import a unit hydrograph from an existing Dimensionless Unit Hydrograph Engineering Library.
- **Export To Library**—Lets you export the current unit hydrograph to an existing Dimensionless Unit Hydrograph Engineering Library.

The tab section includes the following controls:

	**New**	This button creates a new row in the Q/Q_p-T/T_p table.
	**Delete**	This button deletes the currently highlighted row from the Q/Q_p-T/T_p table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the Q/Q_p-T/T_p curve defined by the points in the table.
	**Synchronization Options**	Clicking this button opens a submenu containing the following commands: • **Browse Engineering Library**—Opens the Engineering Library manager dialog, allowing you to browse the Dimensionless Unit Hydrograph Library. • **Synchronize From Library**—Lets you update a set of storm events previously imported from a Dimensionless Unit Hydrograph Library. The updates reflect changes that have been made to the library since it was imported. • **Synchronize To Library**—Lets you update an existing Dimensionless Unit Hydrograph Library using current storm events that were initially imported but have since been modified. • **Import From Library**—Lets you import a storm event from an existing Dimensionless Unit Hydrograph Library. • **Export To Library**—Lets you export the current storm event to an existing Dimensionless Unit Hydrograph Library.

Unit Hydrograph Data Tab

T/T_p Column — Q/Q_p is the ratio of actual flow to the peak flow of the unit hydrograph.

Q/Q_p Column — T/T_p is the ratio of actual time to the time-to-peak.

Notes Tab — This tab provides a text field that lets you enter descriptive notes for the unit hydrograph that is currently highlighted in the list pane.

Library Tab — This tab displays information about the unit hydrograph that is currently highlighted in the list pane. If the hydrograph is derived from an engineering library, the synchronization details can be found here. If the hydrograph was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the hydrograph was not derived from a library entry.

Dimensionless Unit Hydrograph Curves Library Editor

This dialog allows you to create engineering library entries containing the Q/Q_p-T/T_p unit hydrographs that are used with the SCS Unit Hydrograph method. The following controls are available in the Dimensionless Unit Hydrograph Curves dialog box:

	**New**	Creates a new unit hydrograph that uses an automatically created label.
	**Delete**	Deletes the currently highlighted unit hydrograph.
	**Rename**	Lets you rename the currently highlighted unit hydrograph.
	**Report**	Lets you generate a preformatted report that contains the input data associated with the currently highlighted unit hydrograph.
	**Graph**	Lets you plot a graph based on the input data associated with the currently highlighted unit hydrograph.

Modified Rational

The modified rational methods used in Bentley SewerGEMS V8*i* are listed below.

- Rational C
- Time of concentration
(For more information, see “Specifying a Time of Concentration (Tc) Method for a Catchment” on page 6-239.)
- Receding limb multiplier
(For more information, see “Calculation Profile Attributes” on page 8-433.)

Time of Concentration

You can add Time of Concentration (Tc) Methods to a catchment in your model. SewerGEMS V8*i* supports 13 different methods.

For more information, see “Specifying a Time of Concentration (Tc) Method for a Catchment” on page 6-239.

Pipeline Infiltration

To model infiltration along a pipeline, it is possible to specify infiltration as:

- Pipe length
- Pipe area
- Pipe diameter-length
- Count
- Hydrograph
- Pattern load

The first four types of infiltration are constant rates while the last two are time varying inflows. For more information on entering data for each type of infiltration, see “Conduit—Infiltration” on page 15-827.

Note: Infiltration loading will be ignored by the calculation engine if it is assigned to a conduit whose upstream element is a Pond Outlet Structure.

Conduit infiltration flow is added after the upstream section of the conduit by the calculation engine. Therefore, when viewing flow results,you will see the infiltration flow in the middle section and the downstream section of the conduit.

Hydrograph Curve Dialog Box

This dialog box allows you to enter Hydrograph Time vs. Flow data to create Hydrographs for use with the Hydrograph Infiltration Load type.

The dialog box contains the Hydrograph Time vs. Flow table along with the following controls:

	**New**	This button creates a new row in the time-flow table.
	**Delete**	This button deletes the currently highlighted row from the time-flow table.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.
	**Graph**	Opens a graph window plotting the hydrograph curve defined by the points in the table

The table contains the following columns:

Column	Description
Hydrograph Time	This field allows you to define time of the hydrograph curve point.
Flow	This field allows you to define flow at the specified time for the hydrograph curve point.

Pond Infiltration

Ponds can lose water by infiltration into groundwater. Any water lost by infiltration does not show up in downstream links.

The user can specify "None" (default) if pond infiltration is not being considered. The user can specify two alternative ways of entering pond infiltration rates:

1. a Constant Flow rate given in flow units
2. an Average Infiltration rate in depth per unit time which is multipied by the area of the pond surface at that time step to determine the infiltration rate.

In applying either method, the model also has a stability filter when the pond water depth is below 0.5 ft so that the infiltration rate will linearly reduce to zero as the depth decreases to zero.

Chapter

Calculating Your Model 8

Calculation Options Manager

You create calculation profiles in the Calculation Options Manager. The Calculation Options Manager consists of a list pane that displays all of the calculation profiles associated with the current project, and a toolbar that contains some common commands.

To display the Calculation Options Manager, select **View > Calculation Options**.

The Calculation Options manager allows you to create option profiles that contain various calculation settings. The dialog box contains a list pane that displays all of the option profiles currently contained in the project, along with a toolbar.

The toolbar contains the following buttons:

	**New**	Creates a new calculation profile. Define the attributes for the profile in the Property Editor.
	**Delete**	Deletes the currently highlighted option profile.
	**Rename**	Lets you rename the currently highlighted calculation profile.
	**Help**	Displays online help for the Calculation Options manager.

If the Property Editor is open, highlighting a option profile in the list causes the settings that make up the profile appear there. If the Property Editor is not open, you can display the settings that make up the profile by highlighting the desired profile and clicking the Properties button in the Calculation Options Manager.

Creating Calculation Profiles

Calculation profiles contain attributes that define how your model is calculated in Bentley SewerGEMS V8*i*. You create calculation profiles in the Calculation Options Manager. You can create several calculation profiles with different attributes depending on the requirements of your project.

Bentley SewerGEMS V8*i* contains a default calculation profile called “Base Calculation Options.” If you do not create additional calculation profiles, Bentley SewerGEMS V8*i* will use this default profile whenever you calculate your model.

Creating a Calculation Profile

To create a calculation profile:

1. Open the Calculation Options Manager by selecting **View > Calculation Options.**
2. Click the **New** button. A new profile appears in the list with a default name.
3. Type a new name for the profile.
4. Double-click the new profile to display its attributes in the Property Editor. Edit the attributes as required.

Editing a Calculation Profile

You edit the attributes of a calculation profile in the Property Editor.

If you select a calculation profile while the Property Editor is open, the attributes for that profile appear there. If the Property Editor is not open, you can display the attributes of the calculation profile by double-clicking the profile in the Calculation Options Manager.

Deleting a Calculation Profile

To delete a calculation profile:

1. Open the Calculation Options Manager by selecting **View > Calculation Options.**
2. Select the profile you want to delete.
3. Press the Delete key, click the Delete button in the Calculation Options Manager, or right-click the profile and select **Delete** from the shortcut menu.

Renaming a Calculation Profile

To rename a calculation profile:

1. Open the Calculation Options Manager by selecting **View > Calculation Options.**
2. Select the profile you want to rename.
3. Click the Rename button or right-click and select **Rename** from the shortcut menu.
4. Type a new name for the profile, then press **Enter**.

Calculation Profile Attributes

A Calculation Options Profile contains the information described in the following table.

Table 8-1: Calculation Profile Attributes

Attribute	Description
General	
Label	Lets you specify a name for the options profile.
Notes	Lets you enter descriptive text to be associated with the current calculation profile.
Calculation Options	
Engine Type	Lets you choose between the **Implicit** and **Explicit (SWMM 5)** calculation engines. Selecting Explicit (SWMM 5) as the engine type makes several SWMM attribute fields available in the Property Editor for elements. For more information on Property Editor element attributes, see "Editing Attributes in the Property Editor" on page 15-821. **Note: If a catchment is using the EPA SWMM runoff method and not using the default infiltration method specified in the SWMM calc options then neither hydrology or network will calculate.** **If the user is not using the EPA SWMM runoff method, then any combination of other runof methods can be used.**

Table 8-1: Calculation Profile Attributes

Attribute	Description
Base Date	Select the calendar date on which the simulation begins.
Analysis Start Time	Select the clock time at which the simulation begins.
Output Increment	Lets you set the Output Increment you want to use to display the output hydrographs of a network analysis.
Total Simulation Time	Lets you specify the duration of the simulation.
Calculation Time Step	Lets you specify the computational time step in hydrodynamic calculations. See "Troubleshooting DynamicWave Model Calculations" on page 8-448 for more information.
Hydrologic Time Step	Lets you specify the computational time step in hydrology runoff calculations.
Pattern Setup	Pattern setups allow you to match unit sanitary (dry weather) loads with appropriate loading patterns. Specify a default Pattern Setup to associate with each Calculation Option profile. Each scenario can use a different Pattern Setup, thus allowing you to model different loading alternatives for different extended period simulations.
Receding Limb Multiplier	Helps define the shape of the modified rational hydrograph. The receding limb of all modified rational hydrographs are taken to occur over a duration obtained by multiplying this value by the Tc. Refer to local design policy and practices to determine the appropriate value of Tc. The default value is 1.0.
Pressure Friction Method	Lets you select a default friction method for all pressure pipes in your model. Select **Mannings**, **Hazen-Williams**, **Darcy-Weisbach**, or **Kutters**. The Property Editor attributes for pressure pipes are updated with the default friction method selected here.
Advanced Calculation Options	
Y Iteration Tolerance	Implicit numerical scheme Newton iteration converge criteria for depth/elevation. We strongly recommend using the default value.

Table 8-1: Calculation Profile Attributes

Attribute	Description
LPI Coefficient	A coefficient for advanced LPI supercritical-subcritical mix flow computation algorithm. We strongly recommend using the default value.
NR Weighting Coefficient	Implicit numerical scheme Newton iteration weighting factor. A Larger value tends to increase stability, but at the cost of performance. We suggest using the default value or values between 0.6 and 0.85, but values between 0.1 and 1.0 are valid. Use 0.98 - 1.0 in very difficult cases. See "Troubleshooting DynamicWave Model Calculations" on page 8-448 for more information.
NR Iterations	Allowed maximum Newton iterations in one time step during the numerical solution process. This attribute value is used to prevent the numerical solver from unlimited iterations under situations when the solver is unable to reach convergence.
Relaxation Weighting Coefficient	Implicit numerical scheme relaxation factor for network computations. The range of valid values is between 0.5 and 0.9.
Computation Distance	Implicit numerical scheme computational distance interval. The model will automatically insert additional computational sections based on this value, a small value increases stability while large value increases performance. Computational distance is the lesser of • one half the pipe length or • default distance See "Troubleshooting DynamicWave Model Calculations" on page 8-448 for more information.

Table 8-1: Calculation Profile Attributes

Attribute	Description
Start Type	By default the engine assumes a dry condition to start the simulation; i.e., all inflows are zero and all depths are zero in all elements at the beginning. For situations in which you want a non-zero starting flow or the catchments' hydrology calculations to give none-zero flows, the model provides two options to start the simulation: select **Warm Up Time** for a warm-up start or **Transition Start** for a transitional start. In a warm-up start, the model estimates initial depths based on none-zero inflows and uses a user-specified warm-up time (entered in the Warm Up Time field) to approach steady state conditions prior to starting the actual simulation. In a transition start, the model initiates the simulation from zero flows and immediately reaches the actual none-zero flows in the first time step and continued the simulation. See "Troubleshooting DynamicWave Model Calculations" on page 8-448 for more information.
Warm Up Time	Lets you enter a warm-up time for warm-up starts. This field is available only when you select **Warm Up Time** as the Start Type.
SWMM Options	
Inertial Terms	Lets you define 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.
Allow Ponding at Gravity Structures	Lets you select whether or not to allow excess water to collect at gravity structures and be re-introduced into the system as conditions permit. Select **True** to allow ponding at gravity structures in your model. In order for ponding to actually occur at a particular node, you must enter a non-zero value for the node's Ponded Area attribute.

Table 8-1: Calculation Profile Attributes

Attribute	Description
Antecedent Dry Period	Lets you 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 subcatchment surfaces.
Use Variable Time Step?	Lets you select whether or not to use a variable time step. The variable step is computed for each time period to prevent an excessive change in water depth at each node. Select **True** to use a variable time step. a safety factor (between 10 and 200%) that will be applied against the variable time step as automatically derived to preserve the Courant stability criterion.
Time Step Multiplier (%)	Lets you enter a safety factor (between 10 and 200%) to be applied against the variable time step as automatically derived to preserve the Courant stability criterion. This field is available only if you select **True** in the Use Variable Time Step? field.
Default Infiltration Method	Lets you determine how to model infiltration of rainfall into the upper soil zone of subcatchments. Select **Horton**, **Green-Ampt**, or **SCS CN**. If you change this attribute, you will have to re-enter values for the infiltration parameters in each subcatchment.
Routing Method	Lets you determine which method to use to route flows through the conveyance system. Select **Uniform Flow**, **Kinematic Wave**, or **Dynamic Wave**.

Table 8-1: Calculation Profile Attributes

Attribute	Description
Integration Method	Lets you determine which integration method to use: **Modified Euler** or **Picard Iterations** (a method of successive approximations).
Minimum Surface Area (Acres)	Lets you enter the minimum surface area to be used at nodes when computing changes in water depth. If you enter 0, then the default value of 12.566 ft^2 (i.e., the area of a 4-ft diameter manhole) is used.
Time Step For Conduit Lengthening	Lets you enter the 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.
Dry Step (hours)	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 hydrologic time step.

What is the Difference Between the Implicit and SWMM Engines?

The "engine" refers to the type of numerical finite difference solution used to solve the St. Venant equations, which describe unsteady one-dimensional, free surface flow. Bentley SewerGEMS V8*i* contains two types of engines:

- The implicit engine uses a four-point implicit finite difference solver which tends to be more stable than an explicit solver. The implicit engine in Bentley SewerGEMS V8*i* is based on the solver in the National Weather Service FLDWAV model.
- The SWMM engine uses the solver from the EPA Stormwater Management Model version 5. This is an explicit solver which, while more prone to stability problems, exactly matches the results from SWMM 5.

Inflow hydrographs are also handled differently by the two engines. The implicit engine interpolates flows between the final flow in the hydrograph and the end time. The SWMM engine assumes that all flows after the final inflow point are zero.

You select the engine you want to use in the Calculation Options manager.

Note: If a catchment is using the EPA SWMM runoff method and not using the default infiltration method specified in the SWMM calculation options then neither hydrology or network will calculate.

If the user is not using the EPA SWMM runoff method, then any combination of other runof methods can be used.

SWMM Treats Pump and Their Discharge Lines Differently Than the Implicit Engine. How Do I Handle the Differences, Especially If I Want to Use Both Engines?

SWMM can "model" a force main without accounting for the hydraulics of the force main from the pump. You simply specify the discharge node for the force main and the water arrives there. In Bentley SewerGEMS V8*i*, the hydraulics of the discharge conduit are explicitly considered.

If you want to get consistent results for a pump line in SWMM and the Bentley SewerGEMS V8*i* implicit engine, the force main should be represented as a "Virtual conduit" in Bentley SewerGEMS V8*i*. For more information about virtual conduits, see "What Is A Virtual Conduit?" on page 6-259.

Calculation Executive Summary Dialog Box

The Calculation Executive Summary dialog box opens automatically after you compute a model. This dialog box reports a summary of the calculations performed on your model. You can also see this report by clicking **Report > Calculation Executive Summary**.

Click **Detailed Summary** for more information about the calculation (see "Calculation Detailed Summary Dialog Box" on page 8-440) or **Close** to close the dialog box.

The Executive Summary dialog box displays the following information:

Scenario Label	Displays the currently selected scenario.
Run Completed	Displays the date and time at which the run was completed.
Storm Event Label	Displays the currently selected hydrology alternative.
Global Storm Event	Displays the current global storm event.
Return Event	Displays the recurrence period of the global storm event.

Total Inflow Volume	Displays the total volume of flow that enters the model during the course of the analysis.
Total System Overflow Volume	Displays the total volume of flow that exits the system due to overflows. This value does not include volume of flow that exits at outfalls.
Total Gutter Volume Change	Displays the gutter flow volume difference between the initial state and the state at the end of the computation.
Total System Outflow Volume	Displays the total volume of flow that exits the outfalls during the course of the analysis.
Total System Volume Change	Displays the total system flow volume difference between the initial state and the state at the end of the computation.
Continuity Error	Displays the total system computational continuity balance error which accounts for total inflows into the system, outflows, flow losses due to overflows, and volume changes. Generally speaking, a high continuity error (>5%) is an indication that the system is hydraulically very challenging and the hydraulic engine may have difficulty achieving stable results.
Total N-R Iterations	Displays the number of Newton iterations used in the calculation. This attribute value is used to prevent the numerical solver from unlimited iterations under situations when the solver is unable to reach convergence. A more hydraulically-complex model requires a larger number of iterations than a less complex model.

Calculation Detailed Summary Dialog Box

The Calculation Detailed Summary dialog box provides a detailed reports of the calculations performed on your model. You can open this dialog box by calculating your model and clicking Detailed Summary in the Calculation Executive Summary dialog box. You can also see this report by clicking **Report > Calculation Detailed Summary**.

Click the tabs in the Calculation Detailed Summary dialog box to review the details of the report:

- “Calculation Options Tab” on page 8-441
- “Catchment Summary Tab” on page 8-442
- “General Summary Tab” on page 8-443
- “Node Summary Tab” on page 8-444
- “Gutter Summary Tab” on page 8-445

Calculation Options Tab

This tab displays the current settings in the Property Editor for calculation options. To change this, click **View > Calculation Options** and change the settings in the Property Editor.

The Calculation Options tab displays the following information:

Label	Displays the currently selected calculation options.
Total Simulation Time	Displays the duration of the model simulation.
Calculation Time Step	Displays the calculation time step used in calculations of your model. See “Troubleshooting DynamicWave Model Calculations” on page 8-448 for more information.
Output Increment	Displays the output increment used in the calculation of your model.
Hydrologic Time Step	Displays the hydrologic time step used in calculations of your model.
Y Iteration Tolerance	Displays the Y iteration tolerance used in calculations of your model.
LPI Coefficient	Displays the LPI coefficient used in calculations of your model.

NR Weighting Coefficient	Implicit numerical scheme Newton iteration weighting factor. A Larger value tends to increase stability, but at the cost of performance. We suggest using the default value or values between 0.6 and 0.85, but values between 0.1 and 1.0 are valid. Use 0.98 - 1.0 in very difficult cases. See "Troubleshooting DynamicWave Model Calculations" on page 8-448 for more information.
NR Iterations	Allowed maximum Newton iterations in one time step during the numerical solution process. This attribute value is used to prevent the numerical solver from unlimited iterations under situations when the solver is unable to reach convergence.
Relaxation Weighting	Displays the relaxation weighting used in calculations of your model.
Computational Distance	Displays the computational distance used in calculations of your model. See "Troubleshooting DynamicWave Model Calculations" on page 8-448 for more information.

Catchment Summary Tab

This tab displays a table of data about catchments in your model. You cannot modify the table, it is read-only. To edit a catchment, select it in the Drawing Pane and edit its attributes in the Property Editor.

The Catchment Summary tab displays the following information:

Label	Displays the label of each catchment in your model.
Runoff Method	Displays the runoff method associated with each catchment.
Loss Method	Displays the loss method associated with each catchment.
Total Rainfall Depth	Displays the total rainfall depth.
Area	Displays the catchment size.

Runoff Volume	Displays the calculated runoff volume.
Flow (Peak)	Displays the calculated peak flow.
Time to Peak	Displays how long after the simulation starts that peak runoff happens.

General Summary Tab

This tab displays a table of calculations for elements in your model. You cannot modify the table, it is read-only. To edit an element, select it in the Drawing Pane and edit its attributes in the Property Editor.

The General Summary tab displays the following information:

Label	Displays the label of each element in your model.
Element Type	Displays the kind of element being reported on.
Branch	Provides information for engineers who are troubleshooting a model or looking to simplify a particularly complex model. The Bentley SewerGEMS V8*i* calculation engine includes a heuristic routine that decomposes a network into its component branches, and each branch is solved independently using an implicit solver. Each branch comprises a series of connected elements. Elements with the same branch ID are solved together.
Time to Max Flow	Displays how long after the simulation starts that maximum flow happens.
Flow (Maximum)	Displays the calculated flow through elements.
Velocity (Maximum)	Displays the calculated velocity of flow through elements.
Maximum HGL	Displays the calculated maximum hydraulic grade line for each element.

Node Summary Tab

This tab displays a table of calculations for nodes in your model. Nodes include catch basins, manholes, outfalls, ponds (but not cross-sections, catchments, etc.). You cannot modify the table, it is read-only. To edit a node, select it in the Drawing Pane and edit its attributes in the Property Editor.

The Node Summary tab displays the following information:

Label	Displays the label of each node in your model.
Element Type	Displays the kind of node being reported on.
Branch	Provides information for engineers who are troubleshooting a model or looking to simplify a particularly complex model.The Bentley SewerGEMS V8*i* calculation engine includes a heuristic routine that decomposes a network into its component branches, and each branch is solved independently using an implicit solver. Each branch comprises a series of connected elements. Elements with the same branch ID are solved together.
Time to Max Inflow	Displays how long after the simulation starts that maximum flow into the nodes occurs.
Flow (Maximum in)	Displays the calculated flow into the nodes. (Outfalls cannot have inflow.)
Time to Maximum Inlet Flow	Displays how long after the simulation starts that maximum flow into the nodes occurs.
Flow (Surface Maximum)	Displays the maximum flow into each of the nodes of your model.
Time to Max Captured Flow	Displays how long after the simulation starts that maximum captured flow occurs in each catch basin.
Flow (Captured Maximum)	Displays the calculated captured flow in each catch basin.
Time to Maximum Overflow	Displays how long after the simulation starts that maximum overflow occurs at each of the nodes in your model.

Flow (Overflow Maximum)	Displays the maximum overflow at each of the nodes of your model.

Gutter Summary Tab

This tab displays a table of data about gutters in your model. You cannot modify the table, it is read-only. To edit a gutter, select it in the Drawing Pane and edit its attributes in the Property Editor.

The Gutter Summary tab displays the following information:

Label	Displays the label of each gutter in your model.
Open Cross Section	Displays the kind of open cross section that defines the gutter.
Flow (Maximum)	Displays the calculated flow through the gutters.
Time to Max Flow	Displays how long after the simulation starts that maximum flow happens.
Velocity (Maximum)	Displays the calculated velocity of flow through gutters.
Maximum HGL	Displays the calculated maximum hydraulic grade line for each gutter.

SWMM Engine Summary Report

When a SWMM engine run finishes it produces a summary report which is written to Notepad (as opposed to the Calculation Summary report when the Implicit engine solves). This report is a Notepad file and is not saved with the model once the file is closed. This report provides useful overall information on the run. If you want to review this result later in the session, you can minimize it instead of closing it. If you want to save it for later use, execute a File > Save command.

User Notifications

User notifications are messages about your model. These messages can warn you about potential issues with your model, such as slopes that might be too steep or elements that slope in the wrong direction. These messages also point you to errors in your model that prevent Bentley SewerGEMS V8*i* from solving your model.

To see user notifications:

1. Compute your model.
2. If needed, open the User Notification manager by clicking **View > User Notifications** (Ctrl + 3). 
3. Or, if the calculation fails to compute because of an input error, when your model is finished computing, Bentley SewerGEMS V8*i* prompts you to view user notifications to validate the input data.

   You must fix any errors identified by red circles before Bentley SewerGEMS V8*i* can compute a result.

   Errors identified by orange circles are warnings that do not prevent the computation of the model.
4. In the User Notifications manager, if a notification pertains to a particular element, you can double-click the notification to magnify and display the element in the center of the drawing pane.
5. As needed, use the element label to identify the element that generates the error and use the user notification message to edit the element's properties to resolve the error.

User Notifications Manager

The User Notifications Manager displays warnings and error messages that are turned up by Bentley SewerGEMS V8*i*'s validation routines. If the notification references a particular element, you can zoom straight to that element by either double-clicking the notification, or right-clicking it and selecting the **Zoom To** command.

- Warnings are denoted by an orange icon and do not prevent the model from calculating successfully. 
- Errors are denoted by a red icon, and the model will not successfully calculate if errors are found. 

The User Notifications Manager consists of a toolbar and a tabular view containing a list of warnings and error messages. The toolbar consists of the following buttons:

	**Details**	Displays the User Notification Details dialog box, which includes information about any warning or error messages. For more information, see “User Notification Details Dialog Box” on page 8-448.
	**Save**	Saves the user notifications as a comma-delimited .csv file. You can open the .csv file in Microsoft Excel or Notepad.
	**Report**	Displays a User Notification Report.
	**Copy**	Copies the currently highlighted warning or error message to the Windows clipboard.
	**Zoom To**	If the warning or error message is related to a specific element in your model, click this button to center the element in question in the drawing pane.
	**Help**	Displays online help for the User Notification Manager.

The User Notification Manager displays warnings and error messages in a tabular view. The table includes the following columns:

Message ID	This column displays the message ID associated with the corresponding message.
Scenario ID	This column displays the scenario associated with the corresponding message. This column will display “Base” unless you ran a different scenario.
Element Type	This column displays the element type associated with the corresponding message.
Element ID	This column displays the element ID associated with the corresponding message.
Label	If the notification is caused by a specific element, this column displays the label of the element associated with the corresponding message.

Message	This column displays the description associated with the corresponding message.
Time	If the user notification occurred during a specific time step, this column displays the time step. Otherwise, this column is left blank.
Source	This column displays the validation routine that triggered the corresponding message.

User Notification Details Dialog Box

This dialog lists the elements that are referred to by a time-sensitive user notification message. In the User Notification dialog, there is a time column that displays the time-step during which time-sensitive messages occur. These messages will say “during this time-step” or “for this time-step”, and do not display information about the referenced element or elements. Double-clicking one of these messages in the User Notifications dialog opens the User Notification Details dialog, which does provide information about the referenced element(s).

You can double-click messages in the User Notification Details dialog to zoom the drawing pane view to the referenced element.

Troubleshooting DynamicWave Model Calculations

If your model does not successfully calculate, try the following steps:

1. If you are running hydrology (rainfall on catchments) as well as hydraulics, check the outflow hydrographs from catchments to make certain they are reasonable.
2. Check the model for errors:
   – Use the Validate command and look at the warnings and/or errors that are reported.
   – When you Calculate the model, validation routines are performed that are not included during a Validate operation. Review the warnings and/or errors that are returned from both levels of validation.
   – Common data problems to look for include:
     - Incorrect channel or conduit slopes: Reasonable slopes are generally small and rarely negative. You can view slopes along a reach visually by using the Profiles feature. You can find unusually large or negative slopes through User Notifications and FlexTable reports. You can Color Code link elements by Slope and look for excessive values. If the model has parallel conduits connected by a flat (slope value of 0) conduit, try making that conduit Inactive.

    - Incorrect Channel or Conduit size: Look for unusual size changes along a reach. Color code drawing by Diameter to look for this type of discrepancy.
    - Very low flows: If flows are less than 0.01 cfs (0.001 m3/s), depths may fall below accuracy tolerances.
  - Look for areas displaying common modeling difficulties to verify input data is correct:
    - Flow splits at weirs and orifices that are dry at certain points during an extended period simulation.
    - Hydrograph rapidly changes within a short time (minutes).
    - Very sharp flood wave.
    - Near-critical slopes.
    - Significant and abrupt changes in the conduit size, shape and/or slope.
    - Looped networks.
    - Backwater up to a control structure.
    - Significant backwater conditions.
    - Flow control structures on relatively small storage nodes (ponds, wetwells).
    - System inflows vastly exceeding the system capacity resulting in mass flooding.
    - Unusually small ponds compared to their inflow.
    - Many pumping stations in the system.
    - Look for a mix of very long and very small pipes, especially when using the SWMM engine. Eliminate or combine short pipes because their effect on routing is small. Break exceptionally long pipes into mutliple pipes that are each roughly the same length as other pipes in the network.
    - Examine the User Notifications that are displayed after calculating.
    - Examine Graphs and water surface Profiles. Create Flow plots at splits and at pump discharge areas and look for jagged peaks in the plot.
3. Default values for calculation options will work for the majority of cases, but some systems need small adjustments to converge. When the calculation is moving very slowly (you can observe that the model is stuck at certain times) or the results show apparent instabilities, it is an indication that the model is experiencing difficulties in converging to a stable and robust result. Try adjusting calc options in this order:
  - Initial conditions: Options include warm start or transitional start. Try both and see if one gives better results for your system.

- Computational Distance, Calculation Time Step, and NR Weighting Coefficient: Loop through the following process:
- Try an NR Weighting Coefficient value of between 0.9 and 0.99 with the default Computational Distance and Calculation Time Step.
- Set NR back to default and try reducing the Computational Distance value.
- Set Computational Distance back to default and try reducing the Calculation Time Step value.
- Keep the Calculation Time Step the same and repeat the above steps.
- Try increasing the the NR Iterations to 20.

Note: There is no absolute rule on whether the time step or the NR weighting coefficient should be changed or to what specific value; normally you should reduce the time step and increase the NR coefficient but sometimes the opposite can also help.

4. Isolate problems areas: Isolate the problem area by incrementally deleting small sections of your model and re-computing. This may help you narrow down the source data that the engine has trouble with. It may expose data entry issues or areas that are exhibiting common modeling difficulties.
5. Determine at what time step the problem occurs. Look for what is happening at that time. Is a weir beginning to overflow? Is it the first time a large pump comes on?
6. Switch to using the SWMM numerical engine. If there are problems when using the SWMM engine, try changing the Routing Method from Dynamic Wave (default) to Kinematic Wave or Uniform Flow. These methods do not handle backups as accurately as dynamic wave but they tend to be more stable.

Note: Headlosses at nodes are ignored during periods of supercritical flow.

Chapter

9 Using Scenarios and Alternatives

Scenarios and alternatives let you create, analyze, and recall an unlimited number of variations of your model. In Bentley SewerGEMS V8*i* powerful two-level design, scenarios contain alternatives to give you precise control over changes to the model.

Understanding Scenarios and Alternatives

Scenario management in Bentley SewerGEMS V8*i* can dramatically increase your productivity in the "What If?" areas of modeling, including calibration, operations analysis, and planning.

If you've never used scenarios and alternatives before, we recommend reading all of the topics in this section to gain a complete understanding of how they work. By investing a little time now to understand management, you can avoid unnecessary editing and data duplication. Take advantage of scenario management to get a lot more out of your model, with much less work and expense.

Topics in this section include:

- "Advantages of Automated Scenario Management" on page 9-452
- "A History of What-If Analyses" on page 9-452
- "The Scenario Cycle" on page 9-454
- "Scenario Attributes and Alternatives" on page 9-456
- "A Familiar Parallel" on page 9-456
- "Inheritance" on page 9-457
- "Local and Inherited Values" on page 9-459
- "Minimizing Effort through Attribute Inheritance" on page 9-459
- "Minimizing Effort through Scenario Inheritance" on page 9-460

Advantages of Automated Scenario Management

In contrast to the old methods of scenario management (editing or copying data), automated scenario management using inheritance gives you significant advantages:

- A single project file makes it possible to generate an unlimited number of "What If?" conditions without becoming overwhelmed with numerous modeling files and separate results.
- Because the software maintains the data for all the scenarios in a single project, it can provide you with powerful automated tools for directly comparing scenario results. Any set of results is immediately available at any time.
- The Scenario / Alternative relationship empowers you to mix and match groups of data from existing scenarios without having to re-declare any data.
- With inheritance, you do not have to re-enter data if it remains unchanged in a new alternative or scenario, avoiding redundant copies of the same data. Inheritance also enables you to correct a data input error in a parent scenario and automatically update the corrected attribute in all child scenarios.

These advantages, while obvious, may not seem compelling for small projects. It is as projects grow to hundreds or thousands of network elements that the advantages of true scenario inheritance become clear. On a large project, being able to maintain a collection of base and modified alternatives accurately and efficiently can be the difference between evaluating optional improvements and being forced to ignore them.

A History of What-If Analyses

The history of what-if analyses can be divided into two periods:

- “Before Haestad Methods - Distributed Scenarios” on page 9-452
- “With Haestad Methods: Self-Contained Scenarios” on page 9-454

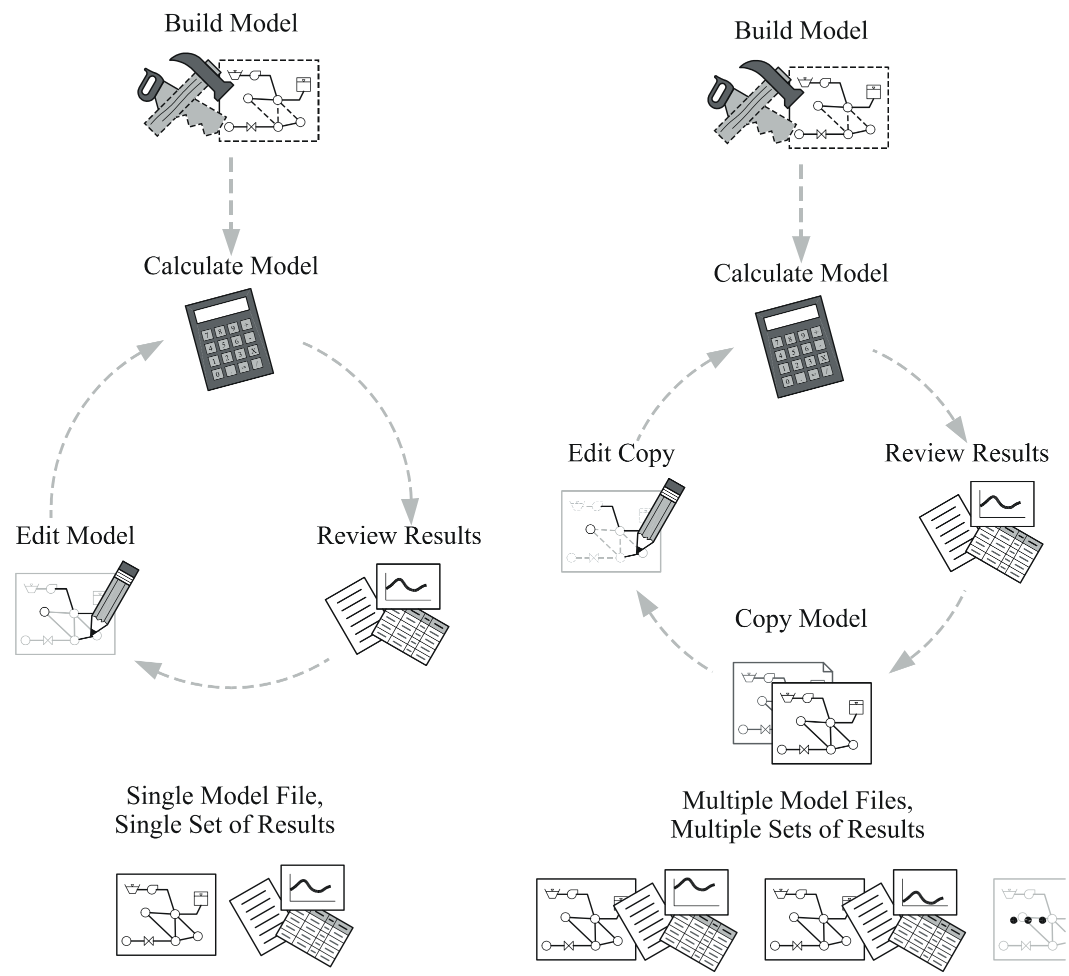
Before Haestad Methods - Distributed Scenarios

Traditionally, there have only been two possible ways of analyzing the effects of change on a software model:

- Change the model, recalculate, and review the results
- Create a copy of the model, edit that copy, calculate, and review the results

Although either of these methods may be adequate for a relatively small system, the data duplication, editing, and re-editing becomes very time-consuming and error-prone as the size of the system and the number of possible conditions increase. Also, comparing conditions requires manual data manipulation, because all output must be stored in physically separate data files.

Figure 9-1: Before Haestad Methods: Manual Scenarios



With Haestad Methods: Self-Contained Scenarios

Effective scenario management tools need to meet these objectives:

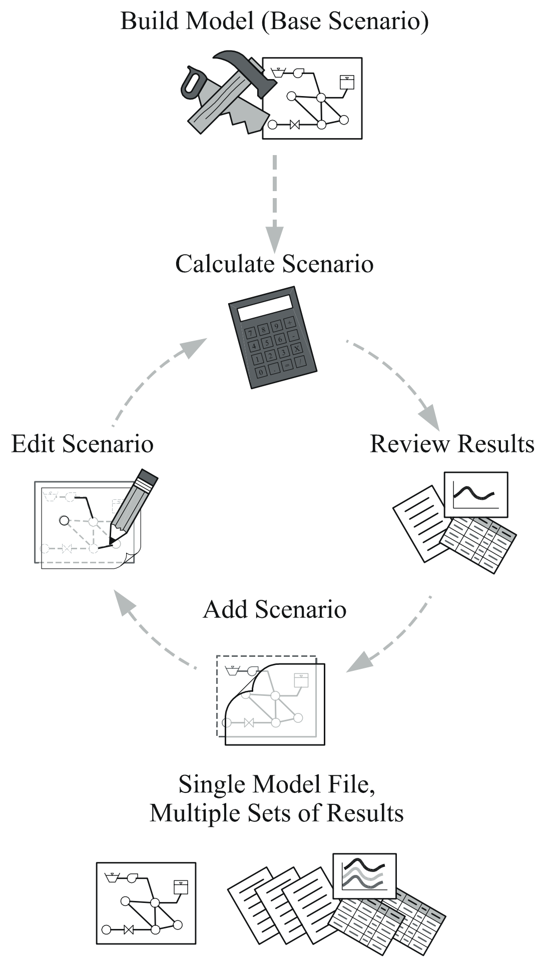
- Minimize the number of project files the modeler needs to maintain (one, ideally).
- Maximize the usefulness of scenarios through easy access to things such as input and output data, and direct comparisons.
- Maximize the number of scenarios you can simulate by mixing and matching data from existing scenarios (data reuse)
- Minimize the amount of data that needs to be duplicated to consider conditions that have a lot in common

The scenario management feature in Bentley SewerGEMS V8*i* successfully meets all of these objectives. A single project file enables you to generate an unlimited number of What If? conditions, edit only the data that needs to be changed, and quickly generate direct comparisons of input and results for desired scenarios.

The Scenario Cycle

The process of working with scenarios is similar to the process of manually copying and editing data, but without the disadvantages of data duplication and troublesome file management. This process lets you cycle through any number of changes to the model, without fear of overwriting critical data or duplicating important information. Of course, it is possible to directly change data for any scenario, but an audit trail of scenarios can be useful for retracing the steps of a calibration series or for understanding a group of master plan updates.

Figure 9-2: Before Haestad Methods: Manual Scenarios


Build Model (Base Scenario)
Calculate Scenario
Edit Scenario
Review Results
Add Scenario
Single Model File,
Multiple Sets of Results

Scenario Attributes and Alternatives

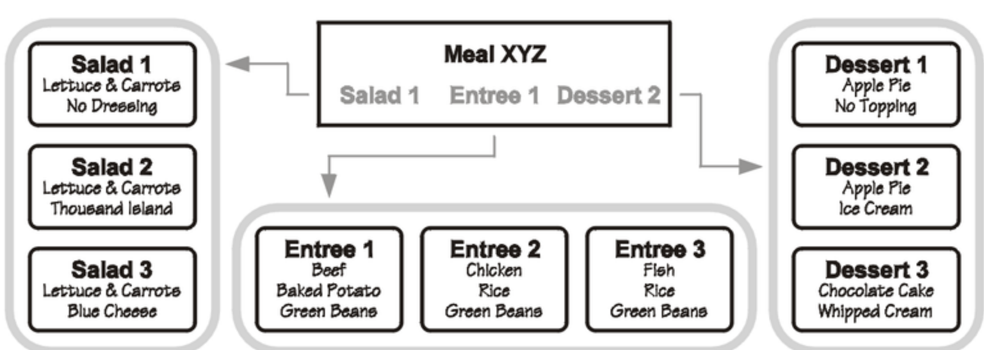
Before we explore scenario management further, a few key terms should be defined:

- **Attribute**—An attribute is a fundamental property of an object, and is often a single numeric quantity. For example, the attributes of a pipe include diameter, length, and roughness.
- **Alternative**—An alternative holds a family of related attributes so pieces of data that you are most likely to change together are grouped for easy referencing and editing. For example, a physical properties alternative groups physical data for the network's elements, such as elevations, sizes, and roughness coefficients.
- **Scenario**—A scenario has a list of referenced alternatives (which hold the attributes), and combines these alternatives to form an overall set of system conditions that can be analyzed. This referencing of alternatives enables you to easily generate system conditions that mix and match groups of data that have been previously created. Note that scenarios do not actually hold any attribute data—the referenced alternatives do.

A Familiar Parallel

Although the structure of scenarios may seem a bit difficult at first, anyone who has eaten at a restaurant should be able to relate fairly easily. A meal (scenario) is comprised of several courses (alternatives), which might include a salad, an entrée, and a dessert. Each course has its own attributes. For example, the entrée may have a meat, a vegetable, and a starch. Examining the choices, we could present a menu as in the following figure:

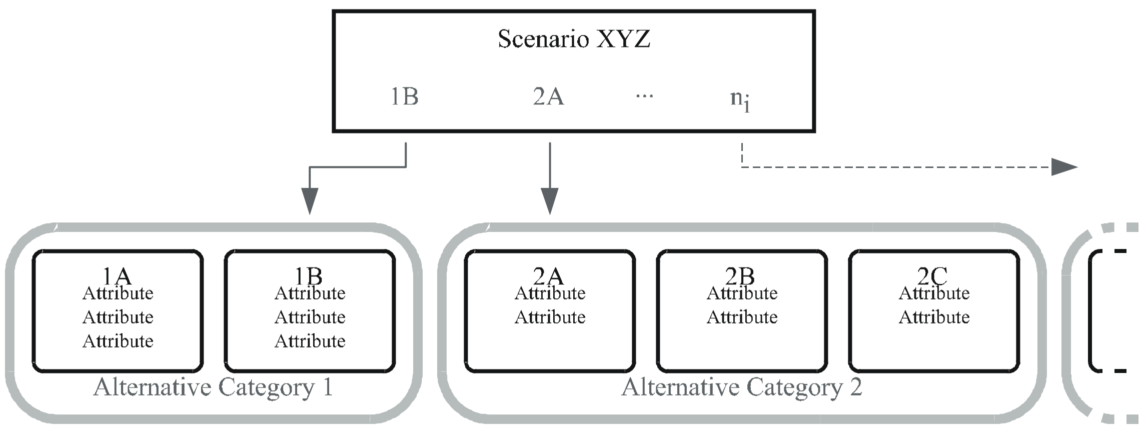
Figure 9-3: A Restaurant Meal Scenario



The restaurant does not have to create a new recipe for every possible meal (combination of courses) that could be ordered. They can just assemble any meal based on what the customer orders for each alternative course. Salad 1, Entrée 1, and Dessert 2 might then be combined to define a complete meal.

Generalizing this concept, we see that any scenario references one alternative from each category to create a big picture that can be analyzed. Note that different types of alternatives may have different numbers and types of attributes, and any category can have an unlimited number of alternatives to choose from.

Figure 9-4: Generic Scenario Anatomy



Inheritance

The separation of scenarios into distinct alternatives (groups of data) meets one of the basic goals of scenario management: maximizing the number of scenarios you can develop by mixing and matching existing alternatives. Two other primary goals have also been addressed: a single project file is used, and easy access to input data and calculated results is provided in numerous formats through the intuitive graphical interface.

But what about the other objective: minimizing the amount of data that needs to be duplicated to consider conditions that have a lot of common input? Surely an entire set of pipe diameters should not be re-specified if only one or two change?

The solution is a familiar concept to most people: inheritance.

In the natural world, a child inherits characteristics from a parent. This may include such traits as eye-color, hair color, and bone structure. There are two significant differences between the genetic inheritance that most of us know and the way inheritance is implemented in software:

- "Overriding Inheritance" on page 9-458
- "Dynamic Inheritance" on page 9-458

Overriding Inheritance

Overriding inheritance is the software equivalent of cosmetics. A child can override inherited characteristics at any time by specifying a new value for that characteristic. These overriding values do not affect the parent, and are therefore considered local to the child. Local values can also be removed at any time, reverting the characteristic to its inherited state. The child has no choice in the value of his inherited attributes, only in local attributes.

For example, suppose a child has inherited the attribute of blue eyes from his parent. Now the child puts on a pair of green- tinted contact lenses to hide his natural eye color. When the contact lenses are on, we say his natural eye color is overridden locally, and his eye color is green. When the child removes the tinted lenses, his eye color instantly reverts to blue, as inherited from his parent.

Dynamic Inheritance

Dynamic inheritance does not have a parallel in the genetic world. When a parent's characteristic is changed, existing children also reflect the change. Using the eye-color example, this would be the equivalent of the parent changing eye color from blue to brown, and the children's eyes instantly inheriting the brown color also. Of course, if the child has already overridden a characteristic locally, as with the green lenses, his eyes will remain green until the lenses are removed. At this point, his eye color will revert to the inherited color, now brown.

This dynamic inheritance has remarkable benefits for applying wide-scale changes to a model, fixing an error, and so on. If rippling changes are not desired, the child can override all of the parent's values, or a copy of the parent can be made instead of a child.

Local and Inherited Values

Any changes that are made to the model belong to the currently active scenario and the alternatives that it references. If the alternatives happen to have children, those children will also inherit the changes unless they have specifically overridden that attribute. The following figure demonstrates the effects of a change to a mid-level alternative. Inherited values are shown as gray text, local values are shown as black text.

Figure 9-5: A Mid-level Hierarchy Alternative Change

Minimizing Effort through Attribute Inheritance

Inheritance has an application every time you hear the phrase, "just like x except for y." Rather than specifying all of the data from x again to form this new condition, we can create a child from x and change y appropriately. Now we have both conditions, with no duplicated effort.

We can even apply this inheritance to our restaurant analogy as follows. Inherited values are shown as gray text, local values are shown as black text.

Note: **Salad 3 could inherit from Salad 2, if we prefer: "Salad 3 is just like Salad 2, except for the dressing."**

Salad Alternative Hierarchy	Attribute: Vegetables	Attribute: Dressing
Salad 1	Lettuce & Carrots	No Dressing
├ Salad 2	Lettuce & Carrots	Thousand Island
└ Salad 3	Lettuce & Carrots	Blue Cheese

- "Salad 2 is just like Salad 1, except for the dressing."
- "Salad 3 is just like Salad 1, except for the dressing."

Note: If the vegetable of the day changes (say from green beans to peas), only Entrée 1 needs to be updated, and the other entrées will automatically inherit the vegetable attribute of "Peas" instead of "Green Beans."

Entree Alternative Hierarchy	Attribute: Meat	Attribute: Starch	Attribute: Vegetable
Entree 1 └ Entree 2 └ Entree 3	Beef Chicken Fish	Baked Potato Rice Rice	Green Beans Green Beans Green Beans

- "Entrée 2 is just like Entrée 1, except for the meat and the starch."
- "Entrée 3 is just like Entrée 2, except for the meat."

Note: Dessert 3 has nothing in common with the other desserts, so it can be created as a "root" or base alternative. It does not inherit its attribute data from any other alternative.

Dessert Alternative Hierarchy	Attribute: Bakery Item	Attribute: Topping
Dessert 1 └ Dessert 2 Dessert 3	Apple Pie Apple Pie Chocolate Cake	No Topping Ice Cream Whipped Cream

- "Dessert 2 is just like Dessert 1, except for the topping."

Minimizing Effort through Scenario Inheritance

Just as a child alternative can inherit attributes from its parent, a child scenario can inherit which alternatives it references from its parent. This is essentially still the phrase just like x except for y, but on a larger scale.

Carrying through on our meal example, consider a situation where you go out to dinner with three friends. The first friend places his order, and the second friend orders the same thing except for the dessert. The third friend orders something totally different, and you order the same meal as hers except for the salad.

The four meal scenarios could then be presented as follows (inherited values are shown as gray text, local values are shown as black text):

- "Meal 2 is just like Meal 1, except for the dessert." The salad and entrée alternatives are inherited from Meal 1.

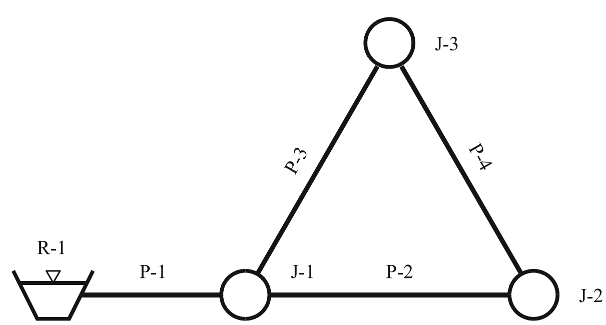
Meal Scenario Hierarchy	Salad Alternative	Entree Alternative	Dessert Alternative
Meal 1 └ Meal 2 Meal 3 └ Meal 4	Salad 1 Salad 1 Salad 3 Salad 2	Entree 2 Entree 2 Entree 3 Entree 3	Dessert 3 Dessert 1 Dessert 2 Dessert 2

- "Meal 3 is nothing like Meal 1 or Meal 2." A totally new base or root is created.
- "Meal 4 is just like Meal 3, except for the salad." The entrée and dessert alternatives are inherited from Meal 3.

Scenario Example - Simple Water Distribution System

Let us consider a fairly simple water distribution system: a single reservoir supplies water by gravity to three junction nodes.

Figure 9-6: Example Water Distribution System



Although true water distribution scenarios include such alternative categories as initial settings, operational controls, water quality, and fire flow, we are going to focus on the two most commonly changed sets of alternatives: demands and physical properties. Within these alternatives, we are going to concentrate on junction baseline demands and pipe diameters.

Building the Model (Average Day Conditions)

During model construction, probably only one alternative from each category is going to be considered. This model is built with average demand calculations and preliminary pipe diameter estimates. At this point we can name our scenario and alternatives, and the hierarchies look like the following (showing only the items of interest):

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	*100 gpm*	*500 gpm*	*100 gpm*

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	*8 inches*	*6 inches*	*6 inches*	*6 inches*

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	*Average Day*	*Preliminary Pipes*

Analyzing Different Demands (Maximum Day Conditions)

In our example, the local planning board also requires analysis of maximum day demands, so a new demand alternative is required. No variation in demand is expected at J-2, which is an industrial site. As a result, the new demand alternative can inherit J-2's demand from Average Day while the other two demands are overridden.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day └ *Maximum Day*	100 gpm *200 gpm*	500 gpm 500 gpm	100 gpm *200 gpm*

Now we can create a child scenario from Average Day that inherits the physical alternative, but overrides the selected demand alternative. As a result, we get the following scenario hierarchy:

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day └ *Max. Day*	Average Day *Maximum Day*	Preliminary Pipes Preliminary Pipes

Since no physical data (pipe diameters) have been changed, the physical alternative hierarchy remains the same as before.

Another Set of Demands (Peak Hour Conditions)

Based on pressure requirements, the system is adequate to supply maximum day demands. Another local regulation requires analysis of peak hour demands, with slightly lower allowable pressures. Since the peak hour demands also share the industrial load from the Average Day condition, Peak Hour can be inherited from Average Day. In this instance, Peak Hour could inherit just as easily from Maximum Day.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day ├ Maximum Day └ *Peak Hour*	100 gpm 200 gpm *250 gpm*	500 gpm 500 gpm 500 gpm	100 gpm 200 gpm *250 gpm*

Another scenario is also created to reference these new demands, as shown below:

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day ├ Max. Day └ *Peak*	Average Day Maximum Day *Peak Hour*	Preliminary Pipes Preliminary Pipes Preliminary Pipes

Note again that we did not change any physical data, so the physical alternatives remain the same.

Correcting an Error

This analysis results in acceptable pressures, until it is discovered that the industrial demand is not actually 500 gpm—it is 1,500 gpm. Because of the inheritance within the demand alternatives, however, only the Average Day demand for J-2 needs to be updated. The changes ripple through to the children. After the single change is made, the demand hierarchy is as follows:

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day ├ Maximum Day └ Peak Hour	100 gpm 200 gpm 250 gpm	*1,500 gpm* 1,500 gpm 1,500 gpm	100 gpm 200 gpm 250 gpm

Notice that no changes need to be made to the scenarios to reflect these corrections. The three scenarios can now be calculated as a batch to update the results.

When these results are reviewed, it is determined that the system does not have the ability to adequately supply the system as it was originally thought. The pressure at J-2 is too low under peak hour demand conditions.

Analyzing Improvement Suggestions

To counter the headloss from the increased demand load, two possible improvements are suggested:

- A much larger diameter is proposed for P-1 (the pipe from the reservoir). This physical alternative is created as a child of the Preliminary Pipes alternative, inheriting all the diameters except P-1's, which is overridden.
- Slightly larger diameters are proposed for all pipes. Since there are no commonalities between this recommendation and either of the other physical alternatives, this can be created as a base (root) alternative.

These changes are then incorporated to arrive at the following hierarchies:

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	8 inches	6 inches	6 inches	6 inches

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
├ Max. Day	Maximum Day	Preliminary Pipes
└ Peak	Peak Hour	Preliminary Pipes
├ *Peak, Big P-1*	Peak Hour	*Larger P-1*
└ *Peak, All Big Pipes*	Peak Hour	*Larger All Pipes*

This time, the demand alternative hierarchy remains the same since no demands were changed. The two new scenarios (Peak, Big P-1, Peak, All Big Pipes) can be batch run to provide results for these proposed improvements.

Next, features like Scenario Comparison Annotation (from the Scenario Manager) and comparison Graphs (for extended period simulations, from the element editor dialog boxes) can be used to directly determine which proposal results in the most improved pressures.

Finalizing the Project

It is decided that enlarging P-1 is the optimum solution, so new scenarios are created to check the results for average day and maximum day demands. Notice that this step does not require handling any new data. All of the information we want to model is present in the alternatives we already have!

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
├ Max. Day	Maximum Day	Preliminary Pipes
└ *Max. Day, Big P-1*	Maximum Day	*Larger P-1*
├ Peak	Peak Hour	Preliminary Pipes
├ Peak, Big P-1	Peak Hour	Larger P-1
└ Peak, All Big Pipes	Peak Hour	Larger All Pipes
└ *Avg. Day, Big P-1*	Average Day	*Larger P-1*

Also note that it would be equally effective in this case to inherit the Avg. Day, Big P-1 scenario from Avg. Day (changing the physical alternative) or to inherit from Peak, Big P-1 (changing the demand alternative). Likewise, Max. Day, Big P-1 could inherit from either Max. Day or Peak, Big P-1.

Neither the demand nor physical alternative hierarchies were changed in order to run the last set of scenarios, so they remain as they were.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	100 gpm	1,500 gpm	100 gpm
├ Maximum Day	200 gpm	1,500 gpm	200 gpm
└ Peak Hour	250 gpm	1,500 gpm	250 gpm

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	8 inches	6 inches	6 inches	6 inches
└ *Larger P-1*	*18 inches*	6 inches	6 inches	6 inches
Larger All Pipes	*12 inches*	*12 inches*	*12 inches*	*12 inches*

Summary

In contrast to the old methods of scenario management (editing or copying data), automated scenario management using inheritance gives you significant advantages:

- A single project file makes it possible to generate an unlimited number of What If? conditions without becoming overwhelmed with numerous modeling files and separate results.

- Because the software maintains the data for all the scenarios in a single project, it can provide you with powerful automated tools for directly comparing scenario results. Any set of results is immediately available at any time.
- The Scenario / Alternative relationship empowers you to mix and match groups of data from existing scenarios without having to re-declare any data.
- With inheritance, you do not have to re-enter data if it remains unchanged in a new alternative or scenario, avoiding redundant copies of the same data. Inheritance also enables you to correct a data input error in a parent scenario and automatically update the corrected attribute in all child scenarios.

These advantages, while obvious, may not seem compelling for small projects. It is as projects grow to hundreds or thousands of network elements that the advantages of true scenario inheritance become clear. On a large project, being able to maintain a collection of base and modified alternatives accurately and efficiently can be the difference between evaluating optional improvements and being forced to ignore them.

To learn more about actually using scenario management in our software, start by running the scenario management tutorial from the Help menu or from within the scenario manager itself. Then load one of the SAMPLE projects and explore the scenarios defined there. For context-sensitive help, press **F1** or the **Help** button any time there is a screen or field that puzzles you.

Scenarios

A Scenario contains all the input data (in the form of Alternatives), calculation options, results, and notes associated with a set of calculations. Scenarios let you set up an unlimited number of “What If?” situations for your model, and then modify, compute, and review your system under those conditions.

You can create scenarios that reuse or share data in existing alternatives, submit multiple scenarios for calculation in a batch run, switch between scenarios, and compare scenario results—all with a few mouse clicks. There is no limit to the number of scenarios that you can create.

Base and Child Scenarios

Note: The calculation options are not inherited between scenarios, but are duplicated when the scenario is first created. The alternatives and data records, however, are inherited. There is a permanent, dynamic link from a child back to its parent.

There are two types of scenarios:

- **Base Scenarios**—Contain all of your working data. When you start a new project, you begin with a default base scenario. As you enter data and calculate your model, you are working with this default base scenario and the alternatives it references.
- **Child Scenarios**—Inherit data from a base scenario, or other child scenarios. Child scenarios allow you to freely change data for one or more elements in your system. Child scenarios can reflect some or all of the values contained in their parent. This is a very powerful concept, giving you the ability to make changes in a parent scenario that will trickle down through child scenarios, while also giving you the ability to override values for some or all of the elements in child scenarios.

Creating Scenarios

You create new scenarios in the Scenario Manager. A new scenario can be a Base scenario or a Child scenario. For information about the differences between the two types of scenarios, see "Base and Child Scenarios" on page 9-467.

To create a new scenario:

1. Select **View** > **Scenarios** to open the Scenario Manager, or click the Scenario Manager tab.
2. Click the **New** button and select whether you want to create a Base scenario or a Child Scenario. When creating a Child scenario, you must first highlight the scenario from which the child is derived in the Scenario Manager tree view.

   By default, a new scenario comprises the Base Alternatives associated with each alternative type.
3. Double-click the new scenario to edit its properties in the Property Editor.

Editing Scenarios

You edit scenarios in two places in Bentley SewerGEMS V8*i*:

- The Scenario Manager lists all of the project's scenarios in a hierarchical tree format, and displays the Base/Child relationship between them.
- The Property Editor displays the alternatives that make up the scenario that is currently highlighted in the Scenario Manager, along with the scenario label, any notes associated with the scenario, and the calculation options profile that is used when the scenario is calculated.

To edit a scenario:

1. Select **View** > **Scenarios** to open the Scenario Manager, or click the Scenario Manager tab.
2. Double-click the scenario you want to edit to display its properties in the Property Editor.
3. Edit any of the following properties as desired:
   – Scenario label - This is the same operation as renaming the scenario in the Scenario Manager.
   – Notes - Add any notes or comments in the Notes field
   – Alternatives
   – Calculation Options

Running Multiple Scenarios at Once (Batch Runs)

Performing a batch run lets you set up and run calculations for multiple scenarios at once. This is helpful if you want to queue a large number of calculations, or manage a group of smaller calculations as a set. The list of selected scenarios for the batch run remain with your project until you change it.

To perform a batch run:

1. Selecting **View** > **Scenarios** to open the Scenario Manager, or click the Scenario Manager tab.
2. Click the **Compute Current Scenario** button, then select **Batch Run** from the shortcut menu.

   The Batch Run Editor dialog box appears.
3. Check the scenarios you want to run, then click the **Batch** button. Each scenario is calculated. You can cancel the batch run between any scenario calculation. The selected scenarios run consecutively.

When the batch run is completed, the scenario that was current stays current, even if it was not calculated.

4. Select a calculated scenario from the Scenario toolbar drop-down list to see the results throughout the program.

Batch Run Editor Dialog Box

The Batch Run Editor dialog box contains the following controls:

Scenario List	Displays a list of all current scenarios. Click the check box next to the scenarios you want to run in batch mode.
Batch	Starts the batch run of the selected scenarios.
Select	Displays a drop-down menu containing the following commands: • **Select All** - Selects all scenarios listed. • **Clear Selection** - Clears all selected scenarios.
Close	Closes the Batch Run Editor dialog box.
Help	Displays context-sensitive help for the Batch Run Editor dialog box.

Scenario Manager

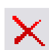
The Scenario Manager lets you create, edit, and manage scenarios. There is one built-in default scenario—the Base scenario. If you wish, you only have to use this one scenario. However, you can save yourself time by creating additional scenarios that reference the alternatives needed to perform and recall the results of each of your calculations. There is no limit to the number of scenarios that you can create.

Note: When you delete a scenario, you are not losing data records because scenarios never actually hold calculation data records (alternatives do). The alternatives and data records referenced by that scenario exist until you explicitly delete them. By accessing the Alternative Manager, you can delete the referenced alternatives and data records.

The Scenario Manager consists of a hierarchical tree view and a toolbar.

The tree view displays all of the scenarios in the project. If the Property Editor is open, highlighting a scenario in the list causes the alternatives that make up the scenario appear there. If the Property Editor is not open, you can display the alternatives and scenario information by highlighting the desired scenario and clicking the Properties button in the Scenario Manager.

The toolbar contains the following controls:

	**New Scenario**	Opens a submenu containing the following commands: • **Child Scenario**—Lets you create a new Child scenario from the currently highlighted Base scenario. • **Base Scenario**—Lets you create a new Base scenario.
	**Delete**	Removes the currently selected scenario.
	**Rename**	Lets you rename the currently selected scenario.
	**Go**	Opens a submenu containing the following commands: • **Scenario**—Lets you calculate the currently highlighted scenario. • **Hierarchy**—Lets you calculate the entire currently highlighted branch—the Base scenario and all Child scenarios currently associated with it. • **Children**—Lets you calculate all of the Child scenarios associated with the currently highlighted scenario. • **Batch Run**—Lets you run a user-defined group of scenarios at once.
	**Make Current**	Causes the currently selected scenario to become the active one, and to be displayed in the drawing pane.
	**Help**	Displays online help for the Alternative Manager.

Alternatives

Alternatives are the building blocks behind scenarios (for more information, see "Scenarios" on page 9-466). They are categorized data sets that create scenarios when placed together. Alternatives hold the input data in the form of records. A record holds the data for a particular element in your system.

Scenarios are composed of alternatives, as well as other calculation options (see "Calculation Options" on page 9-497), allowing you to compute and compare the results of various changes to your system. Alternatives can vary independently within scenarios, and can be shared between scenarios.

Scenarios allow you to specify the alternatives you wish to analyze. In combination with scenarios, you can perform calculations on your system to see what effect each alternative has. Once you have determined an alternative that works best for your system, you can permanently merge changes from the preferred alternative to the base alternative if you wish.

When you first set up your system, the data that you enter is stored in the various base alternative types. If you wish to see how your system behaves, for example, by increasing the diameter of a few select pipes, you can create a child alternative to accomplish that. You can make another child alternative with even larger diameters, and another with smaller diameters. There is no limit to the number of alternatives that you can create.

Types of Alternatives

The exact properties of each alternative are discussed in their respective sections. By breaking up alternatives into these different types, we give you the ability to mix different alternatives any way that you want within any given scenario.

Bentley SewerGEMS V8*i* includes these types of alternatives:

- "Active Topology Alternative" on page 9-475
- "Physical Alternatives" on page 9-478
- "Boundary Condition Alternatives" on page 9-505
- "Initial Conditions Alternative" on page 9-507
- "Hydrology Alternatives" on page 9-508
- "Output Alternatives" on page 9-514
- "Inflow Alternatives" on page 9-516
- "Rainfall Runoff Alternative" on page 9-520
- "Water Quality Alternative" on page 9-522

Base and Child Alternatives

There are two kinds of alternatives: Base alternatives and Child alternatives. Base alternatives contain local data for all elements in your system. Child alternatives inherit data from base alternatives, or even other child alternatives, and contain data for one or more elements in your system. The data within an alternative consists of data inherited from its parent, and the data altered specifically by you (local data).

Remember that all data inherited from the base alternative are changed when the base alternative changes. Only local data specific to a child alternative remain unchanged.

Creating Alternatives

New alternatives are created in the Alternative Manager dialog box. A new alternative can be a Base scenario or a Child scenario. Each alternative type contains a Base alternative in the Alternative Manager tree view.

Note: For information regarding the differences between the two types of alternatives, see "Base and Child Alternatives" on page 9-472**.**

To create a new Alternative:

1. Select **View > Alternatives** to open the Alternative Manager, or click the Alternative Manager tab.
2. To create a new Base alternative, highlight the type of alternative you want to create, then click the **New** button.
3. To create a new Child alternative, right-click the Base alternative from which the child will be derived, then select **New > Child Alternative** from the submenu.
4. Double-click the new alternative to edit its properties in the Alternative Editor.

Editing Alternatives

You edit the properties of an alternative in its own alternative editor. The first column in an alternative editor contains check boxes, which indicate the records that have been changed in this alternative.

- If the box is checked, the record on that line has been modified and the data is local, or specific, to this alternative.
- If the box is not checked, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

To edit an existing alternative, you can use one of two methods:

- Double-click the alternative to be edited in the Alternative Manager.

or

- Highlight the alternative to be edited in the Alternative Manager and click the **Properties** button.

In either case, the Alternative Editor dialog box for the specified alternative appears, allowing you to view and define settings as desired.

Alternative Manager

The Alternative Manager lets you create, view, and edit the alternatives that make up the project scenarios. The dialog box consists of a pane that displays folders for each of the alternative types which can be expanded to display all of the alternatives for that type, and a toolbar.

The toolbar consists of the following buttons:

	**New**	Opens a submenu containing the following commands: • **Base Alternative**—Creates a new Base Alternative of the currently highlighted type. • **Child Alternative**—Creates a new Child Alternative from the currently highlighted Base Alternative.
	**Delete**	Deletes the currently highlighted alternative.

	**Properties**	Opens the Alternative Editor dialog box for the currently highlighted alternative.
	**Rename**	Lets you rename the currently highlighted alternative.
	**Report**	Lets you generate a report of the currently highlighted alternative.
	**Help**	Displays online help for the Alternative Manager.

Alternative Editor Dialog Box

This dialog box presents in tabular format the data that makes up the alternative being edited. Depending on the alternative type, the dialog box contains a separate tab for each element that possesses data contained in the alternative.

Note: Note: As you make changes to records, the check box automatically becomes checked. If you want to reset a record to its parent's values, clear the corresponding check box.

Many columns support Global Editing (see Globally Editing Data), allowing you to change all values in a single column. Right-click a column header to access the Global Edit option.

The check box column is disabled when you edit a base alternative.

The Alternative Editor displays all of the records held by a single alternative. These records contain the values that are active when a scenario referencing this alternative is active. They allow you to view all of the changes that you have made for a single alternative. They also allow you to eliminate changes that you no longer need.

There is one editor for each alternative type. Each type of editor works similarly and allows you to make changes to a different aspect of your system. The first column contains check boxes, which indicate the records that have been changed in this alternative.

If the check box is selected, the record on that line has been modified and the data is local, or specific, to this alternative.

If the check box is cleared, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

Active Topology Alternative

The Active Topology Alternative lets you temporarily remove areas of the network from the current analysis. This is useful for comparing the effect of proposed construction and to gauge the effectiveness of redundancy that may be present in the system.

The Active Topology dialog box is divided into tabs for each element type:

- Pump
- Manhole
- Catch basin
- Outfall
- Pond outlet structure
- Cross section node
- Wet well
- Pressure junction
- Junction chamber
- Conduit
- Channel
- Gutter
- Pressure Pipe
- Catchment
- Pond

For each tab, the same setup applies—the tables are divided into three columns. The first column displays whether the data is Base or Inherited, the second column is the element Label, and the third column allows you to choose whether or not the corresponding element is Active in the current alternative.

To make an element Inactive in the current alternative, clear the check box in the Is Active? field that corresponds to that element’s Label.

Creating an Active Topology Child Alternative

When creating an active topology child alternative, you may notice that the elements added to the child scenario also appear in your model when the base scenario is the current scenario.

To create an active topology alternative so that the elements added to the child scenario do not show up as part of the base scenario:

1. Create a new project.
2. Open the Property Editor.
3. Open the Scenario Manager and make sure the Base scenario is current (active).
4. Create your model by adding elements in the drawing pane.
5. Create a new child scenario and a new child active topology alternative as described in the following steps:
    a. In the Scenario Manager, click the **New** button and select **Child Scenario** from the submenu.
    b. In the Property Editor, which should now display the properties for the newly created scenario, select **<New...>** in the Active Topology field.
    c. In the Create New Alternative dialog box, type the name of the new child active topology alternative name then click **OK**.
6. In the Scenario Manager, select the new child scenario then click the **Make Current** button to make the child scenario the current (active) scenario.
7. Add new elements to your model. These elements will be active only in the new child alternative.
8. To verify that this worked as expected:
    a. In the Scenario Manager, select the base scenario then click the **Make Current** button to make the base scenario the current (active) scenario. The new elements are shown as inactive (they appear grayed out in the drawing pane).
    b. In the Scenario Manager, select the new child scenario then click the **Make Current** button to make the child scenario the current (active) scenario. The new elements are shown as active.

Note: If you add new elements in the base scenario, they will show up in the child scenario. This is normal.

Active Topology Selection Dialog Box

While it is possible to make elements active or inactive by:

1. Checking or unchecking the "Is active?" box in the alternative manager under the Active Topology Manager,
2. Unchecking the "Is active?" box in a FlexTable, or
3. Picking True of False in property grid next to "Is active?" for individual elements,

another way of making elements active or inactive is the Active Topology Selection Tool, which is accessed through Tools>Active Topology Selection.

When the user starts Active Topology Selection, a Select tool opens. Clicking elements in the drawing view while the selection tool is enabled can make them active or inactive according to the commands below.

Making an element "inactive" means that the element remains in the data file but it is not included in any hydraulic analysis calculations. Inactive elements will appear in FlexTables but calculated values will be set to NA.

Changing the active status using this tool only affects the Active Topology Alternative of the current scenario.

The Select tool consists of the following controls:

- **Done**: Select Done when you are finished selecting elements. This brings the user back to the drawing pane.
- **Add**: When this button is selected, clicking elements highlights the elements and makes them Inactive. Clicking on an element that is already inactive causes the tool to give a beep and the element remains inactive.
- **Remove**: While in this mode, clicking on elements that are inactive deselects them, making them Active. Clicking on active elements has no effect.
- **Clear**: Clicking on this button causes all elements to become active in the current scenario.

Right clicking while the Selection tool is open (i.e. opening the right click context menu) brings up a list which enables the user to switch between Add, Remove or Done.

Note: Selecting a node element to become Inactive will also select all adjacent pipes to become Inactive. This is because all pipes must end at a node.

In AutoCAD mode, you cannot use the right-click context menu command Repeat to re-open the Active Topology Selection dialog box.

Physical Alternatives

Each type of network element has a specific set of physical properties that are stored in a physical properties alternative, as listed below:

- "Physical Alternative for Pumps" on page 9-478
- "Physical Alternative for Manholes" on page 9-480
- "Physical Alternative for Catch Basins" on page 9-482
- "Physical Alternative for Outfalls" on page 9-483
- "Physical Alternative for Pond Outlet Structures" on page 9-484
- "Physical Alternative for Cross Section Nodes" on page 9-484
- "Physical Alternative for Wet Wells" on page 9-487
- "Physical Alternative for Pressure Junctions" on page 9-488
- "Physical Alternative for Junction Chambers" on page 9-488
- "Physical Alternative for Conduits" on page 9-489
- "Physical Alternative for Channels" on page 9-498
- "Physical Alternative for Gutters" on page 9-500
- "Physical Alternative for Pressure Pipes" on page 9-503
- "Physical Alternative for Ponds" on page 9-501

Physical Alternative for Pumps

The physical alternative editor for pumps is used to create various data sets for the physical characteristics of pumps. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Pump Curve	Allows you to select the pump curve that is associated with each pump in the alternative. Clicking the **Ellipsis** (…) button opens the Pump Curve Definitions dialog box.
Elevation	Displays the ground elevation for each node in the alternative.

Physical Alternative for Manholes

The physical alternative editor for manholes is used to create various data sets for the physical characteristics of manholes. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Bolted Cover	Indicates that the associated manhole has a bolted cover. If the manhole cover is bolted, then the hydraulic grade line is not reset to the rim elevation at the downstream end of the upstream pipes in the case of a flooding situation (the calculated HGL being higher than the rim elevation).
Width	Displays the width of each box manhole in the alternative.
Length	Displays the length of each box manhole in the alternative.
Diameter	Displays the diameter of each circular manhole in the alternative.
Structure Shape Type	Indicates whether the manhole is circular or box shaped. Clicking a field displays a list box that allows you to switch between the two.
Area (Constant Surface)	Lets you define the area in which ponding occurs at the currently selected element. It is available only when the Surface Storage Type attribute is set to Ponded Area.
Surface Storage Type	Contains list boxes for each manhole that allow you to specify whether the manhole uses the Default Storage Equation or a Surface Depth-Area Curve.

Column	Description
Surface Depth Area Curve	Clicking the **Ellipsis** (…) button in this field opens the Surface Depth-Area Curve editor, allowing you to define the Surface Depth- Area curve for each element in the alternative that uses the Surface Depth-Area Curve Surface Storage Type.
Elevation (Rim)	Lets you define the top elevation of a manhole structure. This elevation is typically flush with the ground surface. However in some cases, the rim elevation may be slightly below the ground surface elevation (sunk) or slightly above the ground surface elevation (raised).
Set Rim to Ground Elevation	Enables or disables a data entry shortcut. If the box is checked, the manhole rim elevation is set equal to the ground elevation automatically.
Elevation (Invert)	Lets you define the elevation at the bottom of the manhole.
Elevation (Ground)	Displays the ground elevation for each node in the alternative.
Headloss Coefficient	Lets you define the headloss coefficient of each manhole in the alternative. This column is only available for manholes that use the Standard Headloss method.
Headloss Coefficient Start	Lets you define the headloss coefficient of the element upstream of the manhole. This column is only available for manholes that use the Generic Headloss method.
Headloss Coefficient Stop	Lets you define the headloss coefficient of the element downstream of the manhole. This column is only available for manholes that use the Generic Headloss method.
Absolute Headloss	Lets you define the absolute headloss of manholes in the alternative. This column is only available for manholes that use the Absolute Headloss method.

Column	Description
Headloss Method	Lets you specify the headloss method that is used by the associated manhole.
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Physical Alternative for Catch Basins

The physical alternative editor for catch basins is used to create various data sets for the physical characteristics of catch basins. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Capture Curve	Clicking the **Ellipsis** (…) button in this field opens the Inflow Capture Curve editor, allowing you to define the inflow capture curve for each element in the alternative that uses the Inflow Capture Curve Inlet Type.
Flow (Maximum in)	Lets you define the maximum inflow for catch basins of the Maximum Inflow Inlet Type.
Inlet Type	Indicates whether the catch basin is Rating Curve or Maximum Inflow type. Clicking a field activates a list box allowing you to switch between the two.
Width	Displays the width of each box catch basin in the alternative.
Length	Displays the length of each box catch basin in the alternative.

Column	Description
Diameter	Displays the diameter of each circular catch basin in the alternative.
Structure Shape Type	Indicates whether the catch basin is circular or box shaped. Clicking a field activates a list box that allows you to switch between the two.
Surface Depth Area Curve	Clicking the **Ellipsis** (…) button in this field opens the Surface Depth-Area Curve editor, allowing you to define the surface depth area curve for each element in the alternative that uses the Surface Depth-Area Curve Surface Storage Type.
Area (Constant Surface)	Lets you define the area in which ponding occurs at the currently selected element. It is available only when the Surface Storage Type attribute is set to Ponded Area.
Surface Storage Type	Contains a drop-down list box that allow you to specify whether the manhole uses the Default Storage Equation or a Surface Depth-Area Curve.
Elevation (Rim)	—Lets you define the top elevation of a catch basin structure. This elevation is typically flush with the ground surface. In some cases, the rim elevation may be slightly below the ground surface elevation (sunk) or slightly above the ground surface elevation (raised).
Set Rim to Ground Elevation	Enables or disables a data entry shortcut. If the box is checked, the catch basin rim elevation is set equal to the ground elevation automatically.

Physical Alternative for Outfalls

The physical alternative editor for outfalls is used to create various data sets for the physical characteristics of outfalls. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.

Column	Description
Label	Displays the label for each element in the alternative.
Elevation (Ground)	Displays the ground elevation for each node in the alternative.

Physical Alternative for Pond Outlet Structures

The physical alternative editor for pond outlet structures is used to create various data sets for the physical characteristics of pond outlet structures. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Control Structure	Lets you specify the previously defined control structure associated with the outlet structure
Has Control Structure	Specifies whether or not the pond has an outlet control structure.
Upstream Pond	Displays the label of the upstream pond.

Physical Alternative for Cross Section Nodes

The physical alternative editor for cross section nodes is used to create various data sets for the physical characteristics of cross section nodes. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Manning's n	Lets you define the Manning's roughness value for all of the cross sections in the alternative. This column is available only when the Single Manning's n Roughness Type is specified.
Channel Manning's n	Lets you specify the Manning's roughness value for each cross section node in the alternative. This column is only available for cross section nodes that have a Trapezoidal Channel or Overbank Channel Section Type.
Right Overbank Manning's n	Lets you specify the Manning's roughness value for the right overbank of each cross section node in the alternative. This column is only available for cross section nodes that have a Trapezoidal Channel or Irregular Channel Section Type.
Left Overbank Manning's n	Lets you specify the Manning's roughness value for the left overbank of each cross section node in the alternative. This column is only available for cross section nodes that have a Trapezoidal Channel or Irregular Channel Section Type.
Roughness Type	Lets you specify the roughness type of each cross section in the alternative.
Transition Length	Lets you define the transition length when the Abrupt Transition Type is chosen.
Transition Type	Lets you specify the transition type of the currently highlighted element. If Abrupt is chosen here, the Transition Length field becomes available for input.
Elevations Modifier	The Elevations modifier is a constant value that will be added to each elevation value. This attribute is only used during SWMM calculations.
Stations Modifier	The Stations modifier is a factor by which the distance between each station will be multiplied when the transect data is processed by SWMM. Use a value of 0 if no such factor is needed. This attribute is only used during SWMM calculations.

Column	Description
Right Bank Station	The distance values appearing in the Station/ Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Left Bank Station	The distance values appearing in the Station/ Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Material	Lets you enter the name of the material used. Alternatively, clicking the **Ellipsis** (…) button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Manning's n Flow Curve	Clicking the **Ellipsis** (…) button in this field opens the Manning's n Flow Curve editor, allowing you to define the Manning's roughness vs. flow curve for each element in the alternative that uses the Manning's n Flow Curve Roughness Type.
Manning's n Depth Curve	Clicking the **Ellipsis** (…) button in this field opens the Manning's n Depth Curve editor, allowing you to define the Manning's roughness vs. depth curve for each element in the alternative that uses the Manning's n Flow Curve Roughness Type.
Bottom Width	Displays the bottom width of each element in the alternative. This column is only available for Trapezoidal cross section types.
Elevation (Invert)	Lets you define the invert elevation for all cross section nodes in the alternative. This column is only available for Trapezoidal cross section types.
Slope (Left Side)	Lets you define the left side slope for all cross section nodes in the alternative. This column is only available for Trapezoidal cross section types.
Slope (Right Side)	Lets you define the right side slope for all cross section nodes in the alternative. This column is only available for Trapezoidal cross section types.

Column	Description
Height	Lets you define the height, or channel depth for all cross section nodes in the alternative. This column is only available for Trapezoidal cross section types.
Section Type	Lets you specify the cross section type for all cross section nodes in the alternative. The choices include Trapezoidal and Generic cross section types.

Physical Alternative for Wet Wells

The physical alternative editor for wet wells is used to create various data sets for the physical characteristics of wet wells. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Exponent	Lets you set the exponent of the area function for the currently highlighted element. It is available only when Wet Well Area Function is chosen as the Wet Well Volume Type.
Coefficient	Lets you set the coefficient of the area function for the currently highlighted element. It is available only when Wet Well Area Function is chosen as the Wet Well Volume Type.
Area	Allows you to define the area of each wet well in the alternative.
Wet Well Depth-Area Curve	Clicking the **Ellipsis** (…) button in this field opens the Wet Well Depth-Area Curve editor, allowing you to define the depth-area curve for each wet well in the alternative that uses the Wet Well Depth-Area Curve Wet Well Volume Type.

Column	Description
Wet Well Volume Type	Lets you specify the volume type of each wet well in the alternative. The type chosen here will affect the availability of other fields in the alternative.
Ponded Area	Lets you define the ponded area of the wet well.
Max. Level	Lets you define the maximum water surface elevation of the wet well.
Elevation (Invert)	Lets you define the invert elevation of each wet well in the alternative.

Physical Alternative for Pressure Junctions

The physical alternative editor for pressure junctions is used to create various data sets for the physical characteristics of pressure junctions. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Elevation	Lets you define the elevation of each pressure junction in the alternative.

Physical Alternative for Junction Chambers

The physical alternative editor for junction chambers is used to create various data sets for the physical characteristics of junction chambers. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Diameter	Lets you set the diameter of the currently highlighted element.
Elevation (Bottom)	Lets you set the bottom elevation of the currently highlighted element.
Elevation (Top)	Lets you set the top elevation of the currently highlighted element.
Elevation (Ground)	Lets you set the ground elevation of the currently highlighted element.
Headloss Coefficient	Lets you set the headloss coefficient for the manhole. This field is only available if you selected the **Standard Headloss Method**.
Headloss Coefficient Start	Lets you set the headloss coefficient at the start section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient Stop	Lets you set the headloss coefficient at the stop section. This field is only available if you selected the **Generic Headloss Method**.
Absolute Headloss	Lets you enter a value for the headloss. This field is only available if you selected the **Absolute Headloss Method**.
Headloss Method	Lets you select the headloss method to use: **Absolute**, **HEC-22**, **Generic**, or **Standard**.

Physical Alternative for Conduits

The physical alternative editor for conduits is used to create various data sets for the physical characteristics of conduits. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Egg Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have an egg Section Type.
Number of Barrels	Lets you specify the number of hydraulically identical conduit barrels that make up the conduit. This column is only available for elements whose Conduit Type is Closed Conduit.
Horseshoe Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have a horseshoe Section Type.
Horseshoe Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have a horseshoe Section Type.
Ellipse Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have a an ellipse Section Type.
Ellipse Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have an ellipse Section Type.
Basket Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have a basket-handle Section Type.
Basket Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have a basket-handle Section Type.
Box Equation Form	Lets you specify the form of equation that should be used when applying culvert data. This column is only available for box conduits that do Apply Culvert Data.
Box Kr	Lets you define the reverse flow loss value for the associated conduit. This column is only available for box conduits that do Apply Culvert Data.
Box Ke	Lets you define the entrance loss value for the associated conduit. This column is only available for box conduits that do Apply Culvert Data.

Column	Description
Box Slope Correction Factor	Lets you define the Slope Correction Factor to be used in inlet control calculations. Normally this factor is –0.5, but for mitered inlets, HDS No. 5 suggests +0.7. This column is only available for box conduits that do Apply Culvert Data.
Box Y	Lets you define the Y equation coefficient that is used in the submerged inlet control equation. This column is only available for box conduits that do Apply Culvert Data.
Box M	Lets you define the M equation coefficient that is used in both forms of the unsubmerged inlet control equation. This column is only available for box conduits that do Apply Culvert Data.
Box C	Lets you define the C equation coefficient that is used in the submerged inlet control equation. This column is only available for box conduits that do Apply Culvert Data.
Box K	Lets you define the K equation coefficient that is used in both forms of the unsubmerged inlet control equation. This column is only available for box conduits that do Apply Culvert Data.
Box Inlet Description	Lets you enter an inlet description for the associated conduit. This column is only available for box conduits that do Apply Culvert Data.
Box Apply Culvert Data	Lets you specify whether the associated box conduit does or does not apply culvert data.
Box Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have an box Section Type.
Box Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have an box Section Type.

Column	Description
Circle Equation Form	Contains list boxes that allow you to specify the form of equation that should be used when applying culvert data. This column is only available for circle conduits that do Apply Culvert Data.
Circle Kr	Lets you define the reverse flow loss value for the associated conduit. This column is only available for circle conduits that do Apply Culvert Data.
Circle Ke	Lets you define the entrance loss value for the associated conduit. This column is only available for box conduits that do Apply Culvert Data.
Circle Slope Correction Factor	Lets you define the Slope Correction Factor to be used in inlet control calculations. Normally this factor is –0.5, but for mitered inlets, HDS No. 5 suggests +0.7. This column is only available for circle conduits that do Apply Culvert Data.
Circle Y	Lets you define the Y equation coefficient that is used in the submerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Circle M	Lets you define the M equation coefficient that is used in both forms of the unsubmerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Circle C	Lets you define the C equation coefficient that is used in the submerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Circle K	Lets you define the K equation coefficient that is used in both forms of the unsubmerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Circle Inlet Description	Lets you enter an inlet description for the associated conduit. This column is only available for circle conduits that do Apply Culvert Data.

Column	Description
Circle Apply Culvert Data	Lets you specify whether the associated box conduit does or does not apply culvert data.
Diameter	Lets you define the diameter of the associated conduits. This column is only available for circular conduits.
Slope (Left Side)	Lets you define the left side slope of the associated conduits. This column is only available for conduits that have a Trapezoidal Section Type.
Slope (Right Side)	Lets you define the right side slope of the associated conduits. This column is only available for conduits that have a Trapezoidal Section Type.
Trapezoidal Channel Depth	Lets you define the depth of the associated conduits. This column is only available for conduits that have a Trapezoidal Section Type.
Base Width	Lets you define the base width of the associated conduits. This column is only available for conduits that have a Trapezoidal Section Type.
Section Type	Lets you specify the section type for each conduit in the alternative. The value that is chosen affects the available input data for other columns.
Outlet Structure	Lets you specify the previously defined outlet structure associated with the conduit.
Has Control Structure	Lets you specify whether or not the associated conduit has a control structure.
Length (User Defined)	Lets you define the length of each channel in the alternative that has a user—defined length.
Has User Defined Length	Lets you specify whether the channel has a user-defined or schematic length.
Channel Manning’s n	Lets you specify the Manning’s roughness value for each channel in the alternative. This column is only available for conduits that have a Trapezoidal Channel or Overbank Channel Section Type.

Column	Description
Right Overbank Manning’s n	Lets you specify the Manning’s roughness value for the right overbank of each channel in the alternative. This column is only available for conduits that have a Trapezoidal Channel or Irregular Channel Section Type.
Left Overbank Manning’s n	Lets you specify the Manning’s roughness value for the left overbank of each channel in the alternative. This column is only available for conduits that have a Trapezoidal Channel or Irregular Channel Section Type.
Elevations Modifier	The Elevations modifier is a constant value that will be added to each elevation value. This attribute is only used during SWMM calculations.
Stations Modifier	The Stations modifier is a factor by which the distance between each station will be multiplied when the transect data is processed by SWMM. Use a value of 0 if no such factor is needed. This attribute is only used during SWMM calculations.
Right Bank Station	The distance values appearing in the Station/ Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Left Bank Station	The distance values appearing in the Station/ Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Material	Lets you enter the name of the material used. Alternatively, clicking the **Ellipsis** (…) button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Invert (Stop)	Lets you define the downstream pipe invert.
Set Invert to Stop Node	Lets you automatically set the upstream pipe invert to the elevation of the upstream node.

Column	Description
Invert (Start)	Lets you define the upstream pipe invert.
Set Invert to Start Node	Lets you automatically set the upstream pipe invert to the elevation of the downstream node.
Exit Loss Coefficient	Lets you define the exit loss coefficient for the associated conduits.
Entrance Loss Coefficient	Lets you define the entrance loss coefficient for the associated conduits.
Contraction Loss Coefficient	Lets you define the contraction loss of the conduit. The contraction loss is due to flow transitioning from large-area, low-velocity flow to small-area, high-velocity flow, such as flow exiting a structure and entering a downstream pipe
Expansion Loss Coefficient	Lets you define the expansion loss of the conduit. Expansion losses are encountered when small-area, high-velocity flow meets a large-area, low-velocity flow, such as a pipe discharging into a structure.
Stop Control Structure	Lets you design a stop control structure, or choose a preexisting one. Click the **Ellipsis** (…) button to open the Conduit Control Structure dialog box to set up the control structure you want to use.
Has Stop Control Structure	Lets you define whether or not the currently highlighted element has a stop control structure, and if so, which type. The value chosen here affects the availability of the other fields. If this is set to True, an icon displays at the start/stop—end of the conduit to display the presence of the structure.
Start Control Structure	Lets you design a start control structure, or choose a preexisting one. Click the **Ellipsis** (…) button to open the Conduit Control Structure dialog box to set up the control structure you want to use.

Column	Description
Has Start Control Structure	Lets you define whether or not the currently highlighted element has a start control structure, and if so, which type. The value chosen here affects the availability of the other fields. If this is set to True, an icon displays at the start/stop—end of the conduit to display the presence of the structure.
Flap Gate?	Lets you choose whether or not the highlighted element has a flap gate. If this is set to **True**, and icon displays at the stop-end of the conduit to display the presence of the structure. If this is set to True and you design control structures without flap gates selected, the flap gate check box will be enabled for your control structures and a message displayed.
Manning's n Flow Curve	Lets you define points that describe a roughness-flow curve for the currently highlighted element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Manning's n-Flow Curve dialog box. To use this field, you must set Roughness Type attribute is set to Manning's n-Flow.
Manning's n Depth Curve	Lets you define points that describe a roughness-depth curve for the currently highlighted element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Manning's n-Depth Curve dialog box. To use this field, you must set Roughness Type attribute is set to Manning's n-Depth Curve.
Manning's n	Lets you define the Manning's roughness value for the associated conduits. This column is only available for conduits that use the Single Manning's n Roughness Type.
Roughness Type	Lets you specify the roughness type for each conduit in the alternative.
Arch Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have an arch Section Type.

Column	Description
Arch Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have an arch Section Type.
Catalog Pipe Equation Form	Let you specify the form of equation that should be used when applying culvert data. This column is only available for catalog pipe conduits that do Apply Culvert Data.
Catalog Pipe Kr	Lets you define the reverse flow loss value for the associated conduit. This column is only available for catalog pipe conduits that do Apply Culvert Data.
Catalog Pipe Ke	Lets you define the entrance loss value for the associated conduit. This column is only available for box conduits that do Apply Culvert Data.
Catalog Pipe Slope Correction Factor	Lets you define the Slope Correction Factor to be used in inlet control calculations. Normally this factor is –0.5, but for mitered inlets, HDS No. 5 suggests +0.7. This column is only available for catalog pipe conduits that do Apply Culvert Data.
Catalog Pipe Y	Lets you define the Y equation coefficient that is used in the submerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Catalog Pipe M	Lets you define the M equation coefficient that is used in both forms of the unsubmerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Catalog Pipe C	Lets you define the C equation coefficient that is used in the submerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.
Catalog Pipe K	Lets you define the K equation coefficient that is used in both forms of the unsubmerged inlet control equation. This column is only available for circle conduits that do Apply Culvert Data.

Column	Description
Catalog Pipe Inlet Description	Lets you enter an inlet description for the associated conduit. This column is only available for circle conduits that do Apply Culvert Data.
Catalog Pipe Apply Culvert Data	Contains list boxes that allow you to specify whether the associated box conduit does or does not apply culvert data.
Catalog Pipe Number of Barrels	Lets you specify the number of hydraulically identical conduit barrels that make up the conduit. This column is only available for elements whose Conduit Type is Closed Conduit.
Catalog Pipe	Lets you specify the previously defined catalog pipe associated with the conduit.
Semi-Ellipse Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have a semi-ellipse Section Type.
Semi-Ellipse Rise	Lets you define the rise (height) of the associated conduit. This column is only available for conduits that have a semi-ellipse Section Type.
Egg Span	Lets you define the span (width) of the associated conduit. This column is only available for conduits that have an egg Section Type.
Design Percent Full	Design Capacity lets you enter the percentage full that you would like the link to maintain. If you want the pipe to be 75% full, enter in the 75 in the field. This value in no way affects the network calculations. It is informational only.

Physical Alternative for Channels

The physical alternative editor for channels is used to create various data sets for the physical characteristics of channels. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.

Column	Description
Label	Displays the label for each element in the alternative.
Scaled Length	Displays the scaled (as opposed to user defined) length of each channel in the alternative. This is a calculated field for channels that do not have a user-defined length.
Set Invert to Start Node	Lets you automatically set the upstream channel invert to the elevation of the upstream node.
Invert (Start)	Lets you define the upstream channel invert.
Set Invert to Stop Node	Lets you automatically set the downstream channel invert to the elevation of the downstream node.
Start Control Structure	Lets you design a start control structure, or choose a preexisting one. Click the **Ellipsis** (…) button to open the Channel Control Structure dialog box to set up the control structure you want to use.
Has Stop Control Structure	Lets you define whether or not the currently highlighted element has a stop control structure, and if so, which type. The value chosen here affects the availability of the other fields. If this is set to True, an icon displays at the start/stop-end of the conduit to display the presence of the structure.
Stop Control Structure	Lets you design a stop control structure, or choose a preexisting one. Click the **Ellipsis** (…) button to open the Channel Control Structure dialog box to set up the control structure you want to use
Has Start Control Structure	Lets you design a stop control structure, or choose a preexisting one. Click the **Ellipsis** (…) button to open the Channel Control Structure dialog box to set up the control structure you want to use.

Column	Description
Flap Gate?	Lets you choose whether or not the highlighted element has a flap gate. If this is set to **True**, and icon displays at the stop-end of the conduit to display the presence of the structure. If this is set to True and you design control structures without flap gates selected, the flap gate check box will be enabled for your control structures and a message displayed.
Invert (Stop)	Lets you define the downstream channel invert.
Length (User Defined)	Lets you define the length of each channel in the alternative that has a user-defined length.
Has User Defined Length	Lets you specify whether the channel has a user-defined or schematic length.
Material	Lets you enter the name of the material used. Alternatively, clicking the **Ellipsis** (…) button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.

Physical Alternative for Gutters

The physical alternative editor for gutters is used to create various data sets for the physical characteristics of gutters. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Manning's n (Gutter)	Lets you define the Manning's roughness of each gutter in the alternative.
Gutter Material	Lets you specify the material of each gutter in the alternative.

Column	Description
Length (User Defined)	Lets you define the length of each gutter in the alternative that has a user defined length.
Has User Defined Length	Lets you specify whether the channel has a user-defined or schematic length.
Depth	Lets you define the depth of the trapezoidal gutter. This column is only available for gutters that have a Trapezoid Open Cross Section type.
Slope (Right Side)	Lets you define the right side slope of the trapezoidal gutter. This column is only available for gutters that have a Trapezoid Open Cross Section type.
Slope (Left Side)	Lets you define the left side slope of the trapezoidal gutter. This column is only available for gutters that have a Trapezoid Open Cross Section type.
Bottom Width	Lets you define the width of the bottom of the trapezoidal gutter. This column is only available for gutters that have a Trapezoid Open Cross Section type.
Open Cross Section	Lets you specify whether the cross section is Generic or Trapezoidal for each gutter in the alternative.
Irregular Channel	Clicking the **Ellipsis** (…) button in this field opens the Irregular Channel editor, allowing you to define the shape of each gutter in the alternative that uses the Irregular Channel Open Cross Section type.

Physical Alternative for Ponds

The physical alternative editor for ponds is used to create various data sets for the physical characteristics of ponds. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Infiltration (Average)	Specify the average infiltration rate for the pond. This value only applies when the Pond Infiltration Method is set to Average Infiltration.
Constant Flow	The flow that exits the pond via infiltration during each time step. This value only applies when the Pond Ifiltration Method is set to Constant Flow.
Pond Infiltration Method	Ponds can lose water by infiltration into groundwater. Any water lost by infiltration does not show up in downstream links. You can specify "None" (default) if pond infiltration is not being considered, or you can use one of the following ways to enter pond infiltration rates: 1. a Constant Flow rate given in flow units 2. an Average Infiltration rate in depth per unit time which is multipied by the area of the pond surface at that time step to determine the infiltration rate. In applying either method, the model also has a stability filter when the pond water depth is below 0.5 ft so that the infiltration rate will linearly reduce to zero as the depth decreases to zero.
Invert (Stop)	Lets you define the downstream invert of the pipes. This column is available only when the Volume method is Pipe.
Invert (Start)	Lets you define the upstream invert of the pipes. This column is available only when the Volume method is Pipe.
Volume Type	Specifies which of the four volume methods (Elevation-Area, Elevation-Volume, Pipe, or Functional) is used for each pond in the alternative. Clicking a field in this column allows you to switch between them.

Column	Description
Pond Constant	"C" value in the expression A*Depth^B+C for Depth in feet. This field is available only when the Functional Volume Type is chosen.
Pond Exponent	"B" value in the expression A*Depth^B+C for Depth in feet. This field is available only when the Functional Volume Type is chosen.
Pond Coefficient	"A" value in the expression A*Depth^B+C for Depth in feet. This field is available only when the Functional Volume Type is chosen.
Depth (Maximum)	Lets you define the maximum depth, or water surface elevation, of the pond.
Elevation (Invert)	Lets you define the invert elevation of each pond in the alternative.
Number of Barrels	Lets you define the number of barrels the Pipe Volume uses. This column is available only when the Volume method is Pipe.
Length	Lets you define the length of the pipes. This column is available only when the Volume method is Pipe.
Pipe Diameter	Lets you define the diameter of the pipes. This column is available only when the Volume method is Pipe.
Elevation-Volume Curve	Opens the Elevation-Volume Curve dialog box, allowing you to define the elevation-volume curve for each pond in the alternative that uses the Elevation-Volume Curve Volume Type.
Elevation-Area Curve	Opens the Elevation-Area Curve dialog box, allowing you to define the elevation-area curve for each pond in the alternative.

Physical Alternative for Pressure Pipes

The physical alternative editor for pressure pipes is used to create various data sets for the physical characteristics of pressure pipes. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Length (User Defined)/ Scaled Length	Lets you enter a value for the length of the currently highlighted element. To use this field, you must check the check box in the Has User Defined Length field. If you clear the check box in that field, the Length (User Defined) field displays the scaled length for the currently highlighted element.
Has User Defined Length?	Lets you choose whether the highlighted element uses scaled or user-defined length. If check box in this field is checked, the Length (User Defined) field is activated.
Manning’s n	Lets you enter a value for the Manning's roughness of the currently highlighted element.
Hazen-Williams C	Lets you enter a value for the Hazen-Williams C coefficient of the currently highlighted element.
Diameter	Lets you set the diameter of the currently highlighted element.
Material	Lets you enter the name of the material used. Alternatively, clicking the Ellipsis (...) button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Invert (Stop)	Lets you set the stop invert for the currently highlighted element.
Set Invert to Stop Node?	Lets you choose whether to set the invert to the stop node. When the check box checked, the invert is set to the stop node.
Invert (Start)	Lets you set the stop invert for the currently highlighted element.

Column	Description
Set Invert to Start Node?	Lets you choose whether to set the invert to the start node. When the check box checked, the invert is set to the stop node.
Minor Loss Coefficient	Lets you set the minor loss coefficient for the currently highlighted element.
Minor Loss Collection	Clicking the **Ellipsis** (…) button in this field opens the Minor Loss Collection dialog box, allowing you to define a collection of minor loss elements for each pressure pipe in the alternative.

Boundary Condition Alternatives

The boundary condition alternative allows you to define boundary condition settings for outfall elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Elevation (User Defined Tailwater)	Lets you enter the user defined tailwater value. This column is only available for elements that use the User Defined Tailwater Boundary Condition Type.

Column	Description
Boundary Condition Type	Lets you specify the type of boundary condition to be used at the associated outfall element. The following choices are available: • **Free Outfall**—For a free outfall control, it is assumed that the downstream discharge conditions do not directly affect the hydraulic response of the structure. This is equivalent to assuming that the downstream TW elevation never rises above the controlling structure outfall invert of the structure. • **Time-Elevation Curve**—For this type of boundary condition, a time-elevation curve is specified to establish at the outfall a tailwater condition that varies over time. • **Elevation (User Defined Tailwater)**—This type of boundary condition allows you to directly enter a tailwater value. • **Elevation Flow Curve**—For this type of boundary condition, an elevation-flow table is specified to simulate at the outfall a channel or outlet structure where flow rate varies over time. • **Boundary Pond**—This type of boundary condition allows you to specify a pond element as the outfall. • **Normal**—When this type of boundary condition is chosen, the boundary condition is the normal depth of the upstream link. If there is more than one upstream link, the one with the highest normal depth is used as the boundary condition. • **Tidal**—For this type of boundary condition, a cyclic time-elevation curve is specified to establish at the outfall a tailwater condition that varies over time in a cyclical fashion.
Tidal Gate	Lets you specify whether or not there is a tidal gate at each outfall in the alternative.
Cyclic Time-Elevation Curve	Clicking the **Ellipsis** (…) button in this field opens the Cyclic Time-Elevation Curve editor, allowing you to define the cyclic time vs. elevation curve for each outfall in the alternative that uses the Tidal Boundary Condition Type.

Column	Description
Boundary Element	Contains list boxes that display all of the elements that are available to be used as boundary ponds. This column is available only when the Boundary Pond Boundary Condition is specified.
Time-Elevation Curve	Clicking the **Ellipsis** (…) button in this field opens the Time-Elevation Curve editor, allowing you to define the time vs. elevation curve for each outfall in the alternative that uses the Time-Elevation Curve Condition Type.

Initial Conditions Alternative

Ponds and wet wells have a specific set of initial condition properties that define the corresponding settings at the beginning of a simulation. These conditions are stored in an initial conditions alternative, as listed below:

- “Initial Conditions Alternative for Ponds” on page 9-507
- “Initial Conditions Alternative for Wet Wells” on page 9-508

Initial Conditions Alternative for Ponds

The initial conditions alternative for ponds allows you to define the initial settings for ponds. The following conditions are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Elevation (Initial)	Lets you define the initial water surface elevation for each of the ponds in the alternative.
Initial Elevation Type	Allows you to choose the initial elevation type for each of the ponds in the alternative.

Initial Conditions Alternative for Wet Wells

The initial conditions alternative for wet wells allows you to define the initial settings for ponds. The following conditions are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Elevation (Initial)	Lets you define the initial water surface elevation for each wet well in the alternative that uses the User Defined Initial Elevation Initial Elevation Type.
Initial Elevation Type	Lets you specify whether each wet well in the alternative has a user defined initial elevation or initially uses the invert value as the starting elevation

Hydrology Alternatives

The hydrology alternative allows you to define hydrologic settings for catchments. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Groundwater Flow Coefficient	Value of A1 in the groundwater flow formula.
Surface Elevation	Elevation of ground surface for the subcatchment that sits above the aquifer.

Column	Description
Receiving Node	Lets you choose the node to which flow discharges from the currently highlighted element. To use this feature, click Select in the Outfall Node field. Move the cursor over the drawing pane and click the element you want to select for the outflow node.
Aquifer	Lets you assign a predefined aquifer to each catchment in the alternative, or create a new one by clicking the **Ellipsis** (…) button.
Area	Lets you define the area of the associated catchments. This column is only available for catchments using the Unit Hydrograph Runoff Method.
Initial Abstraction	Lets you define the initial abstraction (Ia) for each of the catchments in the alternative. The initial abstraction is a parameter that accounts for all losses prior to runoff and consists mainly of interception, infiltration, evaporation, and surface depression storage. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Generic Horton Loss Method.
Recovery Constant	Dry weather regeneration rate constant for the Horton curve. This attribute is active only when the Loss Method attribute is set to (Generic) Horton.
Maximum Volume	Lets you define the maximum volume for each of the catchments in the alternative.
Moisture Deficit	Lets you set the value for Moisture Deficit, which is the saturated moisture content minus the original moisture content, for each of the catchments in the alternative that use the Green and Ampt Runoff Loss Method. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Green and Ampt Loss Method.

Column	Description
SCS CN	Allows you to define the SCS Cn value for elements that use the SCS Cn Loss Method. This column is only available for catchments that use the SCS Cn Loss Method
K	Lets you define the decay coefficient for each of the catchments in the alternative. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Generic Horton Loss Method.
fo	Lets you define the initial infiltration rate at the time that infiltration begins for each of the catchments in the alternative. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Generic Horton Loss Method.
fc	Lets you define the steady—state infiltration rate that occurs for a sufficiently large time period for each of the catchments in the alternative. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Generic Horton Loss Method.
CN Area Collection	Clicking the **Ellipsis** (…) button in this field opens the Cn Area Collection editor, allowing you to define the Cn area collection for each catchment in the alternative that uses the SCS Cn Loss Method.
Storage (Impervious Depression)	Lets you define the depth of impervious storage for each of the catchments in the alternative that use the EPA-SWMM Runoff Method.
Slope	Lets you define the slope of the associated catchments. This column is only available for catchments that use the EPA-SWMM Runoff Method.

Column	Description
Ks	Lets you set the saturated hydraulic conductivity (the rate at which water travels through the soil when it is saturated) for each of the catchments in the alternative. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Green and Ampt Loss Method.
Capillary Suction	Lets you define the capillary suction value for the soil type associated with each of the catchments in the alternative that use the Green and Ampt Runoff Loss Method. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Green and Ampt Loss Method.
fLoss	Lets you define the fLoss absorption value for each of the catchments in the alternative. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the fLoss Loss Method.
Loss Method	Lets you specify the loss method used by the associated catchments. Other columns become available/unavailable depending on the value chosen here.
Subarea Routing	Lets you define the type of subarea routing at the currently highlighted element. This attribute is available only when the Runoff Method attribute is set to EPA-SWMM Runoff.
Percent Routed	Lets you set the percent of runoff routed between subareas. This attribute is available only when the Runoff Method attribute is set to EPA-SWMM Runoff.
Percent Impervious Zero Storage	Lets you set the percent of the impervious area with no depression storage. This attribute is available only when the Runoff Method attribute is set to EPA-SWMM Runoff.

Column	Description
Storage (Pervious Depression)	Lets you define the depth of pervious storage for each of the catchments in the alternative that use the EPA-SWMM Runoff Method. This column is only available for catchments that use the EPA-SWMM Runoff Method.
Manning's n (Pervious)	Lets you define the depth of pervious storage for each of the catchments in the alternative that use the EPA-SWMM Runoff Method. This column is only available for catchments that use the EPA-SWMM Runoff Method.
Manning's n (Impervious)	Lets you define the depth of impervious storage for each of the catchments in the alternative that use the EPA-SWMM Runoff Method. This column is only available for catchments that use the EPA-SWMM Runoff Method.
Percent Impervious	Lets you define the percentage of impervious storage for each of the catchments in the alternative that use the EPA-SWMM Runoff Method. This column is only available for catchments that use the EPA-SWMM Runoff Method.
Characteristic Width	Lets you define the characteristic width value for each of the catchments in the alternative that use the EPA-SWMM Runoff Method. This column is only available for catchments that use the EPA-SWMM Runoff Method.
Runoff Method	Lets you specify the runoff method for each catchment in the alternative.
Tc Data Collection	Clicking the **Ellipsis** (…) button in this field opens the Tc Data Collection editor, allowing you to define the Tc data for each catchment in the alternative that uses the SCS Unit Hydrograph Runoff Method.

Column	Description
Tc	Lets you define the time of concentration for each of the catchments in the alternative that use SCS Unit Hydrograph Method. This column is only available for catchments that use the SCS Unit Hydrograph Runoff Method.
Convolution Time Step	Lets you define the convolution time step. This column is only available for catchments that use the Unit Hydrograph Runoff Method and the Generic Unit Hydrograph Method. Convolution Time Step has two uses: • It represents the time step by which the input unit-hydrograph data is subdivided when used in a calculation. • It represents the duration of excess rainfall that generated the unit hydrograph.
Unit Hydrograph Method	Lets you specify the unit hydrograph method to be used with the associated catchment.
Runoff Method	This column contains list boxes that allow you to specify the runoff method that is used by the associated catchments. This column is only available for elements that are set to the Unit Hydrograph Runoff Method.
Unit Hydrograph Data	Clicking the **Ellipsis** (…) button in this field opens the Unit Hydrograph Data dialog box, allowing you to define Time vs. Flow unit hydrographs for each element in the alternative.
Outflow Node	Lets you specify the element to which flow the catchment outfalls
Apply Groundwater	Specify whether or not the associated element applies groundwater.
Fixed Surf. Water Depth	Fixed depth of surface water at receiving node (ft. or m) (set to zero if surface water depth will vary as computed by flow routing).
Surface-GW Interaction Coefficient	Value of A3 in the groundwater flow formula.

Column	Description
Surf. Water Flow Exponent	Value of B2 in the groundwater flow formula.
Surf. Water Flow Coefficient	Value of A2 in the groundwater flow formula.
Groundwater Flow Exponent	Value of B1 in the groundwater flow formula.

Output Alternatives

The output alternative lets you define output options for network elements, as listed below:

- "Output Alternative for Conduits" on page 9-514
- "Output Alternative for Channels" on page 9-515

Output Alternative for Conduits

The output alternative editor for conduits is used to create various data sets for the output options of conduits. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Output Options	Contains list boxes that allow you to specify whether the Sections Results type is Summary Results or Detailed Results for the associated element. When Summary Results is selected, the result attributes are displayed for the start, end, and middle of the conduit. SewerGEMS V8*i* breaks a conduit up into a number of longitudinal sections as detailed in the topic "Section Count" on page 14-723. When Detailed Results is selected, the result attributes are displayed for each of the longitudinal sections of the conduit. Only Summary Results are displayed in the graphs and reports for conduits. To create a report containing the Detailed Results for each calculated section, you must use the report command in the "Sections Results Dialog Box" on page 6-206.

Output Alternative for Channels

The output alternative editor for channels is used to create various data sets for the output options of channels. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Output Options	Contains list boxes that allow you to specify whether the Sections Results type is Summary Results or Detailed Results for the associated element. When Summary Results is selected, the result attributes are displayed for the start, end, and middle of the channel. SewerGEMS V8*i* breaks a channel up into a number of longitudinal sections as detailed in the topic “Section Count” on page 14-723. When Detailed Results is selected, the result attributes are displayed for each of the longitudinal sections of the channel. Only Summary Results are displayed in the graphs and reports for channels. To create a report containing the Detailed Results for each calculated section, you must use the report command in the “Sections Results Dialog Box” on page 6-206.

Inflow Alternatives

The inflow alternative lets you define loading data for elements capable of accepting an inflow, as listed below:

- “Inflow Alternative for Manholes” on page 9-516
- “Inflow Alternative for Catch Basins” on page 9-517
- “Inflow Alternative for Outfalls” on page 9-517
- “Inflow Alternative for Catchments” on page 9-518
- “Inflow Alternative for Ponds” on page 9-518
- “Inflow Alternative for Cross Section Nodes” on page 9-518
- “Inflow Alternative for Wet Wells” on page 9-519
- “Inflow Alternative for Pressure Junctions” on page 9-519

Inflow Alternative for Manholes

The inflow alternative for manholes allows you to define loading data for manholes. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Catch Basins

The inflow alternative for catch basins allows you to define loading data for catch basins. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Outfalls

The inflow alternative for outfalls allows you to define loading data for outfalls. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Catchments

The inflow alternative for catchments allows you to define loading data for catchments. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Ponds

The inflow alternative for ponds allows you to define loading data for ponds. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Cross Section Nodes

The inflow alternative for cross section nodes allows you to define loading data for cross section nodes. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.

Column	Description
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Wet Wells

The inflow alternative for wet well nodes allows you to define loading data for wet well nodes. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Inflow Alternative for Pressure Junctions

The inflow alternative for pressure junctions allows you to define loading data for pressure junctions. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Inflow Collection	Contains an **Ellipsis** (...) button that allows you to access the Inflow Collection dialog box for the associated element.

Rainfall Runoff Alternative

The rainfall runoff alternative allows you to define runoff data for catchment, pond, and wet well elements, as listed below:

- "Rainfall Runoff Alternative for Global Rainfall" on page 9-520
- "Rainfall Runoff Alternative for Catchments" on page 9-520
- "Rainfall Runoff Alternative for Ponds" on page 9-521
- "Rainfall Runoff Alternative for Wet Wells" on page 9-522

Rainfall Runoff Alternative for Global Rainfall

The rainfall runoff alternative for global rainfall displays information about global storm events in your project. The following fields are available:

Field	Description
Global Storm Event	Lists all of the rainfall curves that have been defined for the current project in the Storm Events dialog box, which is accessible by clicking the **Ellipsis (...)** button.
Source	Displays the location of the library file for storm events that are derived from an engineering library entry.
Return Event	Displays the return event that is associated with the storm event. The return event is a value that reflects the average time between similar storm events.
Depth	Displays the rainfall depth of the storm as defined by the currently selected storm event.
Duration	Displays the duration of the storm as defined by the currently selected storm event.

Rainfall Runoff Alternative for Catchments

The rainfall runoff alternative for catchments allows you to define runoff data for catchment elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Runoff Hydrograph	Provides access to the User-Defined Hydrograph dialog box for the associated element through the **Ellipsis** (…) button.
Use Local Rainfall	Contains check boxes that allow you to specify whether or not the associated catchment uses local rainfall.
Local Storm	Contains list boxes that allow you to choose from a list of rainfall curves that have already been set up in the current project. This column is available only when for catchments that Use Local Rainfall.
Duration	Lets you define the duration of the storm for each catchment in the alternative. This column is only available for catchments that Use Local Rainfall.
Depth	Lets you define the depth of rainfall for each catchment in the alternative. This column is available only when for catchments that Use Local Rainfall.
Return Event	Lets you specify the return frequency of the storm for each catchment in the alternative.
Runoff Hydrograph	Clicking the Ellipsis (…) button in this field opens the User Defined Hydrograph editor, allowing you to define a hydrograph for each element in the alternative.

Rainfall Runoff Alternative for Ponds

The rainfall runoff alternative for ponds allows you to define runoff data for pond elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Evaporation Factor	Fraction of evaporation rate realized. This attribute is only used during SWMM calculations.

Rainfall Runoff Alternative for Wet Wells

The rainfall runoff alternative for wet wells allows you to define runoff data for wet well elements. The following columns are available:

Field	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Evaporation Factor	Fraction of evaporation rate realized. This attribute is only used during SWMM calculations.

Water Quality Alternative

The water quality alternative allows you to define runoff data for manhole, catch basin, outfall, catchment, pond, and wet well elements, as listed below:

- “Water Quality Alternative for Manholes” on page 9-523
- “Water Quality Alternative for Catch Basins” on page 9-523
- “Water Quality Alternative for Outfalls” on page 9-524
- “Water Quality Alternative for Catchments” on page 9-524
- “Water Quality Alternative for Ponds” on page 9-525
- “Water Quality Alternative for Wet Wells” on page 9-525

Water Quality Alternative for Manholes

The water quality alternative allows you to define water quality data for manhole elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Treatment	Lets you define SWMM treatment functions that will be applied at the currently highlighted element. This field is only used during SWMM calculations.
Apply Treatment	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.

Water Quality Alternative for Catch Basins

The water quality alternative allows you to define water quality data for catch basin elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Treatment	Lets you define SWMM treatment functions that will be applied at the currently highlighted element. This field is only used during SWMM calculations.
Apply Treatment	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.

Water Quality Alternative for Outfalls

The water quality alternative allows you to define water quality data for outfall elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Treatment	Lets you define SWMM treatment functions that will be applied at the currently highlighted element. This field is only used during SWMM calculations.
Apply Treatment	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.

Water Quality Alternative for Catchments

The water quality alternative allows you to define water quality data for catchment elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Treatment	Lets you define SWMM treatment functions that will be applied at the currently highlighted element. This field is only used during SWMM calculations.
Apply Treatment	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.

Water Quality Alternative for Ponds

The water quality alternative allows you to define water quality data for pond elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Treatment	Lets you define SWMM treatment functions that will be applied at the currently highlighted element. This field is only used during SWMM calculations.
Apply Treatment	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.

Water Quality Alternative for Wet Wells

The water quality alternative allows you to define water quality data for wet well elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Treatment	Lets you define SWMM treatment functions that will be applied at the currently highlighted element. This field is only used during SWMM calculations.
Apply Treatment	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.

Sanitary Loading Alternative

The sanitary loading alternative lets you define sanitary loading data for manholes, catch basins, wet wells, and pressure junctions, as listed below:

- "Sanitary Loading Alternative for Manholes" on page 9-526
- "Sanitary Loading Alternative for Catch Basins" on page 9-526
- "Sanitary Loading Alternative for Wet Wells" on page 9-527
- "Sanitary Loading Alternative for Pressure Junctions" on page 9-527

Sanitary Loading Alternative for Manholes

The sanitary loading alternative for manholes allows you to define sanitary loading data for manhole elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Sanitary Loads	Clicking the **Ellipses (...)** button allows you to enter sanitary loads in the Sanitary (Dry Weather) Flow dialog box for the currently highlighted element.
Known Flow	Lets you set the known flow for the currently highlighted element.

Sanitary Loading Alternative for Catch Basins

The sanitary loading alternative for catch basins allows you to define sanitary loading data for catch basin elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Sanitary Loads	Clicking the **Ellipses (...)** button allows you to enter sanitary loads in the Sanitary (Dry Weather) Flow dialog box for the currently highlighted element.
Known Flow	Lets you set the known flow for the currently highlighted element.

Sanitary Loading Alternative for Wet Wells

The sanitary loading alternative for wet wells allows you to define sanitary loading data for wet well elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Sanitary Loads	Clicking the **Ellipses (...)** button allows you to enter sanitary loads in the Sanitary (Dry Weather) Flow dialog box for the currently highlighted element.
Known Flow	Lets you set the known flow for the currently highlighted element.

Sanitary Loading Alternative for Pressure Junctions

The sanitary loading alternative for pressure junctions allows you to define sanitary loading data for pressure junction elements. The following columns are available:

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.

Column	Description
Sanitary Loads	Clicking the **Ellipses (...)** button allows you to enter sanitary loads in the Sanitary (Dry Weather) Flow dialog box for the currently highlighted element.
Known Flow	Lets you set the known flow for the currently highlighted element.

User Data Extensions Alternative

The User Data Alternative allows you to edit the data defined in the User Data Extension command for each of the network element types. The User Data Alternative editor contains a tab for each type of network element.

Calculation Options

Each scenario is associated with a set of calculation options. Calculation options are stored in a discrete Calculation Options Profile.

For more information on Calculation Options Profiles, see “Creating Calculation Profiles” on page 8-432 and “Calculation Profile Attributes” on page 8-433.

Chapter

Presenting Your Results 10

Using Background Layers

Use background layers to display pictures behind your network. For example, you might want to display a picture of a neighborhood behind your network, so you can relate elements in your network to structures and roads depicted in the picture. You can add, delete, edit and rename background layers in the Background Layers Manager.

You can add multiple pictures to your project for use as background layers, and turn off the ones you don't want to show and turn on those you do. Additionally, you can create groups of pictures in folders, so you can hide or show an entire folder or group of pictures at once.

To add or delete background layers, open the Background Layers manager: click **View > Background Layers** (Ctrl+6).

You can use shapefiles, AutoCAD DXF files, and raster (also called bitmap) pictures as background images for your model. These raster image formats are supported: bmp, jpg, jpeg, jpe, jfif, gif, tif, tiff, png, and sid.

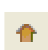
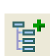
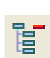
Background Layer Manager

Note: When multiple background layers are overlaid, priority is given to the one that appears highest in the list. In other words, a layer in the first list position is drawn "on top of" all other layers, since they are all below it on the list.

The Background Layer manager lets you add, edit, and remove and manage the background layers that are associated with the project. The dialog box contains a list pane that displays each of the layers currently contained within the project, along with a number of button controls.

When a background layer is added, it appears in the Background Layers list pane, along with an associated check box that is used to control that layer's visibility. Selecting the check box next to a layer causes that layer to become visible in the main drawing pane; clearing it causes it to become invisible. If the layers in the list pane are contained within one or more folders, clearing the check box next to a folder causes all of the layers within that folder to become invisible.

The toolbar consists of the following buttons:

	**New**	Opens a submenu containing the following commands: • **File**—Opens a browse dialog box that allows you to choose the file to use as a background layer. • **Folder**—Creates a folder in the Background Layers list pane.
	**Delete**	Removes the currently highlighted background layer.
	**Rename**	Lets you rename the currently highlighted layer.
	**Edit**	Opens the background layer properties dialog box that corresponds with the currently highlighted background layer.
	**Shift Up**	Moves the currently highlighted object up in the list pane.
	**Shift Down**	Moves the currently highlighted object down in the list pane.
	**Expand All**	Expands all of the branches in the hierarchy displayed in the list pane.
	**Collapse All**	Collapses all of the branches in the hierarchy displayed in the list pane.
	**Help**	Displays online help for the Background Layer Manager.

Working with Background Layer Folders

You can create folders in the Background Layers Manager to organize your background layers and create a group of background layers that can be turned off as one entity. You can also create folders within folders. When you start a new project, Bentley SewerGEMS V8*i* displays an empty folder in the Background Layers Manager called Background Layers. New background layer files and folders are added to the Background Layers folder by default.

To add a background layer folder:

1. Click **View** > **Background Layers** to open the Background Layers Manager.
2. In the Background Layers Manager, click the **New** button, then click **New Folder** from the shortcut menu.

   Or select the default Background Layers folder, then right-click and select **New** > **Folder** from the shortcut menu.

   – If you are creating a new folder within an existing folder, select the folder, then click **New** > **New Folder**. Or right-click, then select **New** > **Folder** from the shortcut menu.
3. Right-click the new folder and select **Rename** from the shortcut menu.
4. Type the name of the folder, then press **Enter**.

To delete a background layer folder:

1. Click **View** > **Background Layers** to open the Background Layers Manager.
2. In the Background Layers Managers, select the folder you want to delete, then click the **Delete** button.

   – You can also right-click a folder to delete, then select **Delete** from the shortcut menu.

To rename a background layer folder:

1. Click **View** > **Background Layers** to open the Background Layers Manager.
2. In the Background Layers Managers, select the folder you want to rename, then click the **Rename** button.

   – You can also right-click a folder to rename, then select **Rename** from the shortcut menu.
3. Type the new name of the folder, then press **Enter**.

   – You can also rename a background layer folder by selecting the folder, then modifying its label in the Properties Editor.

Adding Background Layers

You add background layers to your project using the Background Layers Manager. When you start a new project, Bentley SewerGEMS V8*i* displays an empty folder in the Background Layers Manager called Background Layers. New background layer files and folders are added to the Background Layers folder by default.

To add a background layer:

1. Click **View** > **Background Layers** to open the Background Layers Manager.
2. In the Background Layers Managers, click the **New** button, then click **New File** from the shortcut menu.

   Or right-click on the default Background Layers folder and select **New** > **File** from the shortcut menu.

   – To add a new background layer file to an existing folder in the Background Layer Manager, select the folder, then click **New** > **New File**. Or right-click, then select **New** > **File** from the shortcut menu.
3. Navigate to the file you want to add as a background layer and select it.

   – If you select a .dxf file, the DXF Properties dialog box opens. For more information, see "DXF Properties Dialog Box".

   – If you select a .shp the ShapeFile Properties dialog box opens. For more information, see "Shapefile Properties Dialog Box".

   – If you select a .bmp, .jpg, .jpeg, .jpe, .jfif, .gif, .tif, .tiff, .png, or .sid file, the Image Properties dialog box opens. For more information, see "Image Properties Dialog Box".
4. After you add the background layer, you might have to use the Pan button to move the layer within the drawing area; Zoom Extents does not center a background image.

Deleting Background Layers

To delete a background layer:

Select the background layer you want to delete, then click the **Delete** button.

Or, right-click the background layer, then select **Delete** from the shortcut menu.

Editing Background Layers

You can edit a background layer in two ways: you can edit its properties or its position in a list of background layers displayed in the Background Layers Manager.

To edit the properties of a background layer:

1. Select the background layer you want to edit.
2. Click the **Edit** button. A Properties dialog box opens.
   – You can also right-click the background layer, then select **Edit** from the shortcut menu.

To change the position of a background layer in the list of background layers:

The order of a background layer determines its Z level and what displays if you use more than one background layer. Background layers at the top of the list display on top of the other background layers in the drawing pane; so, background layers that are lower than the top one in the list might be hidden or partially hidden by layers above them in the list.

Select the background layer whose position you want to change in the list of Background Layers Manager, then click the **Shift Up** or **Shift Down** buttons to move the selected background layer up or down in the list.

Renaming Background Layers

To rename a background layer:

Select the background layer you want to rename, then click the **Rename** button.

Or, right-click the background layer that you want to rename, then select **Rename** from the shortcut menu.

Turning Background Layers On and Off

You can choose to turn your background layers off by clearing the check box next to the background layer file or folder than contains it in the Background Layers Manager.

Image Properties Dialog Box

This dialog box opens when you are adding or editing a background-layer image other than a .dxf or .shp. Use the following controls to define the properties of the background layer:

Image Filter	Lets you more clearly display background images that you resize. Set this to **Point**, **Bilinear**, or **Trilinear**. These are methods of displaying your image on-screen. • **Point** works well when you are not changing the size of the image in the display, for example, when you are displaying a 500 x 500 pixel image at 100% using 500 x 500 pixels on-screen. • **Bilinear** and **Trilinear** work well when you display your image on-screen using more or fewer pixels than your image contains, such as displaying a 500 x 500 pixel image at by stretching it to 800 x 800 pixels on-screen. Trilinear gives you smoother transitions when you zoom in and out of the image.
Transparency	Lets you set the transparency level of the background layer. Bentley SewerGEMS V8*i* lets you add transparency to any image type you use as a background. Bentley SewerGEMS V8*i* ignores any transparency that exists in the image before you use it as a background.
Resolution	Lets you select the clarity for MrSID® images that you use as background images. Because using a higher level of clarity or resolution increases the time it takes to display a MrSID image, you can select the resolution that best meets your needs. For formats other than MrSID, this drop-down list contains only one selection.

Use Compression	This checkbox lets you compress the image in memory so that it takes up less RAM while it is being loaded. When you check this option, you may see slight color distortion in the image. **Note: The way the image is compressed depends on your computer's video card. Not all video cards support this feature. If you check this option but your computer's video card does not support image compression, the request for compression will simply be ignored and the image will be loaded uncompressed.**
Image Position Table	Lets you position the background layer with respect to your drawing. • X/Y Image displays the size of the image you are using for a background and sets its position with respect to the origin of your drawing. You cannot change this data. • X/Y Drawing displays where the corners of the image your are using will be positioned relative to your drawing. By default, no scaling is used. However, you can scale the image you are using by setting different locations for the corners of the image you are importing. The locations you set are relative to the origin of your Bentley SewerGEMS V8*i* drawing.

You can also use BMP and JPG image files. For more information, see "How Do I Enter the Scale of a Background Image If it is a File Type without an Inherent Scale?" on page 16-917.

Shapefile Properties Dialog Box

The Shapefile Properties dialog box lets you define a shapefile background layer. Use the following controls to define the properties of the background layer:

Filename	Lists the path and filename of the shapefile to use as a background layer.

Browse	Opens a browse dialog box, letting you select the file to be used as a background layer.
Label	Identifies the background layer.
Unit	Lets you select the unit associated with the spatial data within the shapefile. For example, if the X and Y coordinates of the shapefile represent feet, choose ft. from the drop-down list.
Transparency	Lets you specify the transparency level of the background layer, where 0 has the least transparency and 100 has the most.
Line Color	Sets the color of the layer elements. Click the Ellipsis (...) button to open a Color palette containing more color choices.
Line Thickness	Sets the thickness of the outline of the layer elements. Use values beginning at 0, where 0 is the minimum thickness and larger values are thicker.
Fill Color	Sets the fill color of the layer elements. Click the Ellipsis (...) button to open a Color palette containing more color choices.
Fill Figure	Lets you show or hide the selected fill color for the layer elements. Select this check box to display the selected background color; clear it to turn off the background color and only the outline displays.

To access the Shapefile Properties dialog box, click **New File** in the Background Layers manager, then select an .shp file.

DXF Properties Dialog Box

The DXF Properties dialog box lets you define a .dxf file as the background layer. Use the following controls to define the properties of the background layer:

Filename	Lists the path and filename of the .dxf file to use as a background layer.

Browse	Opens a browse dialog box, letting you select the file to be used as a background layer.
Label	Identifies the background layer.
Unit	Lets you select the unit associated with the spatial data within the shapefile. For example, if the X and Y coordinates of the shapefile represent feet, choose ft. from the drop-down list.
Transparency	Lets you specify the transparency level of the background layer, where 0 has the least transparency and 100 has the most.
Line Color	Sets the color of the layer elements. Click the Ellipsis (...) button to open a Color palette containing more color choices.
Default Color	Lets you use the line color included in the .dxf file or lets you use a custom color that you select in the Line Color field. Select this check box to use the default color included in the .dxf file. cleared this check box if you want to choose a custom color from the Line Color field.
Symbol	Lets you choose the symbol that is displayed for each point element in the .dxf.
Size	Sets the size of the symbol for each point element in the .dxf.

To access the .dxf properties, click **New File** In the Background Layers manager, then select a .dxf file.

Annotating Your Model

You can annotate any of the element types in Bentley SewerGEMS V8*i*.

To work with annotations, open the Element Symbology manager: click **View > Element Symbology** (Ctrl+5).

Element Symbology Manager

The Element Symbology manager allows you to control the way that elements and their associated labels are displayed. The dialog box contains a pane that lists each element type along with the following buttons:

The toolbar consists of the following buttons:

New

Opens a submenu containing the following commands:

- **New Annotation**—Opens the Annotation Properties dialog box, allowing you to define annotation settings for the highlighted element type.
- **New Color Coding**—Opens the Color Coding Properties dialog box, allowing you to define annotation settings for the highlighted element type.
- **Add Folder**—Creates a folder under the currently highlighted element type, allowing you to manage the various color coding and annotation settings that are associated with an element. You can turn off all of the symbology settings contained within a folder by clearing the check box next to the folder. When a folder is deleted, all of the symbology settings contained within it are also deleted.

Delete

Deletes the currently highlighted Color Coding or Annotation Definition or folder.

Rename

Lets you rename the currently highlighted object.

Annotate

Opens a shortcut menu containing the following options:

- **Refresh Annotation**—If you change an annotation's prefix or suffix in the Property Editor, or directly in the database, selecting this command refreshes the annotation.
- **Update Annotation Offset**—If you have adjusted the Initial X or Y offsets, selecting this command resets all annotation Initial X or Y offsets to their default location (or new default location).
- **Update Annotation Height**—If you've adjusted the height multiplier, selecting this command resets all annotation height multiplier to their default values.

	**Shift Up**	Moves the currently highlighted object up in the list pane.
	**Shift Down**	Moves the currently highlighted object down in the list pane.
	**Expand All**	Expands each branch in the tree view pane.
	**Collapse All**	Collapses each branch in the tree view pane.
	**Help**	Displays online help for the Element Symbology Manager.

Using Folders in the Element Symbology Manager

Use folders in the Element Symbology Manager to create a collection of color coding and/or annotation that can be turned off as one entity.

Adding Folders

Use element symbology folders to control whether related annotations and/or color coding displays. To create a folder in the Element Symbology Manager:

1. Click **View > Element Symbology**.
2. In the Element Symbology Manager, right-click an element and select **New** > **Folder**.

   Or, select the element to which you want to add the folder, click the **New** button, then select **New Folder**.
3. Name the folder.
4. You can drag and drop existing annotations and color coding into the folder you create, and you can create annotations and color coding within the folder by right-clicking the folder and selecting **New > Annotation** or **New > Color Coding**.
5. Use the folder to collectively turn on and off the annotations and color coding within the folder.

Note: You can refresh the display of all color-codings/annotations within a folder by right-clicking the folder and selecting the Refresh Group command.

In the MicroStation version, the Refresh Group command will override any local modifications made to color or weight settings applied to individual elements using MicroStation commands. These elements will revert to the SewerGEMS V8*i* symbology settings after a Refresh Group command is initiated.

Deleting Folders

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click the theme folder you want to delete, then select **Delete**.

Or, select the folder you want to delete, then click the **Delete** button.

Renaming Folders

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click the theme folder you want to rename, then select **Rename**.

Or, select the folder you want to rename, then click the **Rename** button.

Adding Annotations

To add an annotation:

1. Click **View** > **Element Symbology**.
2. In the Element Symbology Manager, right-click an element and select **New** > **Annotation**.

   Or, select the element to which you want to add the annotation, click the **New** button, and select **New Annotation**.
3. The Annotation Properties dialog box opens. Select the annotation you want in the Field drop-down list.

   If you don't find the Field you want to use immediately, look carefully through the list of available field selections from top to bottom to make sure you didn't miss the field you want.
4. If needed, set a Prefix or Suffix. Anything you type as a prefix is added directly to the beginning of the label, and anything you type as a suffix is added to the end (so, you may want to include spaces as part of your prefix and suffix).

Note: If you add an annotation that uses units, you can type "%u" in the prefix or suffix field to display the units in the drawing pane.

5. Select the initial X- and Y- offset for the annotation. Offset is measured from the center of the node or polygon or midpoint of the polyline.
6. If needed, set an initial height multiplier. Use a number greater than 1 to make the annotation larger, and a number between 0 and 1 to make the annotation smaller. If you use a negative number, the annotation is flipped (rotated 180 degrees).
7. If you have created selection sets, you can apply your annotation only to a particular selection set by selecting that set from the Selection Set drop-down list. If you have not created any selection sets, then the annotation is applied to all elements of the type you are using.
8. After you finish defining your annotation, click **OK** to close the Annotation Properties dialog box and create your annotation, or **Cancel** to close the dialog box without creating an annotation.

Deleting Annotations

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click an annotation you want to delete, then select **Delete**.

Or, select the annotation you want to delete, then click the **Delete** button.

Editing Annotations

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click the annotation you want to edit, then select **Edit**.

Or, select the annotation you want to edit, then click the **Edit** button.

Note: Changes to annotation settings may not be visible in the drawing pane immediately. To refresh the drawing view to reflect any changes that have been made, you can right-click the annotation that was edited in the Element Symbology Manager and select the Re-Apply Annotation command from the submenu that appears.

Renaming Annotations

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click the annotation you want to rename, then select **Rename**.

Or, select the annotation you want to rename, then click the **Rename** button.

Annotation Properties Dialog Box

The Annotation Properties dialog box allows you to define annotation settings for each element type.

This dialog box allows you to define annotation settings for each element type. The dialog box consists of a list pane on the left and a control section on the right. The control section in the right side of the dialog allow you to edit the settings for the annotation that is currently highlighted in the list pane.

The dialog box consists of the following buttons:

	**New**	Creates a new annotation in the list pane.
	**Delete**	Deletes the annotation that is currently highlighted in the list pane.
	**Rename**	Lets you rename the annotation that is currently highlighted in the list pane.

and the following controls:

List Pane

Label	Displays the name of the current annotation definition.
Initial X Offset	Displays the initial X-axis offset of the annotation in feet.
Initial Y Offset	Displays the initial Y-axis offset of the annotation in feet.

Selected Annotation

Field Name	Lets you specify the attribute that is displayed by the annotation definition.
Prefix	Lets you specify a prefix that is displayed before the attribute value annotation for each element to which the definition applies.

Suffix	Lets you specify a suffix that is displayed after the attribute value annotation for each element to which the definition applies. **Note: If you add an annotation that uses units, you can type "%u" in the prefix or suffix field to display the units in the drawing pane.**
Selection Set	Lets you specify a selection set to which the annotation settings will apply. If the annotation is to be applied to all elements, select the <All Elements> option in this field. <All Elements> is the default setting.
Initial Offset	If checked, the values in the X and Y fields will be applied to all annotations (that are associated with the current annotation definition) in the drawing when the Apply button is clicked.
X	Sets the initial horizontal offset for an annotation. Set this at the time you create the annotation.
Y	Sets the initial vertical offset for an annotation. Set this at the time you create the annotation.
Initial Multiplier	If checked, the value in the Height Multiplier field will be applied to all annotations (that are associated with the current annotation definition) in the drawing when the Apply button is clicked.
Initial Height Multiplier	Sets the initial size of the annotation text. Set this at the time you create the annotation.

Zoom Dependent Visibility

Available through the Properties dialog box of each layer in the Element Symbology manager, this feature can be used to cause elements, decorations, and annotations to only appear in the drawing pane when the view is within the zoom range specified by the Minimum and Maximum Zoom values.

- **Enabled:** Set to true to enable and set to false to disable Zoom Dependent Visibility.

- **Minimum Zoom (%)**: The lowest zoom level at which the element will appear in the drawing pane.
- **Maximum Zoom (%)** : The highest zoom level at which the element will appear in the drawing pane.
- **Apply to Element:** Set to true to apply the zoom minimums and maximums to the symbols in the drawing.
- **Apply to Decorations:** Set to true to apply the zoom minimums and maximums to flow arrows, check valves, and constituent sources in the drawing.
- **Apply to Annotations:** Set to true to apply the zoom minimums and maximums to labels in the drawing.

Color Coding Your Model

Use color coding to help you quickly see what's going on in your Bentley SewerGEMS V8*i* model. Use color coding to change the color and/or size of elements based on the value of data that you select, such as flow or element size.

To work with color coding, open the Element Symbology manager: click **View > Element Symbology** (Ctrl+5).

Adding Color-Coding

To add color coding, including element sizing:

1. Click **View > Element Symbology**.
2. In the Element Symbology Manager, right-click an element and select **New** > **Color Coding**.

   Or, select the element to which you want to add the color coding, click the **New** button, and select **New Color Coding**.
3. The Color Coding Properties dialog box opens. Select the properties for which you want to color code from the Field and Selection Set drop-down lists.
4. In the Options drop-down list, select whether you want to apply color and/or size to the elements you are coding.
   a. Click **Calculate Range**. This automatically sets the maximum and minimum values for your coding. If you want, you can set these values manually.
   b. Click **Initialize**. This automatically creates values and colors in the Color Map. If you want, you can set these values manually.

5. After you finish defining your color coding, click **OK** to close the Color Coding Properties dialog box and create your color coding, or **Cancel** to close the dialog box without creating a color coding.
6. Click **Compute** to compute your network.
7. To see the network color coding and/or sizing change over time:
    a. Click **View > EPS Results Browser**, if needed, to open the EPS Results Browser dialog box.
    b. Click **Play** to use the EPS Results Browser to review your color coding over time.

Deleting Color-Coding

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click the color coding you want to delete, then select **Delete**.

Or, select the color coding you want to delete, then click the **Delete** button.

Editing Color-Coding

Click **View** > **Element Symbology**. In the Element Symbology Manager, right-click the color coding you want to edit, then select **Edit**.

Or, select the color coding you want to edit, then click the **Edit** button.

Note: Changes to color coding settings may not be visible in the drawing pane immediately. To refresh the drawing view to reflect any changes that have been made, you can right-click the annotation that was edited in the Element Symbology Manager and select the Refresh Color Coding command from the submenu that appears.

Renaming Color-Coding

Click **View** > **Element Symbology**. In the Element Symbology manager, right-click the color coding you want to rename, then select **Rename**.

Or, select the color coding you want to rename, then click the **Rename** button.

Color-Coding Properties Dialog Box

This dialog box allows you to define color coding for each element type. The dialog box consists of the following controls:

Properties

Field Name	Lets you select the attribute by which the color coding is applied.
Selection Set	Lets you apply a color coding to a previously defined selection set.
Calculate Range	Automatically finds the minimum and maximum values for the selected attribute and enters them in the appropriate Min. and Max fields.
Min	Lets you define the minimum value of the attribute to be color coded.
Max	Lets you define the maximum value of the attribute to be color coded.
Steps	Lets you specify how many rows are created in the color maps table when you click Initialize. When you click Initialize, a number of values equal to the number of Steps are created in the color maps table. The low and high values are set by the Min and Max values you set.

Color Map

Options	Lets you select whether you want to use color coding, sizing, or both to code and display your elements.

Color Maps Table

Lets you map colors to value ranges for the attribute being color coded. The following buttons are found along the top of the table:

- **New**—Creates a new row in the Color Maps table. 
- **Delete**—Deletes the currently highlighted row from the Color Maps table. 
- **Initialize**—Finds the range of values for the specified attribute, divides it into equal ranges based on the number of Steps you have set, and assigns a color to each range. 
- **Ramp**—Generates a gradient range between two colors that you specify. Pick the color for the first and last values in the list, then Bentley SewerGEMS V8*i* automatically sets intermediate colors for the other values. For example, picking red as the first color and blue as the last color produces varying shades of purple for the other values. 

Above Range Color

Displays the color that is applied to elements whose value for the specified attribute fall outside the range defined in the color maps table. This selection is available if you choose Color or Color and Size from the Options list.

Above Range Size

Displays the size that is applied to elements whose value for the specified attribute fall outside the range defined in the color maps table. This selection is available if you choose Size or Color and Size from the Options list.

Using Profiles

A profile is a graph that plots a particular attribute across a distance, such as ground elevation along a section of piping. As well as these side or sectional views of the ground elevation, profiles can be used to show other characteristics, such as hydraulic grade, pressure, and constituent concentration.

You define profiles by selecting a series of adjacent elements. Only conduits, channels, and gutters can be part of a profile. The profile you create displays the structures you selected, plus the relative ground and water elevations.

To create or use a profile, you must first open the Profiles manager. The Profiles manager is a dockable window that lets you add, delete, rename, edit, and view profiles.

Profiles Manager

The Profiles Manager allows you to create, view, and edit profile views of elements in the network. The dialog box contains a list pane that displays all of the profiles currently contained within the project, along with a toolbar.

The toolbar contains the following buttons:

	**New**	Opens the Profile Setup dialog box, allowing you to create a new profile.
	**Delete**	Deletes the currently highlighted profile.
	**Rename**	Lets you rename the currently highlighted profile.
	**Edit**	Opens the Profile Setup dialog box, allowing you to modify the settings of the currently highlighted profile.
	**View**	Opens the Profile Element Viewer, allowing you to view the currently highlighted profile.
	**Help**	Displays online help for the Profile Manager.

Viewing Profiles

To view a profile:

1. Click **Compute** to calculate flows.
2. Click **View > Profiles** to open the Profile manager.
3. In the Profile manager, select the profile you want to view, then click the **View** button (or double-click the profile you want to view).

Note: You can edit your list of profile elements at any time and compute your network with the Profile Viewer dialog box open, but you must click Refresh to update the display of that dialog box if you do make changes.

4. The Profile Viewer dialog box opens. For more information, see "Profile Viewer Dialog Box" on page 10-555.
5. If necessary, you can click **Chart Settings** to change the look of the profile, and use **Print Preview** and **Print** to print the profile.

Animating Profiles

Animate a profile by:

1. Click **Compute** to calculate flows.
2. Click **View > Profiles** to open the Profiles manager.
3. In the Profiles manager, select the profile you want to see and click the **Profile** button to open the profile in Profile Viewer.
4. If the Scenario Animation dialog box is not open, select **View > Scenario Animation** to open it.
5. If needed, click the Scenario Animation **Option** button to setup the animation.
6. In the Scenario Animation dialog box, move the **Time** slider or click one of the animation buttons and watch the profile change over time in the Profile Viewer. As needed, click the **Pause** button in the Scenario Animation dialog box, to study the profile at a given time.

EPS Results Browser Dialog Box

The EPS Results Browser dialog box allows you to change the currently displayed time step and to animate the main drawing pane. The dialog box contains the following controls:

Time Display	Shows the current time step that is displayed in the drawing pane.
Time Slider	Lets you manually move the slider representing the currently displayed time step along the bar, which represents the full length of time that the scenario encompasses.
Rewind (Full)	Sets the currently displayed time step to the beginning of the simulation.

Play (Backwards)	Sets the currently displayed time step from the end to the beginning.
Rewind (Single Time Step)	Returns the currently displayed time step to the previous time step.
Pause	Stops the animation. Restarts it again with another click.
Forward (Single Time Step)	Advances the currently displayed time step to the following time step.
Play	Advances the currently displayed time step from beginning to end.
Forward (Full)	Sets the currently displayed time step to the end of the simulation.
Speed Slider	Lets you control the length of the delay between time steps during animations.
Options	Opens the Scenario Animation Options dialog box.
Help	Opens Bentley SewerGEMS V8*i* online help.

Animation Options Dialog Box

This dialog box allows you to define the animation settings that are applied when the drawing pane is animated. It contains the following controls:

Frame Options

Increment	Controls the smoothness of the animation. Each time step in a scenario counts as one animation frame. This slider allows you to specify the number of frames that are skipped for each step in the animation. For example, if there are time steps every 3 minutes in the scenario and the slider is set at 3 frames, each step in the animation represents 9 minutes of scenario time when you click the Play button.

Looping Options

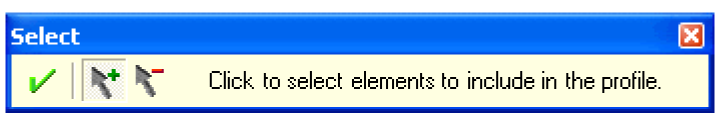
No Loop	Stops the animation at the end of the simulation, if selected.
Loop Animation	Restarts the animation automatically, if selected. When this option is selected, the animation reaches the end of the simulation and then restarts from the beginning.
Rocker Animation	Restarts the animation automatically in reverse. When this option is selected, the animation reaches the end of the simulation and then plays the simulation in reverse. When the beginning of the simulation is reached, the animation advances towards the end again, and so on.

Creating a New Profile

To create a new profile:

1. Click **View** > **Profiles** or click the Profiles Manager button on the View toolbar to open the Profiles manager.
2. Right-click in the Profiles manager and select **New**, or click the **New** button.
3. The Profile Setup dialog box opens. For more information, see "Profile Setup Dialog Box" on page 10-554.
4. Select the Elements you want to use:

    a. Click **Select from Drawing**. The Select dialog box appears:



You must select one path of contiguous elements; you cannot select diverging paths. You can select upstream and downstream elements, but if you begin at an upstream element, select downstream, and then make upstream selections to finish, your profile will be V-shaped, with higher elevations at the beginning and end of the profile than in the middle. Instead, what you might want to do is select elements beginning at a high elevation and selecting elements at increasingly lower elevations towards an outfall.

b. To add elements to the profile, click elements in the drawing pane. (By default, the **Add** button is active in the Select dialog box.) You can only add elements to either end of your selection—all selected elements must be contiguous.

   When there is a plus sign next to the cursor, you can select elements to add to the profile; elements that you successfully select are highlighted red.

c. To remove elements from the profile, click the **Remove** button in the Select dialog box. Thereafter, elements you select in the drawing pane are removed from the profile. You can only remove elements from either end of your selection—all selected elements must be contiguous.

   When there is a minus sign next to the cursor, you can remove elements from the profile; unselected elements are not highlighted.

d. When you are finished adding elements to your profile, click the **Done** button in the Select dialog box.

5. The Profile Setup dialog box opens and displays a list of the elements you selected. If necessary, use the **Reverse** button to reverse the order of these elements, and the **Select from Drawing** or **Remove** buttons to add or remove elements from the list.

   Note: You can edit your list of profile elements at any time and compute your network with the Profile Viewer dialog box open, but you must click Refresh to update the display of that dialog box if you do make changes.

6. Click **Close and Open Profile** to close the Profile Setup dialog box and open the Profile Viewer dialog box.

Editing Profiles

You can edit a profile to change the elements that it uses or the order in which those elements are used. To edit a profile:

1. Click **View > Profiles** to open the Profiles manager.
2. In the Profiles manager, right-click the profile you want to edit, then select **Edit**.

   Or, select the profile you want to edit, then click the **Edit** button.
3. The Profile Setup dialog box opens. Modify the profile as needed and click **OK** to save your changes or **Cancel** to exit without saving your changes.

Deleting Profiles

Click **View** > **Profiles** to open the Profiles manager. In the Profiles manager, right-click the profile you want to delete, then select **Delete**.

Or, select the profile you want to delete, then click the **Delete** button.

Renaming Profiles

Click **View** > **Profiles** to open the Profiles manager. In the Profiles manager, right-click the profile you want to rename, then select **Rename**.

Or, select the profile you want to rename, then click the **Rename** button.

Profile Setup Dialog Box

Setting up a profile is a matter of selecting the adjacent elements on which the profile is based. The Profile Setup dialog box includes the following options:

Label	Displays the list of elements that define the profile.
Select From Drawing	Lets you select and clear elements for the profile. You can select channels, conduits, and ponds for inclusion in your profile. **Note: In AutoCAD mode, you cannot use the shortcut menu, you must re-open the Profile Setup dialog box.**
Reverse	Lets you reverse the profile, so the first node in the list becomes the last, and the last node becomes the first.
Remove All	Removes all elements from the profile.
Remove All Previous	Removes all elements that appear before the selected element in the list. If the selected element is a pipe, the associated node is not removed.
Remove All Following	Removes all elements that appear after the selected element in the list. If the selected element is a pipe, the associated node is not removed.

Open Profile	Closes the Profile Setup dialog box and opens the Profile Viewer dialog box.

After everything is set up to your satisfaction, click **OK** to generate the plot of the profile.

Profile Viewer Dialog Box

This dialog box displays the profile view of the profile run that is plotted from the Profile Manager. It consists of the profile display pane and the following controls:

Zoom Window	Lets you magnify or reduce the display of a section of the graph. To zoom or magnify an area, select the Zoom Window tool, click to the left of the area you want to magnify, then drag the mouse to the right, across the area you want to magnify, so that the area you want to magnify is contained within the marquee that the Zoom Window tool draws. After you have selected the area you want to magnify, release the mouse button to stop dragging. To zoom out, or reduce the magnification, drag the mouse from right to left across the magnified image.
Zoom Extents	Magnifies the profile so that the entire graph is displayed.
Chart Settings	Opens the Chart Options dialog box, letting you view and modify the display settings for the current profile plot. For more information, see “Chart Options Dialog Box” on page 10-587. **Warning! Never delete or rename any of the series entries on the Series Tab of the Chart Options dialog box. These series were specifically designed to enable the display of the Profile Plots.**
Display Labels	Lets you display or hide labels for the elements in your profile plot.

Copy	Copies the contents of the Profile Viewer dialog box as an image to the Windows clipboard, from where you can paste it into another application, such as Microsoft® Word® or Adobe® Photoshop®.
Print	Prints the current view of the profile to your default printer. If you want to use a printer other than your default, use Print Preview to change the printer and print the profile.
Print Preview	Opens a print preview window containing the current view of the profile. You can use the Print Preview dialog box to select a printer and preview the output before you print it. **Note: Do not change the print preview to grayscale, as doing so might hide some elements of the display.**
Refresh	Updates the profile view to reflect changes in input data and results.
EPS Results Browsers	The following EPS Results Browsers are found to the right of the Refresh button: • **Rewind** (Full)—Sets the currently displayed time step to the beginning of the simulation. • **Pause**—Stops the animation. Restarts it again with another click. • **Play**—Advances the currently displayed time step from beginning to end. • **Time Display**—Shows the current time step that is displayed in the drawing pane. • **Time Slider**—Lets you manually move the slider representing the currently displayed time step along the bar, which represents the full length of time that the scenario encom-passes

Viewing and Editing Data in FlexTables

FlexTables lets you view input data and results for all elements of a specific type in a tabular format. You can use the standard set of FlexTables or create customized FlexTables to compare data and create reports.

FlexTables lets you view all elements in the project, all elements of a specific type, or any subset of elements. Additionally, to ease data input and present output data for specific elements, FlexTables can be:

- Filtered (see “Sorting and Filtering FlexTable Data” on page 10-566)
- Globally edited (see “Globally Editing Data” on page 10-565)
- Sorted (see “Sorting and Filtering FlexTable Data” on page 10-566)

If you need to edit a set of properties for all elements of a certain type in your network, you might consider creating a FlexTable and making your changes there, rather than editing each element one at a time, in sequence.

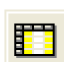
FlexTables can also be used to create results reports that you can print, save as a file, or copy to the Windows clipboard for copying into word processing or spreadsheet software.

To work with FlexTables, select the FlexTables manager or use **View > FlexTables** (Ctrl+7) to open the FlexTables manager if it is closed.

FlexTables Manager

The FlexTables Manager allows you to create, manage, and delete custom tabular reports. The dialog box contains a list pane that displays all of the custom FlexTables currently contained within the project, along with a toolbar.

The toolbar contains the following buttons:

	**New**	Opens a submenu containing the following commands: • **FlexTable**—Creates a new tabular report and opens the FlexTable Setup dialog box, allowing you to define the element type that the FlexTable displays, and the columns that are contained in the table. • **Folder**—Creates a folder in the list pane, allowing you to group custom FlexTables.
	**Delete**	Deletes the currently highlighted FlexTable.
	**Rename**	Lets you rename the currently highlighted FlexTable.
	**Edit**	Opens the FlexTable Setup dialog box, allowing you to make changes to the format of the currently selected table
	**Open**	Lets you open the currently highlighted FlexTable.
	**Help**	Displays online help for the FlexTable Manager.

Working with FlexTable Folders

You can add, delete, and rename folders in the FlexTable Manager to organize your FlexTables into groups of that can be turned off as one entity. You can also create folders within folders. When you start a new project, Bentley SewerGEMS V8*i* displays two items in the FlexTable Manager: Tables - Project (for project-level FlexTables) and Tables - Shared (for FlexTables shared by more than one Bentley SewerGEMS V8*i* project). You can add new FlexTables and FlexTable folders to either item or to existing folders.

To add a FlexTable folder:

1. Click **View > FlexTables** to open the FlexTables Manager.
2. In the FlexTable Manager, select either Tables - Project or Tables - Shared, then click the New button.
   – If you are creating a new folder within an existing folder, select the folder, then click the New button.
3. Click **New Folder** from the shortcut menu.
4. Right-click the new folder and select **Rename** from the shortcut menu.
5. Type the name of the folder, then press **Enter**.

To delete a FlexTable folder:

1. Click **View > FlexTables** to open the FlexTables Manager.
2. In the FlexTables Manager, select the folder you want to delete, then click the Delete button.
   – You can also right-click a folder to delete, then select **Delete** from the shortcut menu.

To rename a FlexTable folder:

1. Click **View > FlexTables** to open the FlexTables Manager.
2. In the FlexTables Manager, select the folder you want to rename, then click the Rename button.
   – You can also right-click a folder to rename, then select **Rename** from the shortcut menu.
3. Type the new name of the folder, then press **Enter**.
   – You can also rename a FlexTable folder by selecting the folder, then modifying its label in the Properties Editor.

FlexTable Dialog Box

FlexTables are displayed in the FlexTable dialog box. The dialog box contains a toolbar, the rows and columns of data in the FlexTable, and a status bar.

The toolbar contains the following buttons:

	**Export to File**	Export to a Shapefile .shp, a Tab Delimited file .txt, or a Comma Delimited File .csv.
	**Copy**	Lets you copy the contents of the selected table cell, rows, and/or columns for the purpose of pasting into a different row or column or into a text editing program such as Notepad.
	**Paste**	Lets you paste the contents of the Windows clipboard into the selected table cell, row, or column. Use this with the Copy button.
	**Edit**	Opens the FlexTable Setup dialog box, allowing you to make changes to the format of the currently selected table

Zoom To

Lets you zoom into and center the drawing pane on the currently selected element in the FlexTable.

Report

Lets you create and view a report of your FlexTable for either the current time step or all time steps.

Options

Opens a submenu containing the following commands:

- **Create Selection Set**—Lets you create a new static selection set (a selection set based on selection) containing the currently selected elements in the FlexTable.
- **Add to Selection Set**—Lets you add the currently selected elements in the FlexTable to an existing selection set.
- **Remove from Selection Set**—Removes the currently selected elements from an existing selection set.
- **Relabel**—Opens an Element Relabeling box where you can Replace, Append, or Renumber

Select In Drawing

Opens a submenu containing the following commands:

- **Select In Drawing**—Selects the currently highlighted element(s) in the drawing pane.
- **Add to Selection**—Adds the currently highlighted element(s) to the group that is currently highlighted in the drawing pane.
- **Remove From Selection**—Removes the currently highlighted element(s) from the group that is currently highlighted in the drawing pane.

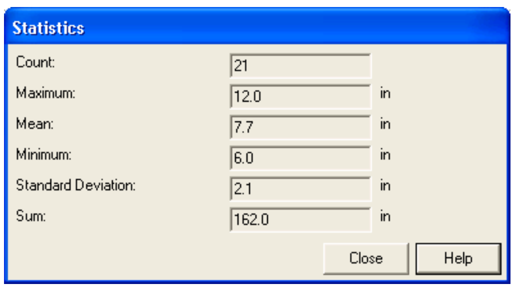
The status bar at the bottom of the FlexTable dialog box contains the following items:

- ***x* of *x* elements displayed**—Number of elements displayed in the FlexTable of the total possible number of that type of element.

- **FILTERED**—If you have applied a filter to the FlexTable, this appears in the status bar. Hold the mouse cursor over this panel to display a tool tip, which lists a summary of active filters.
- **SORTED**—If you have sorted the order of any items in the FlexTable, this appears in the status bar. Hold the mouse cursor over this panel to display a tool tip, which lists a summary of active sorting.

Statistics Dialog Box

The Statistics dialog box displays statistics for the elements in a FlexTable. You can right-click any unitized input or output column and choose the Statistics command to view the count, maximum value, mean value, minimum value, standard deviation, and sum for that column.



Opening FlexTables

You open FlexTables from within the FlexTable Manager.

To open FlexTables:

1. Click **View > FlexTables** or click the FlexTables button on the View toolbar to open the FlexTables Manager.
2. Perform one of the following steps:
   – Right-click the FlexTable you want to open, then select **Open**.
   – Select the FlexTable you want to open, then click the **Open** button.
   – Double-click the FlexTable you want to open.

Creating a New FlexTable

You can create project-level or shared FlexTables.

- Project-level FlexTables are available only for the project in which you create them.
- Shared tables are available in all Bentley SewerGEMS V8*i* projects.

To create a new FlexTable:

Project-level and shared FlexTables are created the same way:

1. Click **View > FlexTables** or click the FlexTables button on the View toolbar to open the FlexTables Manager.
2. In the FlexTables Manager, right-click **Tables - Project** or **Tables - Shared**, then select **New > FlexTable**.

   Or, select **Tables - Project** or **Tables - Shared**, click the **New** button, then select **FlexTable**.
3. The Table Setup dialog box opens.
4. Select the Table Type you want to create. This lets you filter your table by element type.
5. Select the items you want in the FlexTable by moving them to the Selected Columns pane.
6. Click **OK**.
7. The table displays in the FlexTables Manager; you can type to rename the table or accept the default name.

Deleting FlexTables

Click **View > FlexTables** to open the FlexTables Manager. In the FlexTables manager, right-click the FlexTable you want to delete, then select **Delete**.

Or, select the FlexTable you want to delete, then click the **Delete** button. You cannot delete predefined FlexTables.

Note: You cannot delete predefined FlexTables.

Naming and Renaming FlexTables

You name and rename FlexTables in the FlexTable Manager.

To rename FlexTables:

1. Click **View > FlexTables** or click the FlexTables button on the View toolbar to open the FlexTables Manager.
2. Perform one of the following steps:
   – Right-click the FlexTable you want to rename, then select **Rename**.
   – Select the FlexTable you want to rename, then click the **Rename** button.
   – Click the FlexTable you want to rename, to select it, then click the name of the FlexTable.

Note: You cannot rename predefined FlexTables.

Editing FlexTables

You can edit a FlexTable to change the columns of data it contains or the values in some of those columns.

Editable columns:	Columns that contain data you can edit are displayed with a white background. You can change these columns directly in the FlexTable and your changes are applied to your model when you click **OK**. The content in the FlexTable columns can be changed in other areas of Bentley SewerGEMS V8*i*, such as in a Property Editor or managers; but, it might be more efficient to make changes to numerous elements in a FlexTable rather than the Property Editor or a manager. If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.
Non-editable columns:	Columns that contain data you cannot edit are displayed with a yellow background, and correspond to model results calculated by the program and composite values. The content in these columns can be changed in other areas of Bentley SewerGEMS V8*i*, such as in a Property Editor and by running a computation. If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.

To edit a FlexTable:

1. Click **View > FlexTables** to open the FlexTables Manager, then you can:
   – Right-click the FlexTable, then select **Edit**.
   – Double-click the FlexTable to open it, then click **Edit**.
   – Click the FlexTable, to select it, then click the **Edit** button.
2. The Table dialog box opens. .
3. Use the Table dialog box to include and exclude columns and change the order in which the columns appear in the table.
4. Click **OK** after you finish making changes, to save your changes and close the dialog box; or, click **Cancel** to close the dialog box without making changes.

Editing Column-Heading Text

To change the text of a column heading:

1. Click **View > FlexTables** to open the FlexTables Manager.
2. In the FlexTables manager, open the FlexTable you want to edit.
3. Right-click the column heading and select **Edit Column Label**.
4. Type the new name for the label and click **OK** to save those changes and close the dialog box or **Cancel** to exit without making any changes.

Changing Units, Format, and Precision in FlexTables

To change the units, format, or precision in a column of a FlexTable:

1. Click **View > FlexTables** to open the FlexTables Manager.
1. In the FlexTables manager, open the FlexTable you want to edit.
2. Right-click the column heading and select **Units**.
3. Make the changes you want and click **OK** to save those changes or **Cancel** to exit without making any changes.

Navigating in Tables

The arrow keys, Ctrl+Home, Ctrl+End, PgUp, PgDn, and Ctrl+arrow keys navigate to different cells in a table.

Globally Editing Data

Using FlexTables, you can globally edit all of the values in an entire editable column. Globally editing a FlexTable column can be more efficient for editing properties of an element than using the Properties Editor or managers to edit each element in your model individually.

To globally edit the values in a FlexTable column:

1. Click **View** > **FlexTables** to open the FlexTables Manager.
2. In the FlexTables manager, open the FlexTable you want to edit and find the column of data you want to change.

   If necessary, you might need to first create a FlexTable or edit an existing one to make sure it contains the column you want to change.
3. Right-click the column heading and select **Global Edit**.
4. In the **Operation** field, select what you want to do to data in the column: Add, Divide, Multiply, Set, or Subtract.

   Note: The Operation field is only available for numeric data.

5. In the **Global Edit** field, type or select the value you want—for numeric data, you typically type a new value, for other data you might select from a drop-down list or select a check box.

Sorting and Filtering FlexTable Data

You can sort and filter your FlexTables to focus on specific data or present your data in one of the following ways:

To sort the order of columns in a FlexTable:

You can sort the order of columns in a FlexTable in two ways:

- Edit the FlexTable (see “Editing FlexTables”), to open the Table dialog box and change the order of the selected tables using the up and down arrow buttons.

  The top-most item in the Selected Columns pane appears furthest to the left in the resulting FlexTable.
- Open the FlexTable, click the heading of the column you want to move, then click again and drag the column to the new position. You can only move one column at a time.

To sort the contents of a FlexTable:

1. Open the FlexTable you want to edit
2. Right-click a column heading to rank the contents of the column.

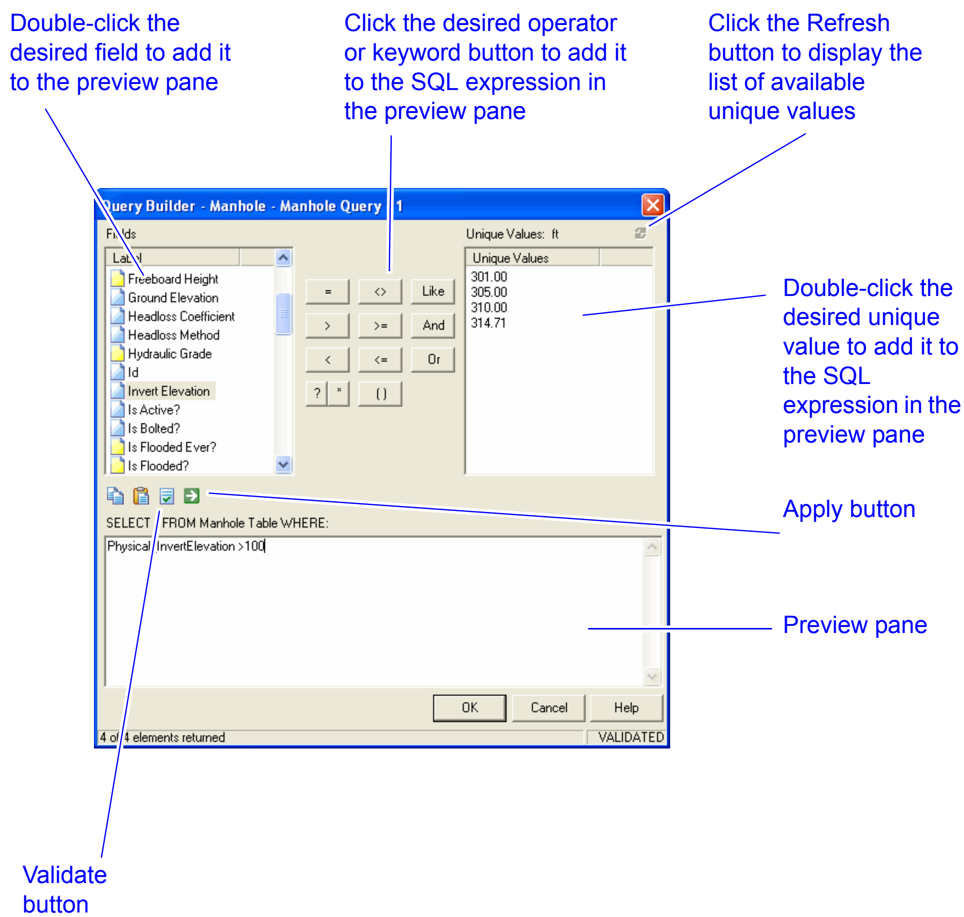
3. Select **Sort Ascending Sort Descending,** or **Custom.**
   - **Sort Ascending**—Sorts alphabetically from A to Z, from top to bottom. Sorts numerically from negative to positive, from top to bottom. Sorts selected check boxes to the top and cleared ones to the bottom.
   - **Sort Descending**—Sorts alphabetically from Z to A, from top to bottom. Sorts numerically from positive to negative, from top to bottom. Sorts cleared check boxes to the top and selected ones to the bottom.
   - **Custom**—Opens the Custom Sort dialog box, which allows you to choose an attribute to sort by in ascending or descending order.

To filter a FlexTable:

You filter a FlexTable by creating a query.

1. Open the FlexTable you want to filter.
2. Right-click the column heading you want to filter, and select **Filter**.

   The Query Builder dialog box opens.
3. All input and results fields for the selected element type appear in the Fields list pane, available SQL operators and keywords are represented by buttons, and available values for the selected field are listed in the Unique Values list pane. Perform the following steps to construct your query:
   a. Double-click the field you wish to include in your query. The database column name of the selected field appears in the preview pane.
   b. Click the desired operator or keyword button. The SQL operator or keyword is added to the SQL expression in the preview pane.
   c. Click the **Refresh** button above the Unique Values list pane to see a list of unique values available for the selected field. The **Refresh** button is becomes disabled after you use it for a particular field.
   d. Double-click the unique value you want to add to the query. The value is added to the SQL expression in the preview pane.
   e. Click the **Validate** button above the preview pane to validate your SQL expression. If the expression is valid, the word “VALIDATED” is displayed in the lower right corner of the dialog box.
   f. Click the **Apply** button above the preview pane to execute the query. If you didn’t validate the expression, the Apply button validates it before executing it.
   g. Click **OK**.



The FlexTable displays columns of data for all elements returned by the query and the word “FILTERED” is displayed in the FlexTable status bar.

To reset a filter:

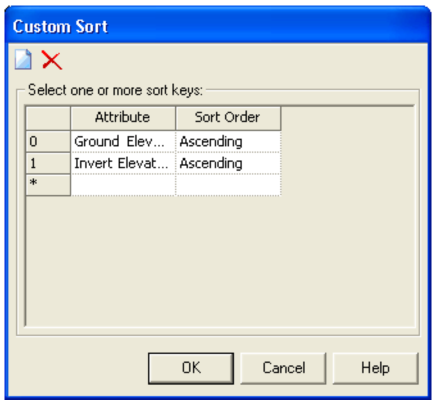
1. Right-click the column heading you want to filter.
2. Select **Filter**.
3. Click **Reset**.

The status pane at the bottom of the Table window always shows the number of rows displayed and the total number of rows available (e.g., 10 of 20 elements displayed). When a filter is active, this message is highlighted.

Note: Table filtering lets you perform global editing (see "Editing FlexTables") on any subset of elements. Only the elements that appear in the filtered table can be edited.

Custom Sort Dialog Box

This dialog box allows you to choose an attribute to sort by, and whether to sort in asending or descending order. You can choose multiple attributes to sort by. When multiple sort attributes are specified, the first attribute specified will take precedence, then the second, and so on.



Customizing Your FlexTable

There are several ways to customize tables to meet a variety of output requirements:

- **Changing the Report Title**—When you print a table, the table name is used as the title for the printed report. You can change the title that appears on your printed report by renaming the table. For more information, see "Naming and Renaming FlexTables" on page 10-563.
- **Adding/Removing Columns**—You can add, remove, and change the order of columns from the Table Setup dialog box. For more information, see "Editing FlexTables" on page 10-564 and "Sorting and Filtering FlexTable Data" on page 10-566.
- **Drag/Drop Column Placement**—With the Table window open, select the column heading of the column that you would like to move and drag the column to its new location. For more information, see "Sorting and Filtering FlexTable Data" on page 10-566.

- **Resizing Columns**—With the Table open, click the vertical separator line between column headings. Notice that the cursor changes shape to indicate that you can resize the column. Drag the column separator to the left or right to stretch the column to its new size.
- **Changing Column Headings**—With the Table window open, right-click the column heading that you wish to change and select **Edit Column Label**.

FlexTable Setup Dialog Box

The Table Setup dialog box allows you to customize any table through the following options:

Table Type	Lets you specify the type of elements that appear in the table. It also provides a filter for the attributes that appear in the Available Columns list. When you choose a table type, the available list only contains attributes that can be used for that table type. For example, only manhole attributes are available for a manhole table.
Available Columns	Contains all the attributes that are available for your table design. The Available Columns list is located on the left side of the Table Setup dialog box. This list contains all of the attributes that are available for the type of table you are creating. The attributes displayed in yellow represent non-editable attributes, while those displayed in white represent editable attributes.
Selected Columns	Contains attributes that appear in your custom designed FlexTable. When you open the table, the selected attributes appear as columns in the table in the same order that they appear in the list. You can drag and drop or use the up and down buttons to change the order of the attributes in the table. The Selected Columns list is located on the right-hand side of the Table Setup dialog box. To add columns to the Selected Columns list, select one or more attributes in the Available Columns list, then click the **Add** button [>] or drag and drop the highlighted attributes to the Selected Columns list.

Column Manipulation Buttons

Lets you select or clear columns to be used in the table, as well as to arrange the order in which the columns appear.
The Add and Remove buttons are located in the center of the Table Setup dialog box.

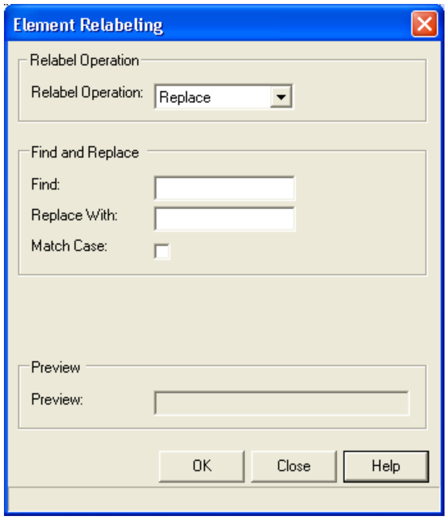
- [>] Adds the selected items from the Available Columns list to the Selected Columns list.
- [>>] Adds all of the items in the Available Columns list to the Selected Columns list.
- [<] Removes the selected items from the Selected Columns list.
- [<<] Removes all items from the Selected Columns list.

Note: You can select multiple attributes in the Available Columns list by holding down the Shift key or the Control key while clicking with the mouse. Holding down the Shift key provides group selection behavior. Holding down the Control key provides single element selection behavior.

To rearrange the order of the attributes in the Selected Columns list, highlight the item to be moved, then move it up or down in the list by dragging or clicking the up or down button located below the Selected Columns list.

Element Relabeling Dialog Box

This dialog is where you perform global element relabeling operations for the Label column of the FlexTable.



The element relabeling tool allows you to perform three types of operations on a set of element labels: Replace, Renumber, and Append. The active relabel operation is chosen from the list box in the Relabel Operations section of the Relabel Elements dialog box. The entry fields for entering the information appropriate for the active relabel operation appear below the Relabel Operations section. The following list presents a description of the available element relabel operations.

- **Replace**—This operation allows you to replace all instances of a character or series of characters in the selected element labels with another piece of text. For instance, if you selected elements with labels CO-1, CO-2, CO-12, and CO-5, you could replace all the COs with the word Conduit by entering CO in the Find field, Conduit in the Replace With field, and clicking the Apply button. The resulting labels are Conduit-1, Conduit-2, Conduit-12, and Conduit-5. You can also use this operation to delete portions of a label. Suppose you now want to go back to the original labels. You can enter Conduit in the Find field and leave the Replace With field blank to reproduce the labels CO-1, CO-2, CO-12, and CO-5. There is also the option to match the case of the characters when searching for the characters to replace. This option can be activated by checking the box next to the Match Case field.
- **Renumber**—This operation allows you to generate a new label, including suffix, prefix, and ID number for each selected element. For example, if you had the labels CO-1, CO-4, CO-10, and Conduit-12, you could use this feature to renumber the elements in increments of five, starting at five, with a minimum number of two digits for the ID number field. You could specify a prefix CO- and a suffix -Z1 in the Prefix and Suffix fields, respectively. The prefix and suffix are appended to the front and back of the automatically generated ID number. The

value of the new ID for the first element to be relabeled, 5, is entered in the Next field. The value by which the numeric base of each consecutive element is in increments, 5, is entered in the Increment field. The minimum number of digits in the ID number, 2, is entered in the Digits field. If the number of digits in the ID number is less then this value, zeros are placed in front of it. Click the Apply button to produce the following labels: CO-05-Z1, CO-10-Z1, CO-15-Z1, and CO-20-Z1.

- **Append**—This operation allows you to append a prefix, suffix, or both to the selected element labels. Suppose that you have selected the labels 5, 10, 15, and 20, and you wish to signify that these elements are actually conduits in Zone 1 of your system. You can use the append operation to add an appropriate prefix and suffix, such as CO- and -Z1, by specifying these values in the Prefix and Suffix fields and clicking the Apply button. Performing this operation yields the labels CO-5-Z1, CO-10-Z1, CO-15-Z1 and CO-20-Z1. You can append only a prefix or suffix by leaving the other entry field empty. However, for the operation to be valid, one of the entry fields must be filled in.

The Preview field displays an example of the new label using the currently defined settings.

Copying, Exporting, and Printing FlexTable Data

You can output your FlexTable several ways:

- Copy FlexTable data via the clipboard
- Export FlexTable data as a text file
- Create a FlexTable report

To copy FlexTable data via the clipboard:

You can copy your FlexTable data via the clipboard and paste it into another Windows application, such as a word-processing application as tab-delimited text.

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to use.
3. Click **Copy**. The contents of the FlexTable are copied to the Windows clipboard.

Caution: Make sure you paste the data you copied before you copy anything else to the Windows clipboard. If you copy something else to the clipboard before you paste your FlexTable data, your FlexTable data will be lost from the clipboard.

4. Paste (Ctrl+v) the data into other Windows software, such as your word-processing application.

To export FlexTable data as a text file:

You can export the data in a FlexTable as tab- or comma-delimited ASCII text, for use in other applications, such as Notepad, spreadsheet, or word processing software.

1. Click **View** > **FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to use.
3. Click **File** > **Export** data.
4. Select either **Tab Delimited** or **Comma Delimited**.
5. When prompted, set the path and name of the .txt file you want to create.

To create a FlexTable report:

Create a FlexTable Report if you want to print a copy of your FlexTable and its values.

1. Click **View** > **FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to use.

   Note: Instead of Print Preview, you can click Print to print the report without previewing it.

3. Click **Report**. A print preview of the report displays to show what your report will look like if printed using your default printer.

   Note: You cannot edit the format of the report.

4. Click **Print** to open the Print dialog box and print the report to a printer that you select.

Using Predefined Tables

Element tables are read-only, predefined FlexTables. There is one predefined table for every element available in Bentley SewerGEMS V8*i*. You can access the element tables by clicking **Report** > **Element Tables** or from the FlexTable manager. Use these tables to review data about the elements in your model.

Reporting

Use reporting to create printable content based on some aspect of your model, such as element properties or results.

You need to compute your model before you can create reports about results, such as the movement of water in your network. But, you can create reports about input data without computing your model, such as conduit diameters. (To compute your model, after you set up your elements and their properties, click the **Compute** button.)

You can access reports by:

- Clicking the **Report** menu
- Right-clicking any element, then selecting **Report**

Reporting includes:

- "Using Standard Reports" on page 10-575
- "Reporting on Element Data" on page 10-576

Using Standard Reports

There are several standard reports available. To access the standard reports, click the **Report** menu, then select the report you want.

You can use these standard reports:

- "Creating a Project Inventory Report" on page 10-575
- "Creating a Scenario Summary Report" on page 10-575

Creating a Project Inventory Report

To create a report that provides an overview of your network, click **Report** > **Project Inventory**. The report dialog box opens and displays your report. You cannot format the report, but you can print it by clicking the **Print** button.

Creating a Scenario Summary Report

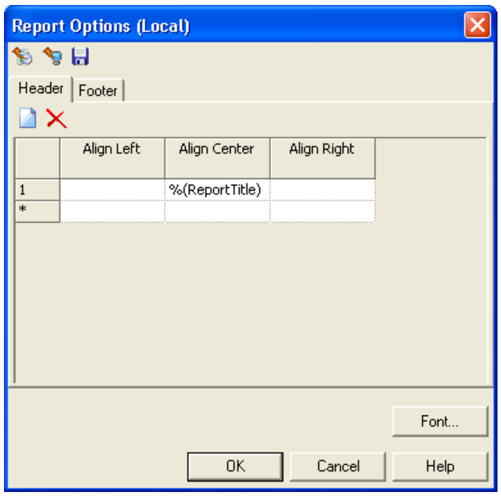
To create a report that summarizes your scenario, click **Report** > **Scenario Summary**. The report dialog box opens and displays your report. You cannot format the report, but you can print it by clicking the **Print** button.

Reporting on Element Data

You can create reports for specific elements in your network by computing the network, right-clicking the element, then selecting **Report**. You cannot format the report, but you can print it by clicking the **Print** button.

Report Options

The Report Options dialog box offers control over how a report is displayed.



	**Load factory default settings to current view**	Changes the display settings used by the current repor to the factory default.
	**Load global settings to current view**	Changes the display settings used by the current report to the previously saved global settings.
	**Save current view settings to global settings**	Saves the display settings used by the current report as the new global settings.

The header and footer can be fully customized and you can edit text to be displayed in the cells or select a pre-defined dynamic variable from the cell's menu.

- %(Company) - The name specified in the project properties.
- % (DateTime) - The current system date and time.
- % (BentleyInfo) - The standard Bentley company information.
- % (BentleyName) - The standard Bentley company name information.

- % (Pagination) - The report page out of the maximum pages.
- % (ProductInfo) - The current product and its build number.
- % (ProjDirectory) - The directory path where the project file is stored.
- % (ProjEngineer) - The engineer specified in the project properties.
- % (ProjFileName) - The full file path of the current project.
- % (ProjStoreFileName) - The full file path of the project.
- % (ProjTitle) - The name of the project specified in the project properties.
- % (ReportTitle) - The name of the report.
- % (Image) - Opens up the Select Image file window.

You can also select fonts, text sizes, and customize spacing.

Graphing

Use graphing to visualize some aspect of your model, such as element properties or results. You need to compute your model before you can create graphs. To compute your model, after you set up your elements and their properties, click the **Compute** button.

Graph Manager

The Graph Manager lets you recall a graph you have created and saved in the current session or in a previous session of Bentley SewerGEMS V8*i*. Graphs listed in the Graph Manager retain any customizations you have applied.

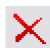
To use the Graph Manager:

1. Compute your model and resolve any errors. (Press F9 or click Analysis > Compute.)
2. Open the Graph Manager, click **View > Graphs**.
3. Create your graph. (For more information, see “Creating a Graph” on page 10-578.)
4. After you create a graph, it is available in the Graph Manager. You can select it by double-clicking it. Also, you can right-click a graph listed in Graph Manager to:
   - Delete it
   - Rename the graph’s label
   - Open it, by selecting **Properties**

Graphs are not saved in Graph Manager after you close Bentley SewerGEMS V8*i*.

Graph Manager

The Graph Manager contains a toolbar with the following buttons:

	**New**	Inserts a new graph of the currently selected elements in your model. If no elements are selected, you are prompted to select one or more elements to graph.
	**Delete**	Deletes the currently highlighted graph.
	**Rename**	Lets you rename the currently highlighted graph.
	**View**	Opens the Graph dialog box, allowing you to view the currently highlighted graph.
	**Help**	Displays online help for the Graph Manager.

Creating a Graph

You can graph computed values, such as flow and velocity. To create a graph:

1. Compute your Bentley SewerGEMS V8*i* network.
2. If necessary, use **Shift+click** to select multiple elements.
3. Right-click an element and select **Graph**. The Graph dialog box opens (see "Graph Dialog Box" on page 10-579).
4. If needed, use the Scenarios drop-down list (for more information, see "Scenario Manager" on page 9-469) to select check boxes to include different or multiple scenarios in the graph. Click **Refresh** after you make any changes, so the graph displays your changes.
5. If needed, use the Elements drop-down list to select check boxes to include different or multiple element properties in the graph. Click **Refresh** after you make any changes, so the graph displays your changes.

Note: Bentley SewerGEMS V8*i* assumes initial flow—flow at time 0—in all networks to be 0; thus, graphs of flow begin at 0 for time 0.

6. If needed, click **Chart Settings** to change the display of the graph. For more information, see "Graph Manager" on page 10-577.

Tip: If you want your graph to display over more time (for example, it displays a 24-hour time period and you want to display a 72-hour period), click View > Calculation Options and change Total Simulation Time in the Property Editor (for more information, see "Editing Attributes in the Property Editor" on page 15-821).

Printing a Graph

To print a graph, click the **Print** button to open the print dialog box or click the **Print Preview** button to see what your graph looks like before clicking **Print**.

Working with Graph Data: Viewing and Copying

Bentley SewerGEMS V8*i* lets you view the data that your graphs are based on. To view your data, create a graph, then, after the Graph dialog box opens, click the **Data** tab.

You can copy this data to the Windows clipboard for use in other applications, such as word-processing software. To copy this data:

1. Click in the top-most cell of the left-most column to select the entire table, click a column heading to select an entire column, or click a row heading to select an entire row.
2. Press **Ctrl+C** to copy the selected data to the clipboard.
3. As needed, press **Ctrl+V** to paste the data as tab-delimited text into other software.

Tip: To print out the data for a graph, copy and paste it into another application, such as word-processing software or Notepad, and print the pasted content.

Graph Dialog Box

The Graph dialog box allows you to view graphs and modify graph settings as desired. After you create a graph, you view it in the Graph dialog box.

The following controls are available:

Graph Tab

	**Add to Graph Manager**	Lets you save the Graph to the graph manager. When you click this button, the graph options (i.e., attributes to graph for a specific scenario) and the graph settings (i.e., line color, font size) are saved with the graph. If you want to view a different set of data (for example, a different scenario), you must change the scenario in the Graph Series Options dialog box. Simply switching the active scenario will not change the graph. Graphs that you add to the Graph manager are saved when you save your model, so that you can use the graph after you close and reopen Bentley SewerGEMS V8*i*.
	**Graph Series Options**	Lets you control what your graph displays. For more information, see "Graph Series Options Dialog Box" on page 10-583.
	**Chart Settings**	Opens the Chart Options dialog box, allowing you to change graph display settings.
	**Print**	Prints the current view in the graph display pane.
	**Print Preview**	Opens the "Print Preview Window", displaying the graph exactly as it will be printed.
	**Copy**	Copies the current view in the graph display pane to the Windows Clipboard.
	**Zoom Extents**	Zooms out so that the entire graph is displayed

	**Zoom Window**	Zooms in on a section of the graph. When the tool is toggled on, you can zoom in on any area of the graph by clicking on the chart to the left of the area to be zoomed, holding the mouse button, then dragging the mouse to the right (or, the opposite extent of the area to be magnified) and releasing the mouse button when the area to be zoomed has been defined. To zoom back out, click and hold the mouse button, drag the mouse in the opposite direction (right to left), and release the mouse button.

Time (VCR) Controls

Lets you evaluate plots over time.

- If you click Restart, the Time resets to zero and the vertical line that marks time resets to the left edge of the Graph display.
- If you click Pause, the vertical line that moves across the graph to mark time pauses, as does the Time field.
- If you click Play, a vertical line moves across the graph and the Time field increments.

The following controls are also available:

- **Time**—Displays the time location of the vertical black bar in the graph display. This is a read-only field, to set a specific time, use the slider button.
- **Slider**—Lets you set a specific time for the graph. A vertical line moves in the graph display and intersects your plots to show the value of the plot at a specific time. Use the slider to set a specific time value.

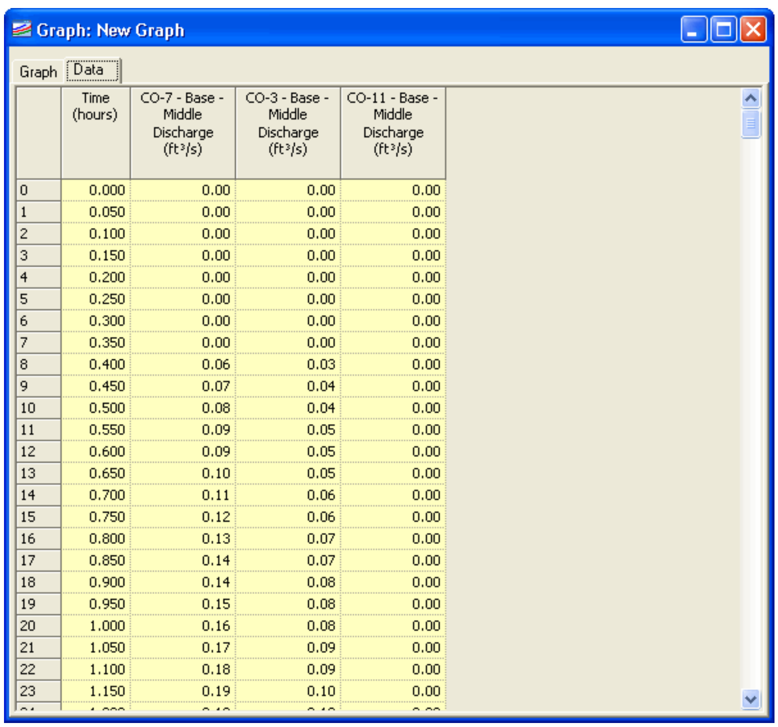
Graph Display Pane

Displays the graph.

Data Tab

Data Table	The Data tab displays the data that comprise your graphs. If there is more than one item plotted, the data for each plot is provided. You can copy and paste the data from this tab to the clipboard for use in other applications, such as Microsoft Excel. To select an entire column or row, click the column or row heading. To select the entire contents of the Data tab, click the heading cell in the top-left corner of the tab. Use Ctrl+C and Ctrl+V to paste your data. The column and row headings are not copied.

The Data tab is shown below.



	Time (hours)	CO-7 - Base - Middle Discharge (ft³/s)	CO-3 - Base - Middle Discharge (ft³/s)	CO-11 - Base - Middle Discharge (ft³/s)
0	0.000	0.00	0.00	0.00
1	0.050	0.00	0.00	0.00
2	0.100	0.00	0.00	0.00
3	0.150	0.00	0.00	0.00
4	0.200	0.00	0.00	0.00
5	0.250	0.00	0.00	0.00
6	0.300	0.00	0.00	0.00
7	0.350	0.00	0.00	0.00
8	0.400	0.06	0.03	0.00
9	0.450	0.07	0.04	0.00
10	0.500	0.08	0.04	0.00
11	0.550	0.09	0.05	0.00
12	0.600	0.09	0.05	0.00
13	0.650	0.10	0.05	0.00
14	0.700	0.11	0.06	0.00
15	0.750	0.12	0.06	0.00
16	0.800	0.13	0.07	0.00
17	0.850	0.14	0.07	0.00
18	0.900	0.14	0.08	0.00
19	0.950	0.15	0.08	0.00
20	1.000	0.16	0.08	0.00
21	1.050	0.17	0.09	0.00
22	1.100	0.18	0.09	0.00
23	1.150	0.19	0.10	0.00

Graph Series Options Dialog Box

Click the **Graph Series Options** button in the Graph dialog box (“Graph Dialog Box” on page 10-579) to use the Graph Series Options dialog box to customize your graph.

This dialog box lets you choose which scenarios, elements, and fields you want to plot (this affects both the Graph and Data tabs). Click to select the check boxes next to those items you want to plot and clear the check boxes for those items you do not want to plot. Click **Close** after you have made your selections and the graph is updated to display the items you chose.

Filter Dialog Box

The Filter dialog box lets you specify your filtering criteria. Each filter criterion is made up of three items:

- **Attribute**—The attribute to filter.
- **Operator**—The operator to use when comparing the filter value against the data in the specific column (operators include: =, >, >=, <, <=, < >).
- **Value**—The comparison value.

Any number of criteria can be added to a filter. Multiple filter criteria are implicitly joined with a logical AND statement. When multiple filter criteria are defined, only rows that meet all of the specified criteria will be displayed. A filter will remain active for the associated table until the filter is reset.

The status pane at the bottom of the Table window always shows the number of rows displayed and the total number of rows available (e.g., 10 of 20 elements displayed). When a filter is active, this message will be highlighted.

Observed Data Dialog Box

Use this feature to display user-supplied time variant data values alongside calculated results in the graph display dialog. Model competency can sometimes be determined by a quick side by side visual comparison of calculated results with those observed and collection in the field.

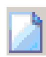
- **Get familiar with your data** - If you obtained your observed data from an outside source, you should take the time to get acquainted with it. Be sure to identify units of time and measurement for the data. Be sure to identify what the data points represent in the model; this helps in naming your line or bar series as it will appear in the graph.
- **Preparing your data** - Typically, observed data can be organized as a collection of points in a table. In this case, the time series data can simply be copied to the clipboard directly from the source and pasted right into the observed data input table. Ensure that your collection of data points is complete. That is, every value must have an associated time value. Oftentimes data points are stored in tab or comma delimited text files; these two import options are available as well. See the "Sample Observed Data Source" topic for an example of the observed data source file format.

- **Specifying the characteristics of your data -** The following charecteristics must be defined:
  - **Time from Start -** An offset of the start time for an EPS scenario.
  - **Y Dimension** - Unit class for the observed data point(s).
  - **Numeric Formatter** - Group of units that correspond to the selected value.
  - **Y Unit** - A preview of the current displayed unit for the selected format.

Note: Go to Tools > Options > Units for a complete list of formats.

Caution: Observed data can only be saved if the graph is saved.

To create Observed Data

1. Click New  .
2. Set hours, dimension, and formatter.
3. Add hours and Y information (or import a .txt or .csv file ).
4. Click Graph  to view the Observed data.
5. Click Close.

Sample Observed Data Source

Below is an example of an Observed Data source for import and graph comparison. The following table contains a flow meter data collection retreived in the field for a given pipe. We will bring this observed data into the model for a quick visual inspection against our model's calculated pipe flows.

Table 10-1: Observed Flow Meter Data (Time in Hours)

Time (hrs)	Flow (gpm)
0.00	125
0.60	120
3.00	110

Table 10-1: Observed Flow Meter Data (Time in Hours)

Time (hrs)	Flow (gpm)
9.00	130
13.75	100
18.20	125
21.85	110

With data tabulated as in the table above, we could simply copy and paste these rows directly into the table in the Observed Data dialog. However if we had too many points to manage, natively exporting our data to a comma delimited text file may be a better import option. Text file import is also a better option when our time values are not formatted in units of time such as hours, as in the table below.

Table 10-2: Observed Flow Meter Data (24-Hr Clock)

Time (24-hr clock)	Flow (gpm)
00:00	125
00:36	120
03:00	110
09:00	130
13:45	100
18:12	125
21:51	110

Below is a sample of what a comma-delimited (*.csv) file would look like:

```
0:00,125

0:36,120

3:00,110

9:00,130
```

```
13:45,100

18:12,125

21:51,110
```

Note: Database formats (such as MS Access) are preferable to simple spreadsheet data sources. The sample described above is intended only to illustrate the importance of using expected data formats.

To import the comma delimited data points:

1. Click the Import toolbar button from the Observed Data dialog.
2. Pick the source .csv file.
3. Choose the Time Format that applies, in this case, HH:mm:ss, and click OK.

Chart Options Dialog Box

Use the Chart Options dialog box to format a graph.

Note: Changes you make to graph settings are not retained for use with other graphs.

To open Chart Options dialog box:

1. Open your project and click **Compute**.
2. Select one or more elements, right-click, then select **Graph**.
3. Click the **Chart Settings** button.

Chart Options Dialog Box - Chart Tab

The Chart tab lets you define overall chart display parameters. This tab is subdivided into second-level sub-tabs:

- “Series Tab”
- “Panel Tab”
- “Axes Tab”
- “General Tab”
- “Titles Tab”

- “Walls Tab”
- “Paging Tab”
- “Legend Tab”
- “3D Tab”

Series Tab

Use the Series tab to display the series that are associated with the current graph. To show a series, select the check box next to the series’ name. To hide a series, clear its check box. The Series tab contains the following controls:

Up/Down arrows	Lets you select the printer you want to use.
Add	Adds a new series to the current graph. The TeeChart Gallery opens, see “TeeChart Gallery Dialog Box”.
Delete	Lets you remove the currently selected series.
Title	Lets you rename the currently selected series.
Clone	Creates a duplicate of the currently selected series.
Change	Lets you edit the currently selected series. The TeeChart Gallery opens, see “TeeChart Gallery Dialog Box”.

Panel Tab

Use the Panel tab to set how your graph appears in the Graph dialog box. The Panel tab includes the following sub-tabs:

Borders Tab

Use the Borders tab to set up a border around your graph. The Borders tab contains the following controls:

Border	Lets you set the border of the graph. The Border Editor opens, see “Border Editor Dialog Box”.
Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the outside of the chart border.

Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the chart border.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Background Tab

Use the Background tab to set a color or image background for your graph. The Background tab contains the following controls:

Color	Lets you set a color for the background of your graph. The Color Editor opens, see "Color Editor Dialog Box".
Pattern	Lets you set a pattern for the background of your graph. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Transparent	Makes the background of the graph transparent.
Background Image	Lets you set an existing image as the background of the graph. Click **Browse**, then select the image (including .bmp, .tif, .jpg, .png,. and .gif). After you have set a background image, you can remove the image from the graph by clicking **Clear**. You can control the Style of the background image: • **Stretch**—Resizes the background image to fill the entire background of the graph. • **Tile**—Repeats the background image as many times as needed to fill the entire background of the graph. • **Center**—Puts the background image in the horizontal and vertical center of the graph. • **Normal**—Puts the background image in the top-left corner of the graph.

Gradient Tab

Use the Gradient tab to create a gradient color background for your graph. The Gradient tab contains the following subtabs and controls:

Format Tab

Visible	Determines whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient. Opens the Color Editor dialog box.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient. Opens the Color Editor dialog box.
End	Lets you select the final color for your gradient. Opens the Color Editor dialog box.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you set the location on the chart background of the gradient's end color.

Sigma Focus	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for your graph. The Shadow tab contains the following controls:

Visible	Lets you display a shadow for your graph. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.
Color	Lets you set a color for the shadow of your graph. You might set this to gray but can set it to any other color.
Pattern	Lets you set a pattern for the shadow of your graph. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Axes Tab

Use the Axes tab set how your axes display. It includes the following controls and subtabs:

Visible	When checked, displays all of your graph's axes; clear it to hide all of the graph's axes.
Behind	When checked, displays all of your graph's axes behind the series display; clear it to display the axes in front of the series display.

Axes	Select the axis you want to edit. The Scales, Labels, Ticks, Title, Minor, and Position tabs and their controls pertain only to the selected axis.

Caution: Do not delete the axes called Custom 0 and Custom 1, as these are reserved axes that are needed by Bentley SewerGEMS V8*i*.

Scales Tab

Use the Scales tab to define your axes scales. The Scales tab contains the following controls:

Automatic	Lets you automatically or manually set the minimum and maximum axis values. Select this check box if you want TeeChart to automatically set both minimum and maximum, or clear this check box if you want to manually set either or both.
Visible	Displays the axis if selected, hides the axis if cleared.
Inverted	Reverses the order in which the axis scale increments. If the minimum value is at the origin, then selecting Inverted puts the maximum value at the origin.
Change	Lets you change the increment of the axis.
Increment	Displays the increment value you set for the axis.
Logarithmic	Lets you use a logarithmic scale for the axis.
Log Base	If you select a logarithmic scale, set the base you want to use in the text box.

Minimum Tab

Auto	Lets you automatically or manually set the minimum axis value.
Change	Lets you enter a value for the axis minimum.

Offset	Lets you adjust the axis scale to change the location of the minimum or maximum axis value with respect to the origin.

Maximum Tab

Auto	Lets you automatically or manually set the maximum axis value.
Change	Lets you enter a value for the axis maximum.
Offset	Lets you adjust the axis scale to change the location of the minimum or maximum axis value with respect to the origin.

Labels Tab

Use the Labels tab to define your axes text. The Labels tab contains the following subtabs and controls:

Style Tab

Visible	Lets you show or hide the axis text.
Multi-line	Lets you split labels or values into more than one line if the text contains a space. Select this check box to enable multi-line text.
Round first	Controls whether axis labels are automatically rounded to the nearest magnitude.
Label on axis	Controls whether Labels just at Axis Minimum and Maximum positions are shown. This applies only if the maximum value for the axis matches the label for extreme value on the chart.
Size	Determines distance between the margin of the graph and the placement of the labels.
Angle	Sets the angle of the axis labels. In addition to using the up and down arrows to set the angle in 90° increments, you can type an angle you want to use.
Min. Separation %	Sets the minimum distance between axis labels.

Style	Lets you set the label style. • **Auto**—Lets TeeChart automatically set the label style. • **Value**—Sets axis labeling based on minimum and maximum axis values. • **Text**—Uses text for labels. Since Bentley SewerGEMS V8*i* uses numeric values, this is not implemented; don't use it. • **None**—Turns off axis labels. • **Mark**—Uses SeriesMarks style for labels. Since Bentley SewerGEMS V8*i* uses numeric values, this is not implemented; don't use it.
Format Tab	
Exponential	Displays the axis label using an exponent, if appropriate.
Values Format	Lets you set the numbering format for the axis labels.
Default Alignment	Lets you select and clear the default TeeChart alignment for the right or left axes only.
Text Tab	
Font	Lets you set the font properties for axis labels. This opens the Windows Font dialog box.
Color	Lets you select the color for the axis label font. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see "Color Editor Dialog Box").
Fill	Lets you set a pattern the axis label font. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".

Shadow	—Lets you set a shadow for the axis labels. • **Visible**—Lets you display a shadow for the axis labels. Select this check box to display the axis label shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Ticks Tab

Use the Ticks tab to define the major ticks and their grid lines. The Ticks tab contains the following controls:

Axis	Lets you set the properties of the selected axis. Opens the Border Editor dialog box.
Grid	Lets you set the properties of the graph's grid lines that intersect the selected axis. Opens the Border Editor dialog box.
Ticks	Lets you set the properties of the tick marks that are next to the labels on the label-side of the selected axis. Opens the Border Editor dialog box.
Len	Sets the length of the Ticks or Inner ticks.
Inner	Lets you set the properties of the tick marks that are next to the labels on the graph-side of the selected axis. Opens the Border Editor dialog box.
Centered	Lets you align between the grid labels the graph's grid lines that intersect the selected axis.
At Labels Only	Sets the axis ticks and axis grid to be drawn at labels only. Otherwise, they are drawn at all axis increment positions.

Title Tab

Use the Title tab to set the axis titles. The Title tab contains the following subtabs and controls:

Style Tab

Title	Lets you type a new axis title.
Angle	Sets the angle of the axis title. In addition to using the up and down arrows to set the angle in 90° increments, you can type an angle you want to use.
Size	Determines distance between the margin of the graph and the placement of the labels.
Visible	Check box that lets you display or hide the axis title.

Text Tab

Font	Lets you set the font properties for axis title. This opens the Windows Font dialog box.
Color	Lets you select the color for the axis title font. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see “Color Editor Dialog Box”).
Fill	Lets you set a pattern the axis title font. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”

Shadow	Lets you set a shadow for the axis title. • **Visible**—Lets you display a shadow for the axis title. Select this check box to display the axis label shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Minor Tab

Use the Minor tab to define those graph ticks that are neither major ticks. The Minor tab contains the following controls and tabs:

Ticks	Lets you set the properties of the minor tick marks. The Border Editor opens, see "Border Editor Dialog Box".
Length	Sets the length of the minor tick marks.
Grid	Lets you set the properties of grid lines that align with the minor ticks. The Border Editor opens, see "Border Editor Dialog Box".
Count	Sets the number of minor tick marks.

Position Tab

Use the Position tab to set the axes position for your graph. The Position tab contains the following controls:

Position %	Sets the position of the axis on the graph in pixels or as a percentage of the graph's dimensions.

Start %	Sets the start of the axis as percentage of width (horizontal axis) and height (vertical axis) of the graph. The original axis scale is fitted to new axis height/width.
End %	Sets the end of the axis as percentage of width (horizontal axis) and height (vertical axis) of the graph. The original axis scale is fitted to new axis height/width.
Units	Lets you select pixels or percentage as the unit for the axis position.
Z %	Sets the Z dimension as a percentage of the graph's dimensions. This is unused by Bentley SewerGEMS V8*i*.

General Tab

Use the General tab to preview a graph before you print it and set up scrolling and zooming for a graph. It includes the following controls:

Print Preview	Lets you see the current view of the document as it will be printed and lets you define the print settings, such as selecting a printer to use. Opens the Print Preview dialog box.
Margins	Lets you specify margins for your graph. There are four boxes, each corresponding with the top, bottom, left, and right margins, into which you enter a value that you want to use for a margin.
Units	Lets you set pixels or percentage as the units for your margins. Percentage is a percentage of the original graph size.
Cursor	Lets you specify what your cursor looks like. Select a cursor type from the drop-down list, then click **Close** to close the TeeChart editor, and the new cursor style displays when the cursor is over the graph.

Zoom Tab

Use the Zoom tab to set up zooming on, magnifying, and reducing the display of a graph. The Zoom tab contains the following controls:

Allow	Lets you magnify the graph by clicking and dragging with the mouse.
Animated	Lets you set a stepped series of zooms.
Steps	Lets you set the number of steps used for successive zooms if you selected the Animated check box.
Pen	Lets you set the thickness of the border for the zoom window that surrounds the magnified area when you click and drag. The Border Editor opens, see "Border Editor Dialog Box".
Pattern	The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Minimum pixels	Lets you set the number of pixels that you have to click and drag before the zoom feature is activated.
Direction	Lets you zoom in the vertical or horizontal planes only, as well as both planes.
Mouse Button	Lets you set the mouse button that you use to click and drag when activating the zoom feature.

Scroll Tab

Use the Scroll tab to set up scrolling and panning across a graph. The Scroll tab contains the following controls:

Allow Scroll	Lets you scroll and pan over the graph. Select this check box to turn on scrolling, clear the check box to turn it off.
Mouse Button	Lets you set the mouse button that you click to use the scroll feature.

Titles Tab

The Titles tab lets you define titles to use for your graph. It includes the following controls and tabs:

Title	Lets you set the location of the titles you want to use. The Titles sub tabs apply to the Title that is currently selected in the Title drop-down list.

Style Tab

Use the Style tab to display and create a selected title. Type the text of the title in the text box on the Style tab. The Style tab contains the following controls:

Visible	Lets you display the selected title.
Adjust Frame	Lets you wrap the frame behind the selected title to the size of the title text. Each title can have a frame behind it (see "Format Tab"). By default, this frame is transparent. If you turn off transparency to see the frame, the frame can be sized to the width of the graph or set to snap to the width of the title text. Select the **Adjust Frame** check box to set the width of the frame to the width of the title text; clear this check box to set the width of the frame to the width of the graph.
Alignment	Lets you set the alignment of the selected title.

Position Tab

Use the Position tab to set the placement of the selected title. The Position tab contains the following controls:

Custom	Lets you set a custom position for the selected title. Select this check box to set a custom position.
Left/Top	Lets you set the location of the selected title relative to the left and top of the graph. If you select the Custom check box, use these settings to position the selected title.

Format Tab

Use the Format tab to set and format a background shape behind the selected title. The Format tab contains the following controls:

Color	Lets you set a color for the fill of the shape you create behind the selected title. The Color Editor opens, see “Color Editor Dialog Box”.
Frame	Lets you define the outline of the shape you create behind the selected title. The Border Editor opens, see “Border Editor Dialog Box”.
Pattern	Lets you set a pattern for the fill of the shape you create behind the selected title. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”.
Round Frame	Lets you round the corners of the rectangular shape you create behind the selected title. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the shape you create behind the selected title as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see.
Transparency	Lets you set transparency for the shape, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Use the Text tab to format the text used in the selected title. The Text tab contains the following controls:

Font	Lets you set the font properties for the text. This opens the Windows Font dialog box.
Color	Lets you select the color for the text. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see “Color Editor Dialog Box”).
Fill	Lets you set a pattern for the text. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”.

Shadow	Lets you set a shadow for the text. • **Visible**—Lets you display a shadow for the text. Select this check box to display the axis label shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Note: To use the Gradient tab, clear the Transparent check box in the Chart > Titles > Format tab.

Use the Gradient tab to create a gradient color background for your axis title. The Gradient tab contains the following controls:

Format Tab

Visible	Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient.

Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for the background for the selected title. The Shadow tab contains the following controls:

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.
Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens, see "Color Editor Dialog Box".

Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Note: To use the Gradient tab, clear the Transparent check box in the Chart > Titles > Format tab.

Use the Bevels tab to create rounded effects for the background for the selected title. The Bevels tab contains the following controls:

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the background for the selected title.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the background for the selected title.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Walls Tab

Use the Walls tab to set and format the edges of your graph. The Walls tab contains the following subtabs:

Left/Right/Back/Bottom Tabs

Use the Left, Right, Back, and Bottom tabs to select the walls that you want to edit. You might have to turn off the axes lines to see the effects (see "Axes Tab" on page 10-591) for the back wall and turn on 3D display to see the effects for the left, right, and bottom walls (see "3D Tab" on page 10-612).

The Left, Right, Back, and Bottom tabs contain the following controls:

Color	The Color Editor opens, see "Color Editor Dialog Box".
Border	The Border Editor opens, see "Border Editor Dialog Box".
Pattern	The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Gradient	Lets you set a color gradient for your walls. The Gradient Editor opens, see "Gradient Editor Dialog Box".
Visible	Lets you display the walls you set up.
Dark 3D	Lets you automatically darken the depth dimension for visual effect. Select a Size 3D larger than 0 to enable this check box.
Size 3D	Lets you increase the size of the wall in the direction perpendicular to it's length (the graph resizes automatically as a result).
Transparent	Lets you set transparency for your background, where 100 is completely transparent and 0 is completely opaque.

Paging Tab

Use the Paging tab to display your graph over several pages. The Paging tab contains the following controls:

Points per Page	Lets you scale the graph to fit on one or many pages. Set the number of points you want to display on a single page of the graph, up to a maximum of 100.
Scale Last Page	Scales the end of the graph to fit the last page.
Current Page Legend	Shows only the current page items when the chart is divided into multiple pages.
Show Page Number	Lets you display the current page number on the graph.

Arrows	Lets you navigate through a multi-page graph. Click the single arrows to navigate one page at a time. Click the double arrows to navigate directly to the last or first pages of the graph.

Legend Tab

Use the Legend tab to display and format a legend for your graph. The Legend tab includes the following controls:

Style Tab

Use the Style tab to set up and display a legend for your graph. The Style tab contains the following controls:

Visible	Lets you show or hide the legend for your graph.
Inverted	Lets you draw legend items in the reverse direction. Legend strings are displayed starting at top for Left and Right Alignment and starting at left for Top and Bottom Legend orientations.
Check boxes	Activates/deactivates check boxes associated with each series in the Legend. When these boxes are unchecked in the legend, the associated series are invisible.
Font Series Color	Sets text in the legend to the same color as the graph element to which it applies.
Legend Style	Lets you select what appears in the legend.
Text Style	Lets you select how the text in the legend is aligned and what data it contains.
Vert. Spacing	Controls the space between rows in the legend.
Dividing Lines	Lets you use and define lines that separate columns in the legend. The Border Editor opens, see “Border Editor Dialog Box”.

Position Tab

Use the Position tab to control the placement of the legend. The Position tab contains the following controls:

Position	Lets you place the legend on the left, top, right, or bottom of the chart.
Resize Chart	Lets you resize your graph to accommodate the legend. If you do not select this check box, the graph and legend might overlap.
Margin	Lets you set the amount of space between the graph and the legend.
Position Offset %	Determines the vertical size of the Legend. Lower values place the Legend higher up in the display
Custom	Lets you use the Left and Top settings to control the placement of the legend.
Left/Top	Lets you enter a value for custom placement of the legend.

Symbols Tab

Use the Symbols tab to add to the legend symbols that represent the series in the graph. The Symbols tab contains the following controls:

Visible	Lets you display the series symbol next to the text in the legend.
Width	Lets you resize the symbol that displays in the legend. You must clear **Squared** to use this control.
Width Units	Lets you set the units that are used to size the width of the symbol.
Default border	Lets you use the default TeeChart format for the symbol. If you clear this check box, you can set a custom border using the Border button.
Border	Lets you set a custom border for the symbols. You must clear Default Border to use this option. The Border Editor opens, see "Border Editor Dialog Box".
Position	Lets you put the symbol to the left or right of its text.

Continuous	Lets you attach or detach legend symbols. If you select this check box, the color rectangles of the different items are attached to each other with no vertical spacing. If you clear this check box, the legend symbols are drawn as separate rectangles.
Squared	Lets you override the width of the symbol, so you can make the symbol square shaped.

Format Tab

Use the Format tab to set and format the box that contains the legend. The Format tab contains the following controls:

Color	Lets you set a color for the fill of the legend's box. The Color Editor opens, see "Color Editor Dialog Box".
Frame	Lets you define the outline of the legend's box. The Border Editor opens, see "Border Editor Dialog Box".
Pattern	Lets you set a pattern for the fill of the legend's box. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Round Frame	Lets you round the corners of the legend's box. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the legend's box as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see.
Transparency	Lets you set transparency for the legend's box, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Use the Text tab to format the text used in the legend. The Text tab contains the following controls:

Font	Lets you set the font properties for the text. This opens the Windows Font dialog box.
Color	Lets you select the color for the text. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see "Color Editor Dialog Box").
Fill	Lets you set a pattern for the text. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Shadow	Lets you set a shadow for the text. • **Visible**—Lets you display a shadow for the text. Select this check box to display the axis label shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Use the Gradient tab to create a gradient color background for your legend. The Gradient tab contains the following controls:

Format Tab

Visible	Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.

Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for the legend. The Shadow tab contains the following controls:

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.
Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens, see “Color Editor Dialog Box”.
Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Use the Bevels tab to create a rounded effects for the legend. The Bevels tab contains the following controls:

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the background for the selected title.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the background for the selected title.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

3D Tab

Use the 3D tab to add a three-dimensional effect to your graph. The 3D tab contains the following controls:

3 Dimensions	Lets you display the chart in three dimensions. Select this check box to turn on three-dimensional display.
3D %	Lets you increase or decrease the three-dimensional effect. Set a larger percentage for more three-dimensional effect, or a smaller percentage for less effect.
Orthogonal	Lets you fix the graph in the two-dimensional work plane or, if you clear this check box, lets you use the Rotation and Elevation controls to rotate the graph freely.
Zoom Text	Lets you magnify and reduce the size of the text in a graph when using the zoom tool. clear this check box if you want text, such as labels, to remain the same size when you use the zoom tool.
Quality	Lets you select how the graph displays as you manipulate and zoom on it.
Clip Points	Trims the view of a series to the walls of your graph's boundaries, to enhance the three-dimensional effect. Turn this on to trim the graph. You only see this effect when the graph is in certain rotated positions.
Zoom	Lets you magnify and reduce the display of the graph in the Graph dialog box.
Rotation	Lets you rotate the graph. You must clear **Orthogonal** to use this control.
Elevation	Lets you rotate the graph. You must clear **Orthogonal** to use this control.
Horiz. Offset	Lets you adjust the left-right position of the graph.
Vert. Offset	Lets you adjust the up-down position of the graph.

Perspective	Lets you rotate the graph. You must clear **Orthogonal** to use this control.

Chart Options Dialog Box - Series Tab

Use the Series tab to set up how the series in your graph display. Select the series you want to edit from the drop-down list at the top of the Series tab.

The Series tab is organized into second-level sub-tabs:

- “Format Tab”
- “Point Tab”
- “General Tab”
- “Data Source Tab”
- “Marks Tab”

Format Tab

Use the Format tab to set up how the selected series appears. The Format tab contains the following controls:

Border	Lets you format the graph of the selected series. The Border Editor opens, see “Border Editor Dialog Box”.
Color	Lets you set a color for the graph of the selected series. The Color Editor opens, see “Color Editor Dialog Box”.
Pattern	Lets you set a pattern for the graph of the selected series. This might only be visible on a three-dimensional graph (see “3D Tab”). The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”.
Dark 3D	Lets you automatically darken the depth dimension for visual effect.
Color Each	Assigns a different color to each series indicator.
Clickable	This is unused by Bentley SewerGEMS V8*i*.

Color Each line	Lets you enable or disable the coloring of connecting lines in a series. This is unused by Bentley SewerGEMS V8*i*.
Height 3D	Lets you set a thickness for the three-dimensional effect in three-dimensional graphs.
Stack	Lets you control how multiple series display in the Graph dialog box. • **None**—Draws the series one behind the other. • **Overlap**—Arranges multiple series with the same origin using the same space on the graph such that they might overlap several times. • **Stack**—Lets you arrange multiple series so that they are additive. • **Stack 100%**—Lets you review the area under the graph curves.
Transparency	Lets you set transparency for your series, where 100 is completely transparent and 0 is completely opaque.
Stairs	Lets you display a step effect between points on your graph.
Inverted	Inverts the direction of the stairs effect
Outline	Displays an outline around the selected series. The Border Editor opens.

Point Tab

Use the Point tab to set up how the points that make up the selected series appear. The Point tab contains the following controls:

Visible	Lets you display the points used to create your graph.
3D	Lets you display the points in three dimensions.
Dark 3D	Lets you automatically darken the depth dimension for visual effect.

Inflate Margins	Adjusts the margins of the points to display points that are close to the edge of the graph. If you clear this option, points near the edge of the graph might only partly display.
Pattern	Lets you set a pattern for the points in your series. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box". You must clear **Default** to use this option.
Default	Lets you select the default format for the points in your series. This overrides any pattern selection.
Color Each	Assigns a different color to each series indicator.
Style	Lets you select the shape used to represent the points in the selected series.
Width/Height	Lets you set a size for the points in the selected series.
Border	Lets you set the outline of the shapes that represent the points in the selected series. The Border Editor opens, see "Border Editor Dialog Box".
Transparency	Lets you set transparency for the points in the selected series, where 100 is completely transparent and 0 is completely opaque.

General Tab

Use the General tab to modify basic formatting and relationships with axes for series in a graph. The General tab contains the following controls:

Show in Legend	Lets you show the series title in the legend. To use this feature, the legend style has to be Series or LastValues (see "Style Tab").
Cursor	Lets you specify what your cursor looks like. Select a cursor type from the drop-down list, then click **Close** to close the TeeChart editor, and the new cursor style displays when the cursor is over the graph.

Depth	Lets you set the depth of the three-dimensional effect (see "3D Tab").
Auto	Lets you automatically size the three-dimensional effect. clear and then select this check box to reset the depth of the three-dimensional effect.
Values	Controls the format of the values displayed when marks are on and they contain actual numeric values
Percents	Controls the format of the values displayed when marks are on and they contain actual numeric values.
Horizontal Axis	Lets you define which axis belongs to a given series, since you can have multiple axes in a chart.
Vertical Axis	Lets you define which axis belongs to a given series, since you can have multiple axes in a chart.
Date Time	This is unused by Bentley SewerGEMS V8*i*.
Sort	Sorts the points in the series using the labels list.

Data Source Tab

Use this tab to connect a TeeChart series to another chart, table, query, dataset, or Delphi database dataset.

This lets you set the number of random points to generate and overrides the points passed by Bentley SewerGEMS V8*i* to the chart control. The Data Source feature can be useful in letting you set its sources as functions and do calculations between the series created by Bentley SewerGEMS V8*i*.

- **Random**—xxxx not sure
- **Number of sample values**—xxxx not sure
- **Default**—xxxx not sure
- **Apply**—xxxx not sure

Marks Tab

Use the Marks tab to display labels for points in the selected series. Series-point labels are called marks. The Marks tab contains the following tabs and controls:

Style Tab

Use the Style tab to set how the marks display. The Style tab contains the following controls:

Visible	Lets you display marks.
Clipped	Lets you display marks outside the graph border. clear this check box to let marks display outside the graph border, or select it to clip the marks to the graph border.
Multi-line	Lets you display marks on more than one line. Select this check box to enable multi-line marks.
All Series Visible	Lets you display marks for all series.
Style	Lets you set the content of the marks.
Draw every	Sets the interval of the marks that are displayed. Selecting 2 would display every second mark, and 3 would display every third, etc.
Angle	Lets you rotate the marks for the selected series.

Arrow Tab

Use the Arrow tab to display a leader line on the series graph to indicate where the mark applies. The Arrow tab contains the following controls:

Border	Lets you set up the leader line. The Border Editor opens, see “Border Editor Dialog Box”.
Pointer	Lets you set up the arrow head (if any) used by the leader line. The Pointer dialog box opens, see “Pointer Dialog Box”.
Arrow head	Lets you select the kind of arrow head you want to add to the leader line.
Size	Lets you set the size of the arrow head.

Length	Lets you set the size of the leader line and arrow head, or just the leader line if there is no arrow head.
Distance	Lets you set the distance between the leader line and the graph of the selected series.

Format Tab

Use the Format tab to set and format the boxes that contains the marks. The Format tab contains the following controls:

Color	Lets you set a color for the fill of the boxes. The Color Editor opens, see “Color Editor Dialog Box”.
Frame	Lets you define the outline of the boxes. The Border Editor opens, see “Border Editor Dialog Box”.
Pattern	Lets you set a pattern for the fill of the boxes. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”.
Round Frame	Lets you round the corners of the boxes. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the boxes as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see.
Transparency	Lets you set transparency for the boxes, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Use the Text tab to format the text used in the marks. The Text tab contains the following controls:

Font	Lets you set the font properties for the text. This opens the Windows Font dialog box.

Color	Lets you select the color for the text. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see "Color Editor Dialog Box").
Fill	Lets you set a pattern for the text. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Shadow	Lets you set a shadow for the text. • **Visible**—Lets you display a shadow for the text. Select this check box to display the axis label shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Use the Gradient tab to create a gradient color background for your marks. The Gradient tab contains the following subtabs and controls:

Format Tab

Visible	Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.

Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for the marks. The Shadow tab contains the following controls:

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.

Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens, see "Color Editor Dialog Box".
Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Use the Bevels tab to create a rounded effects for your marks. The Bevels tab contains the following controls:

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the background for the selected title.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the background for the selected title.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Chart Options Dialog Box - Tools Tab

Use the Tools tab to add special figures in order to highlight particular facts on a given chart. For more information, see "Chart Tools Gallery Dialog Box" on page 10-632. The Tools tab contains the following controls:

Add	Lets you add a tool from the Chart Tools Gallery. To be usable in the current graph, a tool needs to be added and set to Active.

Delete	Deletes the selected tool from the list of those available in the current graph.
Active	Activates a selected tool for the current graph. To be usable in the current graph, a tool needs to be added and set to Active.
Up/Down arrow	These are unused by Bentley SewerGEMS V8*i*.

Note: Each tool has its own parameters, see “Chart Tools Gallery Dialog Box”**.**

Chart Options Dialog Box - Export Tab

Use the Export tab to save your graph for use in another application. The Export tab contains the following controls:

Copy	Lets you copy the contents of the graph to the Windows clipboard, so you can paste it into another application. You must consider the type of data you have copied when choosing where to paste it. For example, if you copy a picture, you cannot paste it into a text editor, you must paste it into a photo editor or a word processor that accepts pictures. Similarly, if you copy data, you cannot paste it into an image editor, you must paste it into a text editor or word processor.
Save	Lets you create a new file from the contents of the graph.

Picture Tab

Use the Picture tab to save your graph as a raster image or to copy the graph as an image to the clipboard. The Picture tab contains the following controls and subtabs:

Format	Lets you select the format of the picture you want to save. GIF, PNG, and JPEG are supported by the Worldwide Web, a metafile is a more easily scalable format. A Bitmap is a Microsoft BMP file that is widely supported on Windows operating systems, whereas TIFF pictures are supported on a variety of Microsoft and non-Microsoft operating systems.

Options Tab

Colors	Lets you use the default colors used by your graph or to convert the picture to use grayscale. This feature is used when you save the picture as a file, not by the copy option.

Size Tab

Width/Height	Lets you change the width and height of the picture. These values are measured in pixels and are used by both the Save and Copy options
Keep aspect ratio	Lets you keep the relationship between the height and width of the picture the same when you change the image size. If you clear this check box, you can distort the picture by setting height or width sizes that are not proportional to the original graph.

Note: Changing the size of a graph using these controls might cause some loss of quality in the image. Instead, try saving the graph as a metafile and resizing the metafile after you paste or insert it into its destination.

Native Tab

The Native tab contains the following controls:

Include Series Data	This is unused by Bentley SewerGEMS V8*i*.
File Size	Displays the size of an ASCII file containing the data from the current graph.

Data Tab

The Data tab contains the following controls:

Series	Lets you select the series from which you copy data.
Format	Lets you select a file type to which you can save the data. This is not used by the Copy function.
Include	Select the data you want to copy.

Text separator	Lets you specify how you want rows of data separated. This is supported by the Save function and only by the Copy function if you first saved using the text separator you have selected, before you copy.

Chart Options Dialog Box - Print Tab

Use the Print tab to preview and print your graph. The Print tab contains the following controls and subtabs:

Printer	Lets you select the printer you want to use.
Setup	Lets you configure the printer you want to use. For example, if the selected printer supports printing on both sides of a page, you might want to turn on this feature.
Print	Prints the displayed graph to the selected printer.

Page Tab

Orientation	Lets you set up the horizontal and vertical axes of the graph. Many graphs print better in Landscape orientation because of their width:height ratio.
Zoom	Lets you magnify the graph as displayed in the print preview window. Use the scrollbars to inspect the graph if it doesn't fit within the preview window after you zoom. Changing the zoom does not affect the size of the printed output.
Margins	Lets you set up top, bottom, left, and right margins that are used when you print.
Margin Units	Lets you set the units used by the Margins controls: percent or hundredths of an inch.

Format Tab

Print Background	When checked, prints the background of the graph.

Quality	You do not need to change this setting. The box is cleared by default.
Proportional	Lets you change the graph from proportional to non-proportional. When you change this setting, the preview pane is automatically updated to reflect the change. This box is checked by default.
Grayscale	Prints the graph in grayscale, converting colors into shades of gray.
Detail Resolution	Lets you adjust the detail resolution of the printout. Move the slider to adjust the resolution.
Preview Pane	Displays a small preview of the graph printout.

Border Editor Dialog Box

The Border Editor dialog box lets you define border properties for your graph. The Border Editor dialog box contains the following controls:

Visible	Displays or hides the border. Select this check box to display the border.
Color	Lets you select a color for the border. The Color Editor dialog box opens, see “Color Editor Dialog Box”.
Ending	Lets you set the ending style of the border.
Dash	Lets you select the dash style, if you have a selection other than Solid set for the border style.
Width	Lets you set the width of the border.
Style	Lets you set the style for the border. Solid is an uninterrupted line.
Transparency	Lets you set transparency for your border, where 100 is completely transparent and 0 is completely opaque.

Gradient Editor Dialog Box

Use the Gradient Editor dialog box to set a blend of two or three colors as the fill. Click **OK** to apply the selection. The Gradient Editor contains the following controls and tabs:

Format Tab

Visible	Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.

Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

To access the Gradient Editor dialog box, click Chart Settings in the Graph dialog box, then click the Tools tab. Select the Axis tab and Color Band tool, then click the Gradient button.

Color Editor Dialog Box

Use the Color Editor dialog box to select a color. Click the basic color you want to use then click **OK** to apply the selection. The Color Editor dialog box contains the following controls:

Transparency	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.
Custom	Lets you define a custom color to use. The Color dialog box opens, see "Color Dialog Box".
OK/Cancel	Click **OK** to use the selection. Click **Cancel** to close the dialog box without making a selection.

To access the Color Editor dialog box, click a Color button in the Chart Options dialog box.

Color Dialog Box

Use the Color dialog box to select a basic color or to define a custom color. After you select the color you want to use, click **OK** to apply the selection.

Basic colors	Lets you click a color to select it.
Custom colors	Displays colors you have created and selected for use.
Color matrix	Lets you use the mouse to select a color from a range of colors displayed.
Color|Solid	Displays the currently defined custom color.

Hue/Sat/Lum	Lets you define a color by entering values for hue, saturation, and luminosity.
Red/Green/Blue	Lets you define a color by entering values of red, green, and blue colors.
Add to Custom Colors	Adds the current custom color to the Custom colors area.

To access the Color dialog box, click the Custom button in the Color Editor dialog box.

Hatch Brush Editor Dialog Box

Use the Hatch Brush Editor dialog box to set a fill. The Hatch Brush Editor dialog box contains the following controls and tabs:

Visible	Displays or hides the pattern. Select this check box to display the selected pattern.

- "Hatch Brush Editor Dialog Box - Solid Tab"
- "Hatch Brush Editor Dialog Box - Hatch Tab"
- "Hatch Brush Editor Dialog Box - Gradient Tab"
- "Hatch Brush Editor Dialog Box - Image Tab"

Hatch Brush Editor Dialog Box - Solid Tab

Use the Solid tab to set a solid color as the fill. The Solid tab contains the following controls:

Transparency	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.
Custom	Lets you define a custom color to use. The Color dialog box opens, see "Color Dialog Box".
OK/Cancel	Click **OK** to use the selection. Click **Cancel** to close the dialog box without making a selection.

Hatch Brush Editor Dialog Box - Hatch Tab

Use the Hatch tab to set a pattern as the fill. Click **OK** to apply the selection. The Hatch tab contains the following controls:

Hatch Style	Select the pattern you want to use. These display using the currently selected background and foreground colors.
Background/ Foreground	Select the color you want to use for the background and foreground of the pattern. This opens the Color Editor, see "Color Editor Dialog Box".
%	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.

Hatch Brush Editor Dialog Box - Gradient Tab

Use the Gradient tab to set a blend of two or three colors as the fill. Click **OK** to apply the selection. The Gradient tab contains the following controls:

Format Tab	
Visible	Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
Angle	Lets you customize the direction of the gradient beyond the Direction selections.
Colors Tab	
Start	Lets you set the starting color for your gradient.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.

End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.
Options Tab	
Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Hatch Brush Editor Dialog Box - Image Tab

Use the Image tab to select an existing graphic file or picture to use as the fill. Click **OK** to apply the selection. The Image tab contains the following controls:

Browse	Lets you navigate to then select the graphic file you want to use. When selected, the graphic displays in the tab.
Style	Lets you define how the graphic is used in the fill. • **Stretch**—Resizes the image to fill the usable space. • **Tile**—Repeats the image to fill the usable space. • **Center**—Puts the image in the horizontal and vertical center. • **Normal**—Puts the image in the top-left corner

Pointer Dialog Box

Use the Pointer dialog box to set up a pointers for use with leader lines. The Pointer dialog box contains the following controls:

Visible	Sets whether a pointer displays or not.
3D	Lets you display the pointer in three dimensions.
Dark 3D	Lets you automatically darken the depth dimension for visual effect.
Inflate Margins	Adjusts the margins of the pointers to display pointers that are close to the edge of the graph. If you clear this option, pointers near the edge of the graph might only partly display.
Pattern	Lets you set a pattern for the pointers. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”. You must clear **Default** to use this option.
Default	Lets you select the default format for the pointers. This overrides any pattern selection.
Color Each	Assigns a different color to each pointer.
Style	Lets you select the shape used to represent the pointers.
Width/Height	Lets you set a size for the pointers.
Border	Lets you set the outline of the shapes that represent the pointers. The Border Editor opens, see “Border Editor Dialog Box”.
Transparency	Lets you set transparency for the pointers, where 100 is completely transparent and 0 is completely opaque.

To access the Pointer dialog box, click Chart Settings in the Graph dialog box, then click Series > Marks > Arrow.

Change Series Title Dialog Box

Use the Change Series Title dialog box to change the title of a selected series. Type the new series title, then click **OK** to apply the new name or **Cancel** to close the dialog box without making a change.

To access the Change Series title dialog box, click **Chart Settings** in the Graph dialog box, then click the Series tab, then the **Title** button.

Chart Tools Gallery Dialog Box

Use the Chart Tools Gallery dialog box to add tools to your graph. For more information, see "Chart Options Dialog Box - Tools Tab" on page 10-621.

Click one of the following links to learn more about the Chart Tools Gallery dialog box:

- "Chart Tools Gallery Dialog Box - Series Tab"
- "Chart Tools Gallery Dialog Box - Axis Tab"
- "Chart Tools Gallery Dialog Box - Other Tab"

Chart Tools Gallery Dialog Box - Series Tab

Use the Series tab to add tools related to the series in your chart. The Series tab contains the following tools:

Cursor

Displays a draggable cursor line on top of the series. After you have added the Cursor tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Style	Lets you select a horizontal line, vertical line, or both as the format of the tool.
Snap	Causes the cursor tool to adhere to the selected series.
Follow Mouse	Causes the cursor tool to follow your movements of the mouse.

Pen	Lets you define the cursor tool. The Border Editor opens, see “Border Editor Dialog Box”.

Drag Marks

Lets you drag series marks. To use this tool, you must display the marks for a selected series, see “Marks Tab”. After you have added the Drag Marks tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Reset Positions	Moves any marks you have dragged back to their original position.

Drag Point

Lets you drag a series point. After you have added the Drag Point tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Style	Lets you constrain the movement of the series point to one axis or both (no constraint).
Mouse Button	Lets you select the mouse button you click to drag.
Cursor	Lets you select the appearance of the cursor when using the tool.

Draw Line

Lets you draw a line on the graph by dragging. After you have added the Draw Line tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Pen	Lets you define the line. The Border Editor opens, see “Border Editor Dialog Box”.
Button	Lets you select the mouse button you click to drag.

Enable Draw	Enables the Draw Line tool. Select this check box to let you draw lines, clear it to prevent you from drawing lines.
Enable Select	Lets you select and move lines that you have drawn. Select this check box, then click and drag the line you want to move. clear this check box if you want to prevent lines from being moved.
Remove All	Removes all lines you have drawn.

Gantt Drag

Lets you move and resize Gantt bars by dragging. This is unused by Bentley SewerGEMS V8*i*.

Image

Displays a picture using the selected series axes as boundaries. After you have added the Image tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Browse	Lets you navigate to and select the image you want to use. Browse is unavailable when there is a selected image. To select a new image, first clear the existing one.
Clear	Lets you remove a selected image. Clear is unavailable when there is no selected image.
Mode	Lets you set up the image you select. • **Normal**—Puts the background image in the top-left corner of the graph. • **Stretch**—Resizes the background image to fill the entire background of the graph. The image you select conforms to the series to which you apply it. • **Center**—Puts the background image in the horizontal and vertical center of the graph. • **Tile**—Repeats the background image as many times as needed to fill the entire background of the graph.

Mark Tips

Displays data in tooltips when you move the cursor over the graph. After you have added the Mark Tips tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool
Style	Lets you select what data the tooltips display.
Action	Sets when the tooltips display. Select **Click** if you want the tooltips to display when you click, or select **Move** if you want the tooltips to display when you move the mouse.
Delay	Lets you delay how quickly the tooltip displays.

Nearest Point

Lets you define and display an indicator when you are near a point in the selected series. After you have added the Nearest Point tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Fill	Lets you set the fill for the nearest-point indicator. The Hatch Brush Editor opens, see “Hatch Brush Editor Dialog Box”.
Border	Lets you set the outline of the nearest-point indicator. The Border Editor opens, see “Border Editor Dialog Box”.
Draw Line	Creates a line from the tip of the cursor to the series point.
Style	Sets the shape for the indicator
Size	Sizes the indicator.

Pie Slices

Outlines or expands slices of pie charts when you move the cursor or click them. This is unused by Bentley SewerGEMS V8*i*.

Series Animation

Animates series points. After you have added the Series Animation tool to your graph, you can modify the following settings:xxxx seems broken.

Series	Lets you select the series to which you want to apply the tool.
Steps	Lets you select the steps used in the animation. Set this control towards 100 for smoother animation and away from 100 for quicker, but less smooth animation.
Start at min. value	Lets you start the animation at the series' minimum value. clear this check box to set your own start value.
Start value	Sets the value at which the animation starts. To use this control, you must clear **Start at min. value**.
Execute!	Starts the animation.

Chart Tools Gallery Dialog Box - Axis Tab

Use the Axis tab to add tools related to the axes in your chart. The Axis tab contains the following tools:

Axis Arrows

Lets you add arrows to the axes. The arrows permit you to scroll along the axes. After you have added the Axis Arrows tool to your graph, you can modify the following settings:

Axis	Select the axis to which you want to add arrows.
Border	Lets you set the outline of the arrows. The Border Editor opens, see "Border Editor Dialog Box".
Fill	Lets you set the fill for the arrows. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Length	Lets you set the length of the arrows.
Inverted Scroll	Lets you change the direction in which the arrows let you scroll.

Scroll	Changes the magnitude of the scroll. Set a smaller percentage to reduce the amount of scroll caused by one click of an axis arrow, or set a larger percentage to increase the amount of scroll caused by a click.
Position	Lets you set an axis arrow at the start, end, or both positions of the axis.

Color Band

Lets you apply a color band to your graph for a range of values you select from an axis. After you have added the Color Band tool to your graph, you can modify the following settings:

Axis	Select the axis that you want to use to define the range for the color band.
Border	Lets you set the outline of the color band. The Border Editor opens, see "Border Editor Dialog Box".
Pattern	Lets you set the fill of the color band. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Gradient	Lets you set a gradient for the color band. A gradient overrides any solid color fill you might have set. The Gradient Editor opens, see "Gradient Editor Dialog Box".
Color	Lets you set a solid color for the color band. The Color Editor opens, see "Color Editor Dialog Box".
Start Value	Sets where the color band begins. Specify a value on the selected axis.
End Value	Sets where the color band ends. Specify a vale on the selected axis.
Transparency	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.

Draw Behind	Lets you position the color band behind the graphs. If you clear this check box, the color band appears in front of your graphs and hides them, unless you have transparency set.

Color Line

Lets you apply a color line, or plane in three dimensions, at a point you set at a value on an axis. After you have added the Color Line tool to your graph, you can modify the following settings:

Axis	Select the axis that you want to use to define the location for the line.
Border	Lets you set the outline of the color line. The Border Editor opens, see "Border Editor Dialog Box".
Value	Sets where the color line is. Specify a value on the selected axis.
Allow Drag	Lets you drag the line or lock the line in place. Select this check box if you want to permit dragging. clear this check box if you want the line to be fixed in one location.
Drag Repaint	Lets you smooth the appearance of the line as you drag it.
No Limit Drag	Lets you drag the line beyond the axes of the graph, or constrain the line to boundaries defined by those axes. Select this check box to permit unconstrained dragging.
Draw Behind	Lets you position the color line behind the graphs. If you clear this check box, the color band appears in front of your graphs. This is more noticeable in 3D graphs.
Draw 3D	Lets you display the line as a 2D image in a 3D chart. If you have a 3D chart (see "3D Tab"), clear this check box to display the line as a line rather than a plane.

Chart Tools Gallery Dialog Box - Other Tab

Use the Other tab to add tools to your chart, including annotations. The Other tab contains the following tools:

3D Grid Transpose

Swaps the X and Z coordinates to rotate the series through 90 degrees. This is unused by Bentley SewerGEMS V8*i*.

Annotation

Lets you add text to the chart. After you have added the Annotation tool to your graph, you can modify the following settings:

Options Tab	
Text	Lets you enter the text you want for your annotation.
Text alignment	Sets the alignment of the text inside the annotation box.
Cursor	Lets you set the style of the cursor when you move it over the annotation.
Position Tab	
Auto	Lets you select a standard annotation position.
Custom	Lets you select a custom position for the annotation. Select this check box to override the Auto setting and enable the Left and Top controls.
Left/Top	Lets you set a position from the Left and Top edges of the graph tab for the annotation.
Callout Tab	
Border	Lets you set up the leader line. The Border Editor opens, see “Border Editor Dialog Box”.
Pointer	Lets you set up the arrow head (if any) used by the leader line. The Pointer dialog box opens, see “Pointer Dialog Box”.
Position	Sets the position of the callout.

Distance	Lets you set the distance between the leader line and the graph of the selected series.
Arrow head	Lets you select the kind of arrow head you want to add to the leader line.
Size	Lets you set the size of the arrow head.

Format Tab

Color	Lets you set a color for the fill of the boxes. The Color Editor opens, see "Color Editor Dialog Box".
Frame	Lets you define the outline of the boxes. The Border Editor opens.
Pattern	Lets you set a pattern for the fill of the boxes. The Hatch Brush Editor opens, see "Hatch Brush Editor Dialog Box".
Round Frame	Lets you round the corners of the boxes. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the boxes as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see
Transparency	Lets you set transparency for the boxes, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Font	Lets you set the font properties for text. This opens the Windows Font dialog box.
Color	Lets you select the color for the text font. Double-click the colored square between Font and Fill to open the Color Editor dialog box.
Fill	Lets you set a pattern for the text font. The Hatch Brush Editor opens.

Shadow	Lets you set a shadow for the text. • **Visible**—Lets you display a shadow for the text. Select this check box to display the shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Format	**Format**—Lets you set up the gradient's properties. • **Visible**—Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient. • **Direction**—Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner. • **Angle**—Lets you customize the direction of the gradient beyond the Direction selections.

Colors	Lets you set the colors used for your gradients. The Start, Middle, and End selections open the Color Editor, see “Color Editor Dialog Box”. • **Start**—Lets you set the starting color for your gradient. • **Middle**—Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient. • **End**—Lets you select the final color for your gradient. • **Gamma Correction**—Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output. • **Transparency**—Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.
Options	Lets you control the affect of the start and end colors on the gradient, the middle color is not used. • **Sigma**—Lets you use the options controls. Select this check box to use the controls in the Options tab. • **Sigma Focus**—Lets you set the location on the chart background of the gradient’s end color. • **Sigma Scale**—Lets you control how much of the gradient’s end color is used by the gradient background.

Shadow Tab

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.

Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the outside of the legend.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the legend.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Page Number

Lets you add a page number annotation. For more information, see "Annotation".

Rotate

Lets you rotate the chart by dragging. After you have added the Rotate tool to your graph, you can modify the following settings:

Inverted	Reverses the direction of the rotation with respect to the direction you move the mouse.
Style	Lets you rotate horizontally, vertically, or both. Rotation is horizontal rotation about a vertical axis, whereas elevation is vertical rotation about a horizontal axis.
Outline	Lets you set the outline. The Border Editor opens, see "Border Editor Dialog Box".

TeeChart Gallery Dialog Box

Use the TeeChart Gallery dialog box to change the appearance of a series.

Series

The available series chart designs include:

- **Standard**
- **Stats**
- **Financial**
- **Extended**
- **3D**
- **Other**
- **View 3D**—Lets you view the chart design in two or three dimensions. Select this check box to view the charts in 3D, clear it to view them in 2D.
- **Smooth**—Smooths the display of the charts. Select this check box to smooth the display, clear it to turn off smoothing.

Functions

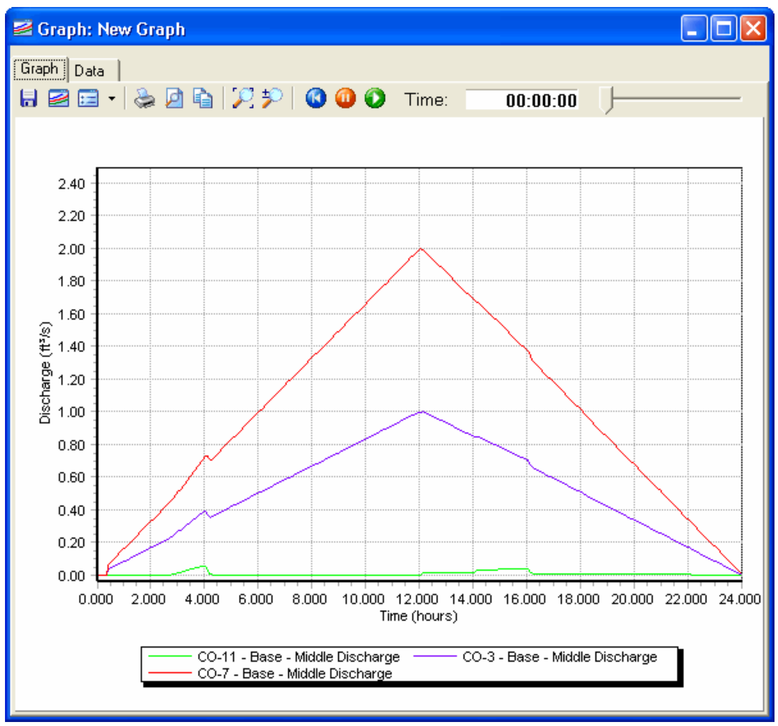
The available function chart designs include:

- **Standard**
- **Financial**
- **Stats**
- **Extended**
- **View 3D**—Lets you view the chart design in two or three dimensions. Select this check box to view the charts in 3D, clear it to view them in 2D.
- **Smooth**—Smooths the display of the charts. Select this check box to smooth the display, clear it to turn off smoothing.

Customizing a Graph

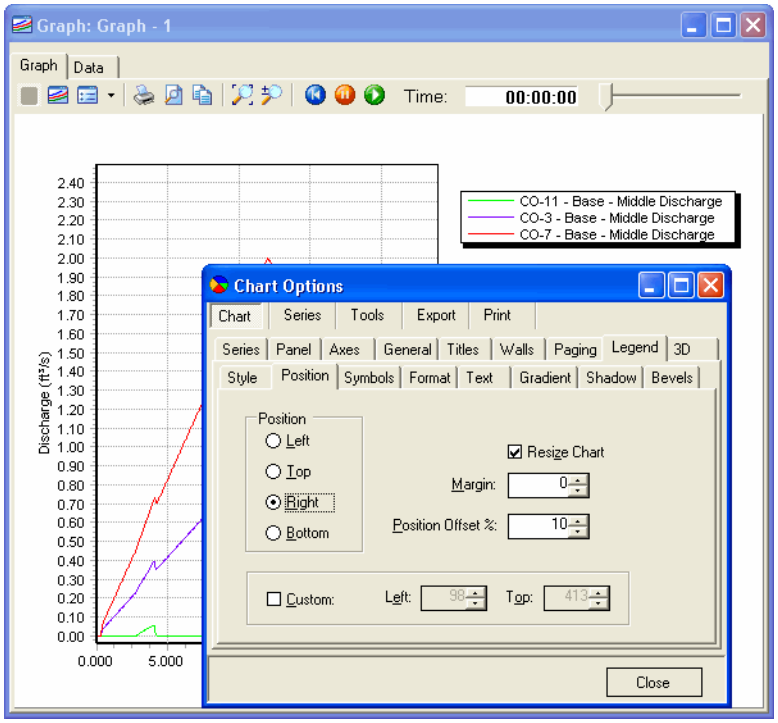
To customize a graph:

1. If you do not have your own model, open **Sample-1.swg**, one of the sample models that is included with Bentley SewerGEMS V8*i*.
2. Create a graph.
   a. Click **Compute**.
   b. Close the Calculation Executive Summary.
   c. Save your model.
   d. Right click an element, in Sample-1.swg, shift+click **CO-11**, **CO-3**, and **CO-7** to select them, then right-click one of them and select **Graph**.



   e. Click **Save** in the Graph dialog box, to add the graph to the Graph manager.

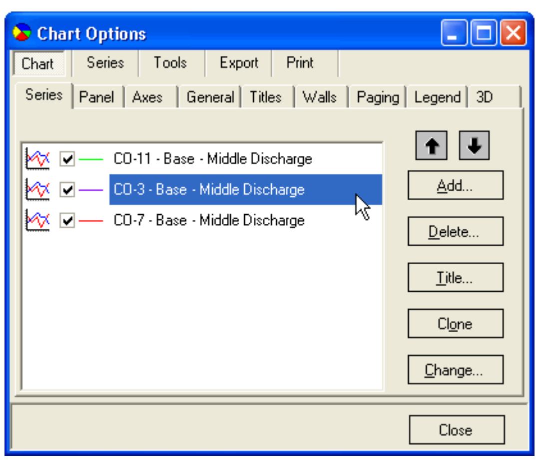
3. Move the legend.
   a. Click **Chart Settings**, to open the Chart Options dialog box.
   b. Click the **Chart** button, **Legend** tab, and **Position** subtab.
   c. Click the **Right** button in the Position area to set the legend to the right side of the graph. You can use other controls on this subtab to move the legend.



4. Change the line colors and weights.

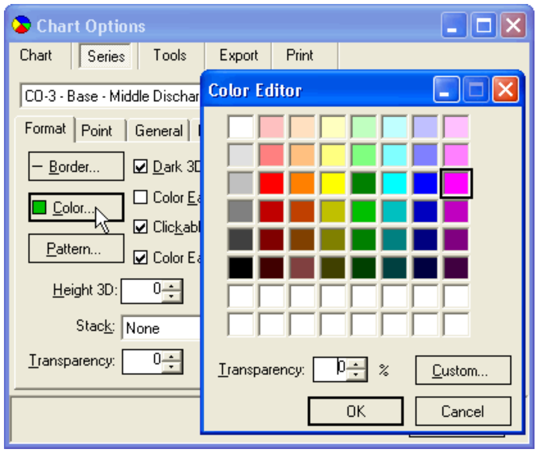
   a. Click **Chart Settings**, to open the Chart Options dialog box.

   b. In the Chart, Series tab click the series that you want to edit, to select and highlight it. You can select more than one series by Ctrl+ or Shift+clicking them.



   c. Click **Series** and select the **Format** tab.

d. Click the **Color** button and select a new color, to change the color of the line. The Color Editor dialog box opens (for more information, see "Color Editor Dialog Box" on page 10-627).



e. Click **OK** after you click the color you want to use. The series that are changed are those that you highlighted in the Chart, Series tab.

f. Click **Outline** to change the thickness of a line. The Border Editor dialog box open (for more information, see "Border Editor Dialog Box" on page 10-625).

g. Select **Visible**.

h. Change the **Width**.

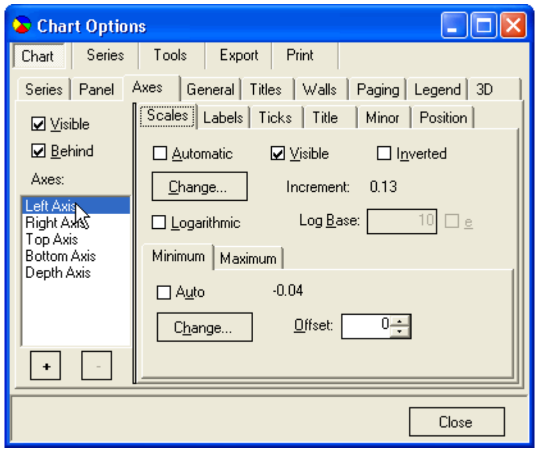
i. Make sure the Transparency is set to **0** if you want the line to appear opaque.

j. Click **OK** after you define the line width and attributes. The series that are changed are those that you highlighted in the Chart, Series tab.

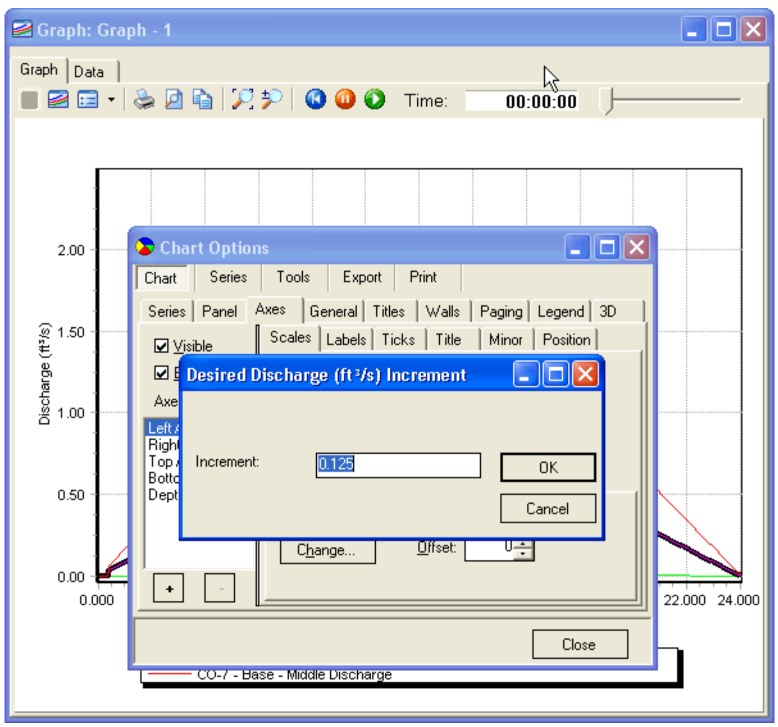
5. Change the interval between labels, grid, and ticks.

a. Click **Chart** > **Axes** > **Scales** > **Change** to change the interval between labels on the axes.

b. Select the Axis you want to change from the list of axes in the Axes area.



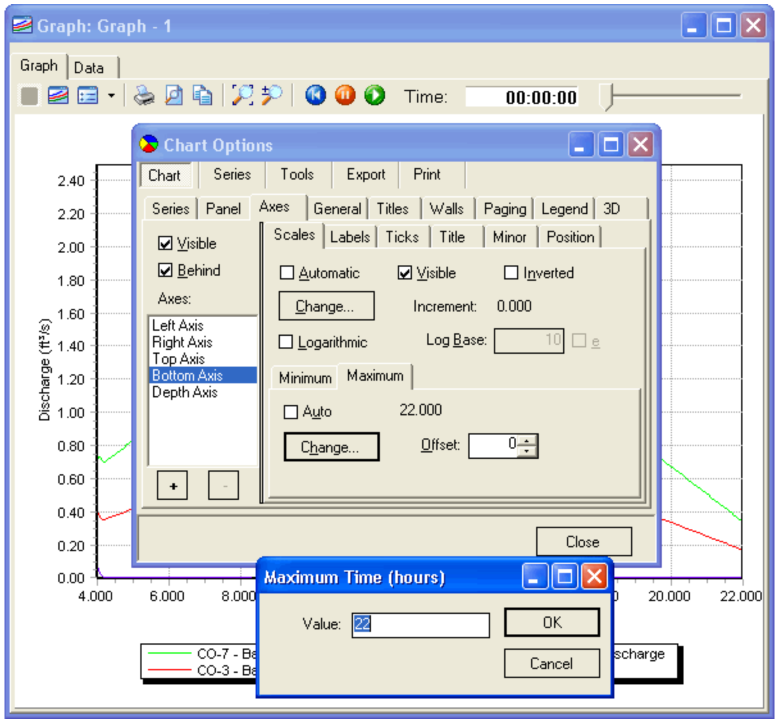
c. In the Increment dialog box, type the new value and click **OK**. This also changes the distance between major and minor ticks.



d. If needed, change the axis you have selected for changes.

e. Click **Chart > Axes > Minor** and change the **Count** to change the interval between minor ticks on the axes.

6. You can show and hide a grid associated with the major ticks.
    a. Click **Chart** > **Axes** > **Ticks** and click **Grid**.
    b. Select the axis on which you want to change the grid.
    c. In the Border Editor dialog box, select or clear **Visible** to show or hide the grid. (For more information, see "Border Editor Dialog Box" on page 10-625.)
7. You can show and hide a grid associated with the minor ticks.
    a. Click **Chart** > **Axes** > **Minor** and click **Grid**.
    b. Select the axis on which you want to change the grid.
    c. In the Border Editor dialog box, select or clear **Visible** to show or hide the grid.
8. You can set the minimum and maximum range for an axis.
    a. Click **Chart** > **Axes** > and select **Scales**.
    b. Select the axis on which you want to change the grid.
    c. Use the Minimum tab to change the minimum value for an axis. Clear the **Auto** check box.
    d. Click **Change**.
    e. Set the minimum value for the axis.
    f. Use the Maximum tab to change the maximum value for an axis. Clear the **Auto** check box.
    g. Click **Change**.
    h. Set the maximum value for the axis.



9. Change the background colors.

   a. Click **Chart** > **Panel** > and select **Background**.

   b. Use the **Color** and **Pattern** buttons to set a background color and/or pattern for the graph (see "Color Editor Dialog Box" on page 10-627 and "Hatch Brush Editor Dialog Box" on page 10-628).

10. Change the number of decimal places used in axis labels.

    a. Click **Chart** > **Axes** > **Labels** and select **Format**.

    b. Select the axis you want to change.

    c. Change the number of decimal places by making a selection from the **Values Format** drop-down list.

11. Change the fonts used by the axes and titles.

    a. Click **Chart** > **Axes** > **Labels** and select **Text**.

    b. Select the axis you want to change.

    c. Click **Font** to open the Font dialog box and change the format of the fonts used by the axis labels.

    d. Click **Chart** > **Axes** > **Title** and select **Text**.

e. Select the axis you want to change.

f. Click **Font** to open the Font dialog box and change the format of the fonts used by the axis title.

12. Add a text box to the graph.

    a. Click **Tools** > **Add** > **Other** > **Annotation**.

    b. In the Text pane, type the text you want in your annotation.

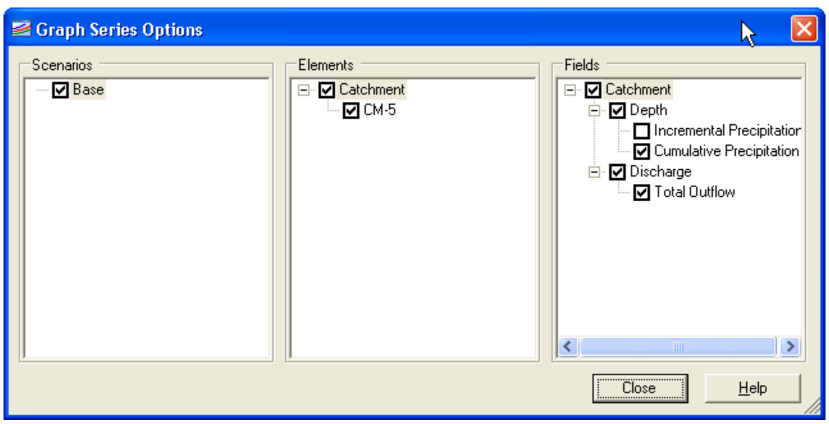
13. Plot rainfall and flow on the same graph for a catchment.

    a. Open **Sample-1.swg**.

    b. Click **Compute**.

    c. Right-click **CM-5** and select **Graph**.

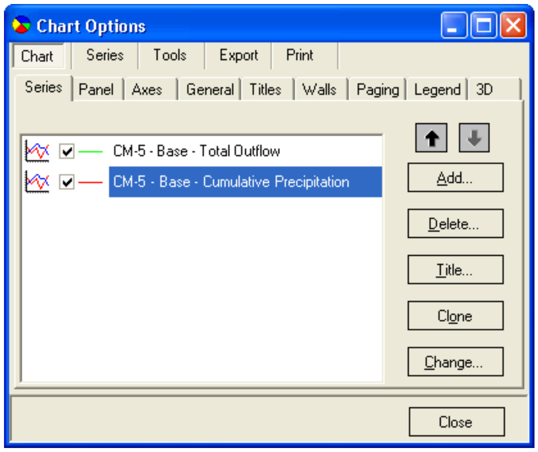
    d. In the Graph dialog box, click **Graph Series Options**.



    e. In the Graph Series Options dialog box, select **Precipitation (Cumulative)** and **Total Outflow**, then click **Close**. The Graph dialog box displays two graphs, one for total outflow and the other for cumulative precipitation.

    f. Click **Chart Settings**. The Graph Options dialog box opens.

    g. Click the **Precipitation (Cumulative)** series to select it.



h. Click **Change**.

i. Click the **Bar** graph type, to select it, then click **OK**.

j. Change the axis used by the bar graph.

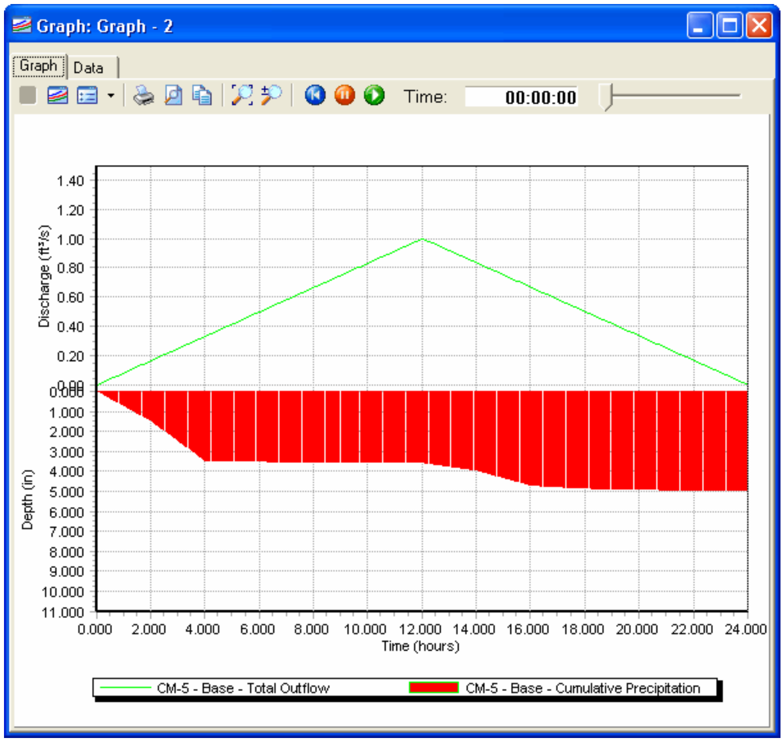
k. Click **Series**, then select **Precipitation (Cumulative)** in the drop-down list.

l. Click the General subtab, then change Vertical Axis from Left to **Custom 0**.

m. To disable marks, click the **Series** > **Marks** > **Style** subtab, and clear the **Visible** check box.

n. If you want to invert the Y-axis for the Precipitation (Cumulative) series, click **Chart** > **Axis**, select the **Custom 0** axis from the Axes list, and select the **Inverted** check box.

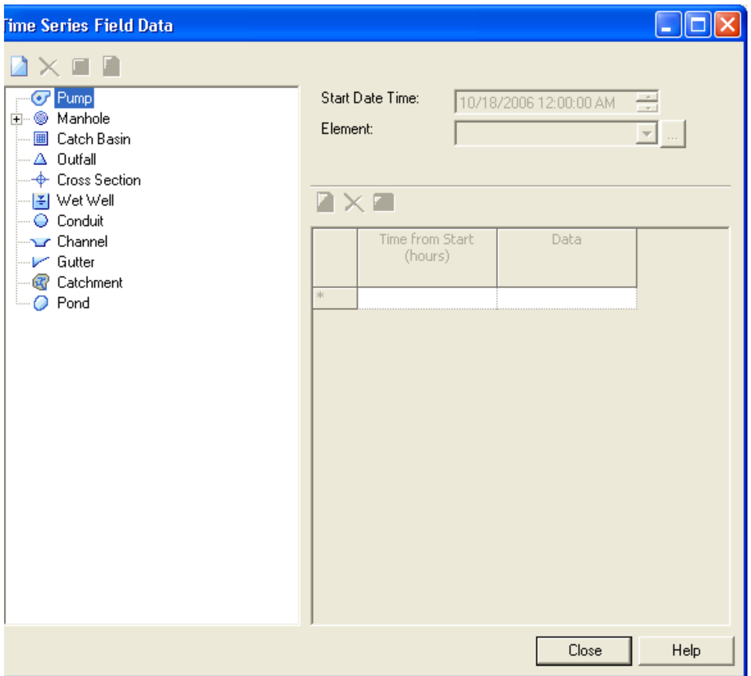
o. Close the Chart Options dialog box when you finish.



Time Series Field Data

The Time Series Field Data dialog allows you to enter your observed field data and compare it to the calculated results from the model in graph format. The dialog is accessed by clicking the Components menu and selecting Time Series Field Data.

Use this feature to display user-supplied time variant data values alongside calculated results in the graph display dialog. Model competency can sometimes be determined by a quick side by side visual comparison of calculated results with those observed and collection in the field



- **Get familiar with your data -** If you obtained your observed data from an outside source, you should take the time to get acquainted with it. Be sure to identify units of time and measurement for the data. Be sure to identify what the data points represent in the model; this helps in naming your line or bar series as it will appear in the graph.
- **Preparing your data -** Typically, observed data can be organized as a collection of points in a table. In this case, the time series data can simply be copied to the clipboard directly from the source and pasted right into the observed data input table. Ensure that your collection of data points is complete. That is, every value must have an associated time value. Oftentimes data points are stored in tab or comma delimited text files; these two import options are available as well.
- **Specifying the characteristics of your data -** To add time series field data, select the element for which the data applies and click the New button. Select the associated attribute for the data and click OK. You can change the label of the data set by highlighting it and clicking the Rename button. This label will appear in the Graph Series Options dialog when you create a graph. The following information must also be defined:
  – Start Date Time - Specify the date and time the field data was collected. This data will not affect the graph, but is valuable to you as a reference.

– Element - Choose the element that represents the field data measurement location. Click the ellipsis button to select the element from the drawing.

– Time From Start - Specify an offset of the start time and date for an EPS scenario.

– Attribute Value - Enter the value for the specified attribute at the specified Time from Start.

You can perform a quick graphical check on the data import by clicking the Graph button at the top of the data table.

If the number of observations is large, it is best to use the Copy/Paste commands. Copy the data from the original source to the clipboard, then go to the top of the Time from Start or Property (e.g. Flow) column and hit CTRL-V to paste the values into the appropriate column.

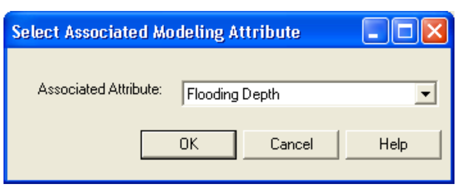
Click the Close button when done.

The data is saved with the model file. If you modify the source data file, the changes will not appear until time series data is imported again.

To add the time series field data to a graph, first create the graph of the property from an EPS model run (e.g. right click on element and pick Graph). In the Graph options dialog, select Time Series Field Data and then the name of the time series (in the Field pane (right pane). The field data will appear in the graph as points (by default) while the model results will appear as a continuous line. This can be changed using the Chart Settings button at the top of the graph (third from left).

Select Associated Modeling Attribute Dialog Box

This dialog appears when you create a new field data set in the Time Series Field Data dialog. Choose the attribute represented in the time series data source. The available attributes will vary depending on the element type chosen.



Print Preview Window

The Print Preview window can be used to print documents from Bentley SewerGEMS V8*i*, such as reports and graphs. This window lets you see the current view of the document as it will be printed and lets you define the print settings, such as selecting a printer to use.

The following controls are available in the Print Preview window:

Contents	Displays a table of contents for the document, if one is created.
Print	Opens the Print dialog box and lets you print the document as it appears in the preview pane. You can change printers in the Print dialog box, if you want.
Copy	Copies the document to the Windows clipboard, so you can paste it into other applications.
Find	Lets you search for words in the document. To find a word, click **Find**, in the Find dialog box, type the word you want to find, then click **Find Next**. Words that are found are highlighted in the print preview; click **Find Next** to continue searching the document.
Single/Multiple Page View	Displays the document as a single page or multiple pages in the preview pane.
Zoom In/Out	Enlarges or reduces the display of the document in the print preview. **Note: Changing the zoom only affects how the document displays on-screen, it does not affect how the document prints.**
Zoom Combo	Lets you select or type the amount of zoom used to display the document, where 100% is full size.
Previous Page	Displays to the previous page in the document.
Next Page	Displays the next page in the document.

Page	Displays the current page number and the total pages in the current chart. You can type the page number you want to display, and press **Enter** to display it.
Backward/Forward	Displays the page that was previously displayed. Backward and Forward are based on your navigation in the document and not on the page order of the document. For example, if you navigated from page 2 directly to page 6, clicking **Backward** would display page 2 again; if you then clicked **Forward**, page 6 would display again.

Contours

Using SewerGEMS V8*i* you can visually display calculated results for many attributes using contour plots.

The Contours dialog box is where all of the contour definitions associated with a project are stored. Choose View > Contours to open the Contours dialog box.

The dialog box contains a list pane that displays all of the contours currently contained within the project, along with a toolbar.

New	Opens the Contour Definition dialog box, allowing you to create a new contour.
Delete	Deletes the currently selected contour.
Rename	Renames the currently selected contour.
Edit	Opens the Contour Definition dialog box, where you can modify the settings of the currently selected contour.

Export	Clicking this button opens a submenu containing the following commands: • **Export to Shapefile** - Exports the contour to a shapefile, opening the Export to File Manager to select the shapefile. • **Export to DXF** - Exports the contour as a .dxf drawing. • **Export to Native Format** - Opens the Shapefile Properties dialog box, allowing you to add it to the Background Layers Manager.
View Contour Browser	Opens the Contour Browser dialog, allowing you to display detailed contour results for points in the drawing view.
Refresh	Regenerates the contour.
Shift Up	Moves the currently selected contour up in the list pane.
Shift Down	Moves the currently selected contour down in the list pane.
Help	Displays online help for the Contours.

Contour Definition

The Contour Definition dialog box contains the information required to generate contours for a calculated network.

Contour

Field	Select the attribute to apply the contour.

Selection Set	Apply an attribute to a previously defined selection set or to one of the following predefined options: • **All Elements** - Calculates the contour based on all elements in the model, including spot elevations. • **All Elements Without Spots** - Calculates the contour based on all elements in the model, except for spot elevations.
Minimum	Lowest value to be included in the contour map. It may be desirable to use a minimum that is above the absolute minimum value in the system to avoid creating excessive lines near a pump or other high-differential portions of the system.
Maximum	Highest value for which contours will be generated.
Increment	Step by which the contours increase. The contours created will be evenly divisible by the increment and are not directly related to the minimum and maximum values. For example, a contour set with 10 minimum, 20 maximum, and an increment of 3 would result in the following set: [12, 15, 18] not [10, 13, 16, 19].
Index Increment	Value for which contours will be highlighted and labeled. The index increment should be an even multiple of the standard increment.
Smooth Contours	The Contour Smoothing option displays the results of a contour map specification as smooth, curved contours.
Line Weight	The thickness of contour lines in the drawing view.

Color by Range	Contours are colored based on attribute ranges. Use the Initialize button to create five evenly spaced ranges and associated colors. **Initialize**—This button, located to the right of the Contour section, will initialize the Minimum, Maximum, Increment, and Index Increment values based on the actual values observed for the elements in the selection set. **Tip:** Initialization can be accomplished by clicking the Initialize button to automatically generate values for the minimum, maximum, increment, and index increment to create an evenly spaced contour set. **Ramp**—Automatically generate a gradient range between two colors that you specify. Pick the color for the first and last values in the list and the program will select colors for the other values.
Color by Index	The standard contours and index contours have separately controlled colors that you can make the contours more apparent.

Contour Plot

The Contour Plot window displays the results of a contour map specification as accurate, straight-line contours.

View the changes in the mapped attribute over time by using the animation feature. Choose Analysis > EPS Results Browser and click the **Play** button to automatically advance through the time step increments selected in the Increment bar.

The plot can be printed or exported as a .DXF file. Choose File > Export > DXF to export the plot.

Tip: Although the straight-line contours generated by this program are accurate, smooth contours are often more desirable for presentation purposes. You can smooth the contours by clicking Options and selecting Smooth Contours.

Note: Contour line index labels can be manually repositioned in this view before sending the plot to the printer. The Contour Plot Status pane displays the Z coordinate at the mouse cursor.

Contour Browser Dialog Box

The Contour Browser dialog box displays the X and Y coordinates and the calculated value for the contour attribute at the location of the mouse cursor in the drawing view.

Enhanced Pressure Contours

Normal contouring routines only include model nodes, such as junctions, tanks and reservoirs. When spot elevations are added to the drawing, however, you can create more detailed elevation contours and enhanced pressure contours.

These enhanced contours include not only the model nodes but also the interpolated and calculated results for the spot elevations. Enhanced pressure contours can help the modeler to understand the behavior of the system even in areas that have not been included directly in the model.

Using Named Views

The Named View dialog box is where you can store the current views X and Y coordinates. When you set a view in the drawing pane and add a named view, the current view is saved as the named view. You can then center the drawing pane on the named view with the **Go To View** command.

Choose View > Named Views to open the Named View dialog box.

The toolbar contains the following controls:

New	Contains the following commands: • **Named View**—Opens a Named View Properties box to create a new named view. • **Folder**—Opens a Named Views Folder Properties box to enter a label for the new folder.
Delete	Deletes the named view or folder that is currently selected.
Rename	Rename the currently selected named view or folder.
Go to View	Centers the drawing pane on the named view.
Shift Up and Shift Down	Moves the selected named view or folder up or down.
Expand All or Collapse All	Expands or collapses the named views and folders.
Help	Displays online help for Named Views.

Using Aerial View

The Aerial View is a small navigation window that provides a graphical overview of your entire drawing. You can toggle the Aerial View window on or off by selecting View > Aerial View to open the Aerial View window.

A Navigation Rectangle is displayed in the Aerial View window. This Navigation Rectangle provides a you-are-here indicator showing you current zoom location respective of the overall drawing. As you pan and zoom around the drawing, the Navigation Rectangle will automatically update to reflect your current location.

You can also use the Aerial View window to navigate around your drawing. To pan, click the Navigation Rectangle to drag it to a new location. To zoom, click anywhere in the window to specify the first corner of the Navigation Rectangle, and click again to specify the second corner.

In AutoCAD mode, see the AutoCAD online help for a detailed explanation.

In Stand-Alone mode, with Aerial View window enabled (by selecting the View > Aerial View), click and drag to draw a rectangular view box in the aerial view. The area inside this view box is displayed in the main drawing window. Alternately, any zooming or panning action performed directly in the main window updates the size and location of the view box in the Aerial View window.

The Aerial View window contains the following buttons:

Zoom Extents—Display the entire drawing in the Aerial View window.

Zoom In—Decrease the area displayed in the Aerial View window.

Zoom Out—Increase the area displayed in the Aerial View window.

Help—Opens the online help.

To resize the view box directly from the Aerial View window, click to define the new rectangular view box. To change the location of the view box, hover the mouse cursor over the current view rectangle and click to drag the view box frame to a new location.

Chapter

Working in ArcGIS Mode 11

GIS Basics

Bentley SewerGEMS V8*i* provides three environments in which to work: Bentley SewerGEMS V8*i* Modeler Mode, AutoCAD Integrated Mode, and ArcMap Integrated Mode. Each mode provides access to differing functionality—certain capabilities that are available within Bentley SewerGEMS V8*i* Modeler mode may not be available when working in ArcMap Integrated mode, and vice-versa. In addition, you can use ArcCatalog to perform actions on any Bentley SewerGEMS V8*i* database. Some of the advantages of working in GIS mode include:

- Full functionality from within the GIS itself, without the need for data import, export, or transformation
- The ability to view and edit multiple scenarios in the same geodatabase
- Minimizes data replication
- GIS custom querying capabilities
- Lets you build models from scratch using practically any existing data source
- Utilize the powerful reporting and presentation capabilities of GIS

A firm grasp of GIS basics will give you a clearer understanding of how Bentley SewerGEMS V8*i* interacts with GIS software. Click one the following links to learn more:

- "GIS Terms and Definitions"
- "ArcGIS Integration"
- "ArcGIS Applications"

GIS Terms and Definitions

ArcObjects:	ArcObjects is the framework upon which ArcGIS has been built. It is a collection of software components based on the COM protocol, which allows for the customization and extension of the core software functionality.
Coverage:	A collection of data that has a common theme, and is considered a single unit.
Feature Class:	**1.** A classification describing the format of geographic features and supporting data in a coverage. Coverage feature classes for representing geographic features include point, arc, node, route-system, route, section, polygon and region. One or more coverage features are used to model geographic features; for example, arcs and nodes can be used to model linear features such as street centerlines. The tic, annotation, link, and boundary feature classes provide supporting data for coverage data management and viewing. **2.** The conceptual representation of a geographic feature. When referring to geographic features, feature classes include point, line, area, and surface.
Feature Dataset:	A feature dataset is a collection of feature classes that share the same spatial reference.
GEMS Datastore:	The relational database that Bentley SewerGEMS V8*i* uses to store model data. Each Bentley SewerGEMS V8*i* project uses two main files for data storage, the datastore (.MDB) and the Bentley SewerGEMS V8*i* Modeler-specific data (.swg). Although the Bentley SewerGEMS V8*i* datastore is an .mdb file, **cannot** be a geodatabase.
Geocode:	The process of identifying the coordinates of a location given its address. For example, an address can be matched against a TIGER street network to determine the location of a home. Also referred to as address geocoding.
Geodatabase:	Short for geographic database, a geodatabase stores spatial and descriptive data in an efficient manner. Geodatabases are the standard file format for ArcGIS v8 and later.

Layer:	Layers contain spatial data according to similar subject matter. Conceptually, layers in a database or map library environment are exactly like coverages. Layers are the standard GIS data format for ArcView 3.x and earlier.
Metadata:	Additional information (aside from tabular and spatial data) that makes the data useful. Includes characteristics and information that are required to use the data but are not contained within the data itself.
Relate:	A temporary connection between table records using a common item shared by both tables. Each record in one table is connected to those records in the other table that share the same value for the common item.
Relational Database:	A database in which the data is structured in such a way as to associate tables according to attributes that are shared by the tables.
Relational Join:	The process of merging two attribute tables using a common item.
Shapefile:	A file format that stores spatial and attribute data for the spatial features within the dataset. A shapefile consists of a main file, an index file, and a dBASE table. Shapefiles were the standard file storage format for ArcView 3.x and earlier.
Spatial Reference:	The spatial reference for a feature class describes its coordinate system (for example, geographic, UTM, and State Plane), its spatial domain, and its precision. The spatial domain is best described as the allowable coordinate range for X and Y coordinates, m-(measure) values, and z-values. The precision describes the number of system units per one unit of measure. A spatial reference with a precision of 1 will store integer values, while a precision of 1000 will store three decimal places.

ArcGIS Integration

Bentley SewerGEMS V8*i* features full integration with ESRI’s ArcGIS software, including ArcView, ArcEdit, and ArcInfo. The following is a description of the functionality available with each of these packages:

- **ArcView**—ArcView provides the following capabilities:
  - Data Access

- Mapping
- Customization
- Spatial Query
- Simple Feature Editing

ArcView can edit shapefiles and personal geodatabases that contain simple features such as points, lines, polygons, and static annotation. Rules and relationships can not be edited with ArcView.

- **ArcEdit**—ArcEdit provides all of the capabilities available with ArcView in addition to the following:
  - Coverage and geodatabase editing

  ArcEdit can edit shapefiles, coverages, personal geodatabases, and multi-user geodatabases.

- **ArcInfo**—ArcInfo provides all of the capabilities available with ArcEdit in addition to the following:
  - Advanced geoprocessing
  - Data conversion
  - ArcInfo Workstation

  ArcInfo can edit shapefiles, coverages, personal geodatabases, and multi-user geodatabases.

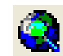
ArcGIS Integration with Bentley SewerGEMS V8*i*

When you install Bentley SewerGEMS V8*i* after you install ArcGIS, integration between the two is automatically configured when you install Bentley SewerGEMS V8*i*.

If you install ArcGIS after you install Bentley SewerGEMS V8*i*, you must manually integrate the two by selecting Run > All Programs > Haestad Methods >Bentley SewerGEMS V8*i* > Integrate Bentley SewerGEMS V8*i* with AutoCAD-ArcGIS. The integration utility runs automatically. You can then run Bentley SewerGEMS V8*i* in ArcGIS mode.

ArcGIS Applications

ArcView, ArcEdit, and ArcInfo share a common set of applications, each suited to a different aspect of GIS data management and map presentation. These applications include ArcCatalog and ArcMap.

- **ArcCatalog**—ArcCatalog is used to manage spatial data, database design, and to view and record metadata. 
- **ArcMap**—ArcMap is used for mapping, editing, and map analysis. ArcMap can also be used to view, edit, and calculate your Bentley SewerGEMS V8*i* model. 

Using ArcCatalog with a Bentley SewerGEMS V8*i* Database

You can use ArcCatalog to manage spatial data, database design, and to view and record metadata associated with your Bentley SewerGEMS V8*i* databases.

For more information, see the following topic:

- "ArcCatalog Geodatabase Components" on page 11-669

ArcCatalog Geodatabase Components

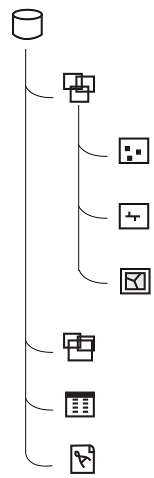
Many of the components that can make up a geodatabase can be directly correlated to familiar Bentley SewerGEMS V8*i*/Bentley SewerGEMS V8*i* conventions. The following diagram illustrates some of these comparisons.

The Bentley SewerGEMS V8*i* ArcMap Client

The Bentley SewerGEMS V8*i* ArcMap client refers to the environment in which Bentley SewerGEMS V8*i* is run. As the ArcMap client, Bentley SewerGEMS V8*i* runs within ESRI's ArcMap interface, allowing the full functionality of both programs to be utilized simultaneously.

Getting Started with the ArcMap Client

An ArcMap Bentley SewerGEMS V8*i* project consists of:



This ArcCatalog Icon	Represents	And in WaterGEMS is Comparable to
	Geodatabase	Project
	Feature Dataset	Scenario
	Point Feature Class	Element Types (Reservoirs, Junctions, Tanks, Pumps, Valves and Meters)
	Line Feature Class	Element Types (Pipes)
	Polygon Feature Class	Element Types (Thiessen Polygons)
	CAD Dataset	AutoCAD Drawing File (.dwg)
	Geodatabase Table	Attribute Data (Elevations, Demand, etc.)
	CAD Drawing	DXF Background

- **A** Bentley SewerGEMS V8*i* **.mdb file**—this file contains all modeling data, and includes everything needed to perform a calculation.
- **A** Bentley SewerGEMS V8*i* **.swg file**—this file contains data such as annotation and color-coding definitions.
- **A geodatabase association**—a project must be linked to a new or existing geodatabase.

Note: You must be in an edit session (Click the ArcMap Editor button and select the Start Editing command) to access the various Bentley SewerGEMS V8*i* editors (dialogs accessed with an ellipsis (...) button) through the Property Editor, Alternatives Editor, or FlexTables, even if you simply wish to view input data and do not intend to make any changes.

There are a number of options for creating a model in the ArcMap client:

- **Create a model from scratch**—You can create a model in ArcMap. You'll first need to create a new project and attach it to a new or existing geodatabase. See "Managing Projects In ArcMap" on page 11-676 and "Attach Geodatabase Dialog" on page 11-678 for further details. You can then lay out your network using the Bentley SewerGEMS V8*i* toolbar. See "Laying out a Model in the ArcMap Client" on page 11-678.

- **Open a previously created** Bentley SewerGEMS V8*i* **project**—You can open a previously created Bentley SewerGEMS V8*i* model. If the model was created in the Stand Alone version, you must attach a new or existing geodatabase to the project. See "Managing Projects In ArcMap" on page 11-676 and "Attach Geodatabase Dialog" on page 11-678 for further details.
- **Import a model that was created in another modeling application**—You can import a model that was created in SewerCAD or EPA SWMM. See "Importing Data From Other Models" on page 4-162 for further details.

Warning! You cannot use a Bentley SewerGEMS V8*i* .mdb file as a geodatabase. Make sure that you do not attempt to use the same file name for both the Bentley SewerGEMS V8*i* database (swg.mdb) and the geodatabase .mdb.

Bentley SewerGEMS V8i Toolbar

The SewerGEMS V8*i* toolbar offers the following functionality:

- **Project**—The Project menu contains the following commands:
  - **Add New Project**—Creates a new Bentley SewerGEMS V8*i* project. When you select this command, a Save As dialog box appears, allowing you to define a name and directory location for the new project.
  - **Add Existing Project**—Opens an existing project. When you select this command, an Open dialog box appears, allowing you to browse to the project to be opened.
  - **ProjectWise**—The ProjectWise submenu contains the following commands:
    - **Open**—Allows you to open a Bentley SewerGEMS V8*i* project from a ProjectWise datasource. You must have a ProjectWise license to perform this operation.
    - **Save As**—Allows you to save the current project to a ProjectWise datasource. You must have a ProjectWise license to perform this operation.
    - **Change Datasource**—Allows you to change the default ProjectWise datasource. You must have a ProjectWise license to perform this operation.
  - **Import**—Opens a submenu containing the following commands:
    - **SWMM v4**—Opens a Windows Browse dialog box, allowing you to choose the SWMM v4 file to import.

Note: Refer to the SWMM Readme.txt file for more information regarding the SWMM 4 to SWMM 5 Converter application

  - **SWMM v5**—Opens a Windows Browse dialog box, allowing you to choose the SWMM v5 file to import.
  - Bentley SewerGEMS V8*i* **Database**—Lets you import a Bentley SewerGEMS V8*i* project database file.
  - **SewerCAD Exchange Database**—Lets you import a SewerCAD export file (.mdb file).
- **Export**—Opens a submenu containing the following command:
  - **SWMM v5**—Lets you export the current project to SWMM format.
- **Project Properties**—Opens the Project Properties dialog.

- **Edit**—The Edit menu contains the following commands:
  - **Select All**—Selects all of the elements in the network.
  - **Select By Element**—Opens a submenu listing all available element types. Select one of the element types from the submenu to select all elements of that type in the model.
  - **Clear Selection**—Deselects the currently selected element(s).
  - **Find Element**—Lets you find a specific element by entering the element's label.
- **Analysis**—The Analysis menu contains the following commands:
  - **LoadBuilder**—Opens the LoadBuilder Manager dialog.
  - **Storm Events**—Opens the Storm Events dialog box, which lets you create, edit, and delete storm events. These storms are available for you to select for a catchment.
  - **Global Storm Events**—Opens the Global Storm Event Settings dialog box, which lets you define project-wide global storm event data.
  - **Pipe Catalog**—Opens the Catalog Pipe dialog box, which lets you create, edit, and view catalog pipes. Catalog pipes are an efficient way to reuse common physical pipe definitions.
  - **Pump Curve Definitions**—Opens the Pump Curve Definitions dialog box, which lets you view, edit, and create pump curve definitions.
  - **SWMM Extensions**—Opens a submenu containing the following SWMM-specific commands:
    - **Evaporation**—Opens the Evaporation dialog box, allowing you to view and edit evaporation data for use in SWMM calculations.
    - **Aquifers**—Opens the Aquifers dialog box, allowing you to view and edit aquifer data for use in SWMM calculations.

    - **Control Sets**—Opens the Control Sets dialog box, allowing you to view, edit, and create control sets for use in SWMM calculations.
    - **Pollutants**—Opens the Pollutants dialog box, allowing you to view and edit pollutant data for use in SWMM calculations.
    - **Land Uses**—Opens the Land Use dialog box, allowing you to view and edit land use data for use in SWMM calculations.
  - **Unit Sanitary Loads**—Opens the Unit Sanitary (Dry Weather) Loads dialog box, which lets you create, edit, and delete unit sanitary loads.
  - **Patterns**—Opens the Pattern Manager where you can create and edit diurnal loading patterns for use with extended period simulations.
  - **Pattern Setups**—Opens the Pattern Setup Manager where you can associate diurnal patterns with the appropriate unit sanitary loads for a given scenario.
  - **Validate**—Runs a diagnostic check on the network data to alert you to possible problems that may be encountered during calculation. This is the manual validation command, and it checks for input data errors. It differs in this respect from the automatic validation that SewerGEMS V8*i* runs when the compute command is initiated, which only checks for network connectivity errors.
  - **Compute Hydrology**—Lets you perform the hydrologic calculations for the current scenario.
  - **Compute**—Calculates the network. Before calculating, an automatic validation routine is triggered, which checks the model for network connectivity errors. Note that this automatic validation does not check for input data errors; to check for these types of errors, you must manually validate the model using the Validate command.
  - **Always Compute Hydrology**—Lets you turn hydrology calculations on and off whenever the model is calculated. Turning hydrology computation off improves performance and is recommended when the hydrology input will not change.
- **View**—The View menu contains the following commands:
  - **Project Manager**—Opens the Project Manager dialog. (See "Managing Projects In ArcMap" on page 11-676)
  - **Scenarios**—Opens the Scenario Manager, which lets you create, view, and manage project scenarios.
  - **Alternatives**—Opens the Alternative Manager, which lets you create, view, and manage alternatives.
  - **Calculation Options**—Opens the Calculation Options Manager, which lets you create, view, and manage calculation settings for the project.
  - **User Notifications**—Opens the User Notifications Manager, allowing you to view warnings and errors uncovered by the validation process.

- **Selection Sets**—Opens the Selection Sets Manager, which lets you create, view, and manage selection sets associated with the project.
- **Network Navigator**—Opens the Network Navigator manager, which lets you quickly navigate to and review any group of elements.
- **Properties**—Turns the Properties Editor display on or off.
- **Flextables**—Opens the FlexTables Manager, which lets you create, view, and manage the tabular reports for the project.
- **Graphs**—Opens the Graph Manager, which lets you create, view, and manage graphs for the project.
- **Profiles**—Opens the Profile Manager, which lets you create, view, and manage the profiles for the project.
- **EPS Results Browser**—Opens the Scenario Animation manager, which lets you manipulate the currently displayed time step and to animate the drawing pane.
- **Auto-Refresh**—Turns automatic updates to the main window view on or off whenever changes are made to the Bentley SewerGEMS V8*i* datastore. When selected, a check mark appears next to this menu command, indicating that automatic updates are turned on.
- **Refresh Drawing**—Updates the main window view according to the latest information contained in the Bentley SewerGEMS V8*i* datastore.
- **Apply** SewerGEMS V8*i* **Renderer**—Allows you to toggle the Bentley SewerGEMS V8*i* renderer on/off. When the renderer is On, Bentley SewerGEMS V8*i* color coding settings are applied and inactive elements are displayed in the color specified in the Inactive Topology Color field of the Bentley SewerGEMS V8*i* Global Options tab (to access Options, click the Bentley SewerGEMS V8*i* menu and select the Tools...Options command.)
- **Show Flow Arrows**—Allows you to toggle on/off flow arrow visibility.

• **Tools**—The Tools menu contains the following commands:

- **ModelBuilder**—Opens the ModelBuilder Connections Manager. (See “ModelBuilder Connections Manager” on page 5-173**)**
- **Thiessen Polygon**—Opens the Thiessen Polygon Input dialog. (See “Generating Thiessen Polygons” on page 11-691)
- **Engineering Libraries**—Opens the Engineering Libraries Manager.
- **Database Utilities**—Opens a submenu containing the following commands:

- **Compact Database**—When you delete data from a Bentley SewerGEMS V8*i* project, such as elements or alternatives, the database store that Bentley SewerGEMS V8*i* uses can become fragmented, causing data storage to be inefficient. This can cause unnecessarily large data files, which impact performance substantially. Compacting the database eliminates the empty data records, thereby defragmenting the datastore and improving the performance of the file.

Note: Every tenth time a file is opened, Bentley SewerGEMS V8*i* will automatically prompt you to compact the database. Click Yes to compact the database, or no to close the prompt dialog box without compacting. Since compacting the database can take time, especially for larger models, you way want to postpone the compact procedure until a later time.

- **Synchronize Drawing**—Synchronizes the current model with the project database.

– **Options**—Opens the Options dialog box, which lets you change global settings such as display pane settings, units, display precision and format used, and element labeling.

- **Report**—The Report menu contains the following commands:
  - **Element Tables**—Opens a submenu containing a command for each of the element types, each of which opens a tabular report containing all of the input data and results for every instance of the selected element in the model.
  - **Layer Symbology**—Opens the Color Coding Properties dialog. Bentley SewerGEMS V8*i* color coding settings are only applied when the Bentley SewerGEMS V8*i* Renderer is toggled On (to turn on the Bentley SewerGEMS V8*i* renderer, click the Bentley SewerGEMS V8*i* menu and select the View...Apply Bentley SewerGEMS V8*i* Renderer command).
  - **Scenario Summary**—Opens the Scenario Summary Report.
  - **Calculation Executive Summary**—Opens the calculation executive summary report, which reports a summary of the calculations performed on your model.
  - **Calculation Detailed Summary**—Opens the calculation detailed summary report, which reports the details of the calculations performed on your model.
  - **Project Inventory**—Opens the Project Inventory Report, which contains the number of each of the various element types that are in the network.
- **Help**—The Help menu contains the following commands:
  - Bentley SewerGEMS V8*i* **Help**—Opens the online help Table of Contents.
  - **Reference Tables**—Opens the online help to the Reference Tables help topic.

- **Introduction to** SewerGEMS V8*i*—Starts the Introduction to Bentley SewerGEMS V8*i* multimedia presentation.
- **QuickStart Lessons**—Opens the online help to the Quick Start Lessons Overview topic.
- **Check for Updates**—Opens your Web browser to our Web site, allowing you to check for Bentley SewerGEMS V8*i* updates.
- **Training**—Opens your browser to the Continuing Education page of our Web site.
- **Services**—Opens a submenu containing the following commands:
  - **Contents**—Opens a locally stored htm page containing links for Haestad Methods products and services.
  - **Multimedia CD**—Starts the Virtual Tour, a multimedia presentation that includes information about Haestad Methods products and services.
  - **Online Forums**—Opens your browser to the online forum page of our Web site.
  - **Haestad.com**—Opens your browser to the main page of our Web site.
  - **CivilQuiz.com**—Opens your browser to our CivilQuiz Web site.
- **About** SewerGEMS V8*i*—Opens the About Bentley SewerGEMS V8*i* dialog box, which displays copyright information about the product, registration information, and the current version number of this release.

Managing Projects In ArcMap

The Bentley SewerGEMS V8*i* ArcMap client utilizes a Project Manager to allow you to disconnect and reconnect a model from the underlying geodatabase, to view and edit multiple projects, and to display multiple projects on the same map.

The Project Manager lists all of the projects that have been opened during the ArcMap session. The following controls are available:

- **Add**—Clicking the Add button opens a submenu containing the following commands:
  - **Add New Project**—Opens a Save As dialog, allowing you to specify a project name and directory location. After clicking the Save button, the Attach Geodatabase dialog opens, allowing you to specify a new or existing geodatabase to be connected to the project.

  - **Add Existing Project**—Opens an Open dialog, allowing you to browse to the Bentley SewerGEMS V8*i* project to be added. If the Bentley SewerGEMS V8*i* project is not associated with a geodatabase, the Attach Geodatabase dialog opens, allowing you to specify a new or existing geodatabase to be connected to the project.
- **Open Project**—Opens the project that is currently highlighted in the Project Manager list pane. You can only edit projects that are currently open. This command is available only when the currently highlighted project is closed.
- **Save Project**—Saves the project that is currently highlighted in the Project Manager list pane. This command is available only when changes have been made to the currently highlighted project.
- **Close Project**—Closes the project that is currently highlighted in the Project Manager list pane. Closed projects cannot be edited, but the elements within the project will still be displayed in the map. This command is available only when the currently highlighted project is open.
- **Remove Project**—Removes the project that is currently highlighted in the Project Manager list pane. This command permanently breaks the connection to the geodatabase associated with the project.
- **Make Current**—Makes the project that is currently highlighted in the Project Manager list pane the current project. Edits made in the map are applied to the current project. This command is available only when the currently highlighted project is not marked current.
- **Help**—Opens the online help.

To add a new project:

1. From the Project Manager, click the Add button and select the Add New Project command. Or, from the Bentley SewerGEMS V8*i* menu, click the Project menu and select the Add New Project command.
2. In the Save As dialog that appears, specify a name and directory location for the new project, then click the Save button.
3. In the Attach Geodatabase dialog that appears, click the Attach Geodatabase button. Browse to an existing geodatabase to import the new project into, or create a new geodatabase by entering a name for the geodatabase and specifying a directory. Click the Save button.
4. Enter a dataset name.
5. You can assign a spatial reference to the project by clicking the Change button, then specifying spatial reference data in the Spatial Reference Properties dialog that appears.
6. In the Attach Geodatabase dialog, click the OK button to create the new project.

To add an existing project:

1. From the Project Manager, click the Add button and select the Add Existing Project command. Or, from the Bentley SewerGEMS V8*i* menu, click the Project menu and select the Add Existing Project command.
2. In the Open dialog that appears, browse to the location of the project, highlight it, then click the Open button.
3. If the project is not associated with a geodatabase, the Attach Geodatabase dialog opens, allowing you to specify a new or existing geodatabase to be connected to the project. Continue to Step 4. If the project has already been associated with a geodatabase, the Attach Geodatabase will not open, and the project will be added.
4. In the Attach Geodatabase dialog, click the Attach Geodatabase button. Browse to an existing geodatabase to import the new project into, or create a new geodatabase by entering a name for the geodatabase and specifying a directory. Click the Save button.

Attach Geodatabase Dialog

The Attach Geodatabase dialog allows you to associate a Bentley SewerGEMS V8*i* project with a new or existing geodatabase, and also provides access to the ArcMap Spatial Reference Properties dialog, allowing you to define the spatial reference for the geodatabase.

The following controls are available:

- **Geodatabase Field**—This field displays the path and file name of the geodatabase that was selected to be associated with the project.
- **Geodatabase Button**—This button opens an Import To or Create New Geodatabase dialog, where you specify an existing geodatabase or enter a name and directory for a new one.
- **Dataset Name**—Allows you to enter a name for the dataset.
- **Spatial Reference Pane**—Displays the spatial reference currently assigned to the geodatabase.
- **Change Button**—Opens the Spatial Reference Properties dialog, allowing you to change the spatial reference for the geodatabase.

Laying out a Model in the ArcMap Client

The Bentley SewerGEMS V8*i* toolbar contains a set of tools similar to the Stand-Alone version. See “Layout Toolbar” on page 2-33 for descriptions of the various element layout tools.

You must be in an edit session (Click the ArcMap Editor button and select the Start Editing command) to lay out elements or to enter element data in ArcMap. You must then Save the Edits (Click the ArcMap Editor button and select the Save Edits command) when you are done editing. The tools in the toolbar will be inactive when you are not in an edit session.

Using LoadBuilder to Assign Loading Data

LoadBuilder simplifies and expedites the process of assigning loading data to your model, using a variety of source data types. LoadBuilder is available from within the ArcMap client interface only.

Note: The loading output data generated by LoadBuilder is a Base Flow, i.e., a single value that remains constant over time.

After running LoadBuilder and exporting the results, you may need to modify your data to reflect changes over time by applying patterns to the base flow values.

LoadBuilder includes:

- “LoadBuilder Manager” on page 11-680
- “LoadBuilder Wizard” on page 11-680

LoadBuilder Manager

The LoadBuilder manager provides a central location for the creation, storage, and management of Load Build templates. The following buttons are available from this dialog box:

	**New**	Opens the LoadBuilder Wizard.
	**Delete**	Deletes an existing LoadBuilder template.
	**Rename**	Rename an existing LoadBuilder template.
	**Edit**	Opens the LoadBuilder Wizard, with the settings associated with the currently highlighted definition loaded.
	**Help**	Opens the context-sensitive online help.

LoadBuilder Wizard

The LoadBuilder wizard assists you in the creation of a new load build template by stepping you through the procedure of creating a new load build template. Depending on the load build method you choose, the specific steps presented in the wizard will vary.

Note: The loading output data generated by LoadBuilder is a Base Flow, i.e., a single value that remains constant over time.

After running LoadBuilder and exporting the results, you may need to modify your data to reflect changes over time by applying patterns to the base flow values.

LoadBuilder wizard includes:

- “Step 1: Load Method to Use” on page 11-681
- “Step 2: Input Data” on page 11-683

- “Step 3: Calculation Summary” on page 11-689
- “Step 4: Results Preview” on page 11-690
- “Step 5: Completing the LoadBuilder Wizard” on page 11-690

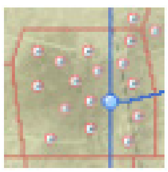
Step 1: Load Method to Use

In this step, the Load Method to be used is specified. The next steps will vary according to the load method that is chosen. The load methods are divided into three categories; the desired category is selected by clicking the corresponding button. Then the method is chosen from the Load Demand types pane.

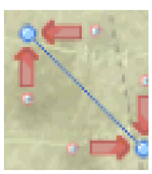
The available load methods are as follows:

Allocation

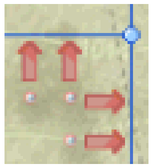
- **Billing Meter Aggregation**—This loading method assigns all meters within a service polygon to the specified loading node for that service polygon.



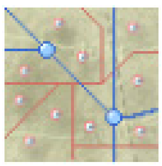
- **Nearest Node**—This loading method assigns customer meter loads to the closest loading junction.



- **Nearest Pipe**—This loading method assigns customer meter loads to the closest pipe, then distributes loads using user-defined criteria.

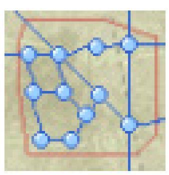


- **Flow Monitoring Distribution**—This loading method asigns loading data from a point load monitoring layer to upstream loading nodes. This method automatically identifies all the upstream manholes up to its adjacent next upstream load monitor, works out the sub-total load contribution of the manhole between the load monitors (the load difference between the monitors) and then equally distributes the effective load to all the contributing manholes.

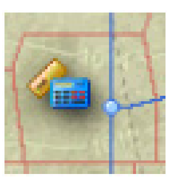


Distribution

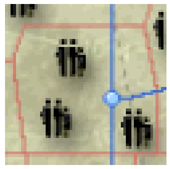
- **Equal Flow Distribution**—This loading method equally divides the total flow contained in a flow boundary polygon and assigns it to the nodes that fall within the flow boundary polygon.



- **Proportional Distribution by Area**—This load method proportionally distributes a lump-sum flow among a number of loading nodes based upon the ratio of total service area to the area of the node's corresponding service polygon.

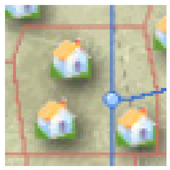


- **Proportional Distribution by Population**—This load method proportionally distributes a lump-sum load among a number of loading nodes based upon the ratio of total population contained within the node's corresponding service polygon.

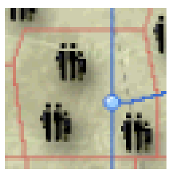


Projection

- **Projection by Land Use**—This method allocates loads based upon the density per land use type of each service polygon.



- **Load Estimation by Population**—This method allocates loads based upon user-defined relationships between load per capita and population data.



In addition to the controls described above, there is also a check box and a menu near the bottom of the dialog box, entitled **Initialize From Previous Run**. If a previously created LoadBuilder template exists in the LoadBuilder Manager display, the settings for this template can be applied to a new LoadBuild of the same type.

Step 2: Input Data

This step will vary according to the load method type that was specified in Step 1, as follows:

- **Billing Meter Aggregation**—Input Data—The following fields require data to be specified:
  - **Service Area Layer**—This field allows you to specify the polygon feature class or shapefile that defines the service area for each demand node.
  - **Manhole ID Field**—This field allows you to specify the source database field that contains identifying label data.

  Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  - **Billing Meter Layer**—This field allows you to specify the point feature class or shapefile that contains the geocoded billing meter data.

- **Load Type Field**—This field allows you to specify the source database field that contains load type data. Load Type is an optional classification that can be used to assign composite loads to nodes, which enables different behaviors, multipliers, and patterns to be applied in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
- **Usage Field**—This field allows you to specify the source database field that contains usage data. The usage field in the source database must contain flow data.
- **Usage Field Units**—This drop-down list allows you to select the unit associated with the usage field value.

• **Nearest Node**—Input Data—The following fields require data to be specified:

- **Node Layer**—This field allows you to specify the feature class or shapefile that contains the nodes that the loads will be assigned to.
- **Node ID Field**—This field allows you to specify the feature class database field that contains the unique identifying label data.

Note: ElementID is the preferred node ID value because it is always unique to any given element.

- **Billing Meter Layer**—This field allows you to specify the feature class or shapefile that contains the geocoded billing meter data.
- **Load Type Field**—This field allows you to specify the source database field that contains load type data. Load Type is an optional classification that can be used to assign composite loads to nodes, which enables different behaviors, multipliers, and patterns to be applied in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
- **Usage Field**—This field allows you to specify the source database field that contains usage data.
- **Usage Field Units**—This drop-down list allows you to select the unit associated with the usage field value.
- **Use Previous Run—**LoadBuilder's most time-consuming calculation when using the Nearest Node strategy is the spatial calculations that are performed to determine proximity between the meter elements and the node elements. When this box is checked, the proximity calculations that were generated from a previous run are used, thereby increasing the overall calculation performance.

• **Nearest Pipe**—Input Data—The following fields require data to be specified:

– **Pipe Layer**—This field allows you to specify the line feature class or shapefile that contains the pipes that will be used to determine meter-to-pipe proximity. Note that the pipes in this layer must connect to the nodes contained in the **Node Layer**.

– **Pipe ID Field**—This field allows you to specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Pipe ID value because it is always unique to any given element.

– **Load Assignment**—This field allows you to specify the method that will be used to distribute the metered loads that are assigned to the nearest pipe to the end nodes of said pipe. Options include:

  - **Distance Weighted**—This method assigns a portion of the total load assigned to a pipe based on the distance between the meter(s) and the nodes at the pipe ends. The closer a meter is to the node at the end of the pipe, the more load will be assigned to it.
  - **Closest Node—**This method assigns the entire total load assigned to the pipe end node that is closest to the meter.
  - **Farthest Node**—This method assigns the entire total load assigned to the pipe end node that is farthest from the meter.
  - **Equal Distribution**—This method assigns an equal portion of the total load assigned to a pipe to each of the pipe's end nodes.

– **Node Layer**—This field allows you to specify the point feature class or shapefile that contains the nodes that will be used to determine node-to-pipe proximity. Note that the nodes in this layer must connect to the pipes contained in the **Pipes Layer**.

– **Node ID Field**—This field allows you to specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

– **Use Previous Run**—LoadBuilder's most time-consuming calculation when using the Nearest Pipe strategy is the spatial calculations that are performed to determine proximity between the meter elements, the pipe elements, and the node elements. When this box is checked, the proximity calculations that were calculated from a previous runs are used, thereby increasing the overall calculation performance.

– **Billing Meter Layer**—This field allows you to specify the point or polyline feature class or shapefile that contains the geocoded billing meter data.

- **Meter Assignment Type**—When a polyline meter layer is selected, this field will be activated. When multiple pipes are associated with (overlapped by) a polyline meter, the option chosen in this field determines the method that will be used to divide the polyline meter load among them. The available options are:
  - **Equal Distribution**—This option will distribute the load equally among the pipes associated with (overlapping) the meter.
  - **Proportional Distribution**—This option will divide the load proportionally according to the ratio of the length of pipe that is associated with (overlapping) the meter to the total length of the meter.
- **Billing Meter ID Field**—Billing Meter ID is used to identify the unique meter. When polylines are used to represent water consumption meters, multiple polylines (multiple records) may designate one actual meter, but each (record in the attribute Table) of the polylines contains the same consumption data with the same billing meter ID.
- **Load Type Field**—This field allows you to specify the source database field that contains load type data. Load Type is an optional classification that can be used to assign composite loads to nodes, which enables different behaviors, multipliers, and patterns to be applied in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
- **Usage Field**—This field allows you to specify the source database field that contains usage data.
- **Usage Field Units**—This drop-down list allows you to select the unit associated with the usage field value.

• **Flow Monitoring Distribution**—Input Data—The following fields require data to be specified:

- **Node Layer**—This field allows you to specify the point feature class or shapefile that contains the nodes that the flow will be assigned to.
- **Node ID Field**—This field allows you to specify the source database field that contains identifying label data.

Note: ElementID is the preferred Node ID value because it is always unique to any given element.

- **Flow Monitoring Layer**—This field allows you to specify the point feature class that contains the flow monitoring meter data.
- **Usage Field**—This field allows you to specify the source database field that contains usage data. The usage field in the source database must contain flow data.

  – **Usage Field Units**—This drop-down list allows you to select the unit associated with the usage field value.

- **Equal Flow Distribution**—Input Data—The following fields require data to be specified:
  – **Manhole Layer**—This field allows you to specify the point feature class or shapefile that contains the manhole data.
  – **Manhole ID Field**—This field allows you to specify the source database field that contains identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  – **Flow Boundary Layer**—This field allows you to specify the polygon feature class or shapefile that contains the flow boundary data.
  – **Load Type Field**—This field allows you to specify the source database field that contains the Load Type data.
  – **Load Type Field Units**—This drop-down list allows you to select the unit associated with the flow field value.

- **Proportional Distribution by Area**—Input Data—The following fields require data to be specified:
  – **Service Area Layer**—This field allows you to specify the polygon feature class or shapefile that defines the service area for each node.
  – **Manhole ID Field**—This field allows you to specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  – **Flow Boundary Layer**—This field allows you to specify the polygon feature class or shapefile that contains the flow boundary data.
  – **Boundary Field**—This field allows you to specify the source database field that contains the boundary label.
  – **Load Type Field**—This field allows you to specify the source database field that contains the load type data.
  – **Load Type Field Units**—This drop-down list allows you to select the unit associated with the Load Type Field value.

- **Proportional Distribution by Population**—Input Data—The following fields require data to be specified:
  – **Service Area Layer**—This field allows you to specify the polygon feature class or shapefile that defines the service area for each node.

- **Manhole ID Field**—This field allows you to specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

- **Flow Boundary Layer**—This field allows you to specify the polygon feature class or shapefile that contains the flow boundary data.
- **Boundary Field**—This field allows you to specify the source database field that contains the boundary label.
- **Load Type Field**—This field allows you to specify the source database field that contains the load data.
- **Load Type Field Units**—This drop-down list allows you to select the unit associated with the load type field value.
- **Population Layer**—This field allows you to specify the polygon feature class or shapefile that contains population data.
- **Population ID Field**—This field allows you to specify the source database field that contains population data.
- **Land Type Field**—This field is optional. It allows you to specify the source database field that contains land use type.

• **Demand Estimation by Land Use**—Input Data—The following fields require data to be specified:

- **Service Area Layer**—This field allows you to specify the polygon feature class or shapefile that defines the service area for each node.
- **Manhole ID Field**—This field allows you to specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

- **Land Use Layer**—This field allows you to specify the polygon feature class or shapefile that contains the land use data.
- **Land Type Field**—This field is optional. It allows you to specify the source database field that contains land use type.
- **Load Densities Per Area**—This table allows you to assign load density values to the various load types contained within your land use layer.

• **Demand Estimation by Population**—Input Data—The following fields require data to be specified:

- **Service Area Layer**—This field allows you to specify the polygon feature class or shapefile that defines the service area for each node.

- **Manhole ID Field**—This field allows you to specify the source database field that contains identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

- **Population Layer**—This field allows you to specify the polygon feature class or shapefile that contains the population data.
- **Population Density Type Field**—This field is optional. It allows you to specify the source database field that contains the population density type data.
- **Population Density Field**—This field allows you to specify the source database field that contains population density data.
- **Load Densities Per Capita**—This table allows you to assign load density values to the various load types contained within your population density layer.

Step 3: Calculation Summary

This step displays the Results Summary pane, which displays the total load, load multiplier, and hydraulic pattern associated with each load type in a tabular format. The number of entries listed will depend on the load build method and data types selected in Step 1. The Results Summary pane contains the following columns:

- **Load Type**—This column contains an entry for each load type contained within the database column specified in step one. (Examples include residential, commercial, industrial, etc.)
- **Consumption**—This column displays the total load associated with each load type entry.
- **Multiplier**—This column displays the multiplier that is applied to each load type entry. Multipliers can be used to account for peak loads, expected future loads, or to reflect unaccounted-for-loads. This field is editable.
- **Pattern**—This column displays the hydraulic pattern associated with each demand type entry. A different pattern can be specified using the menu contained within each cell of this column. New patterns cannot be created from this dialog box; see the Pattern manager help topic for more information regarding the creation of new patterns.

In addition to the functionality provided by the tabular summary pane, the following controls are also available in this step:

- **Global Multiplier**—This field allows you to apply a multiplier to all of the entries contained within the Results Summary Pane. Any changes are automatically reflected in the Total Load text field. Multipliers can be used to account for peak loads, expected future loads, or to reflect unaccounted-for-loads. The Global Multiplier should be used when the conditions relating to these considerations are identical for all usage types and elements.
- **Total Load**—This field displays an updated total of all of the entries contained within the Results Summary Pane, as modified by the local and global multipliers that are in effect.

Step 4: Results Preview

This step displays the calculated results in a tabular format. The table consists of the following information:

- **ElementID**—ElementID is the unique identifying label assigned to all geodatabase elements by the GIS.
- **Label**—Label is the unique identifying label assigned by Bentley SewerGEMS V8*i* Modeler.
- **Load Type**—Load Type is an optional classification that can be used to assign different behaviors, multipliers, and patterns in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
- **Pattern**—Allows you to assign a previously created pattern to each load type in the table.

Step 5: Completing the LoadBuilder Wizard

In this step, the load build template is given a label and the results are exported to an existing or new load alternative. This step contains the following controls:

- **Label**—This field allows a unique label to be assigned to the load build template.
- **Override an Existing Alternative**—Choosing this option will cause the calculated loads to overwrite the loads contained within the existing load alternative that is selected.
- **Append to an Existing Alternative**—Choosing this option will cause the calculated loads to be appended to the loads contained within the existing load alternative that is selected. Loads within the existing alternative that are assigned to a specific node will not be overwritten by newly generated loads assigned to the same node; the new loads will simply be added to them.

- **New Alternative**—Choosing this option will cause the calculated loads to be applied to a new load alternative. The text field next to this button lets you enter a label for the new load alternative. The Parent Alternative field will only be active when this option is selected.
- **Export to SewerCAD—**When this option is chosen, a delimited text file containing flow data will be created that can then be imported in SewerCAD.

LoadBuilder Run Summary

The LoadBuilder Run Summary dialog box details important statistics about the results of a completed LoadBuilder run, including the number of successfully added loads, file information, and informational and/or warning messages.

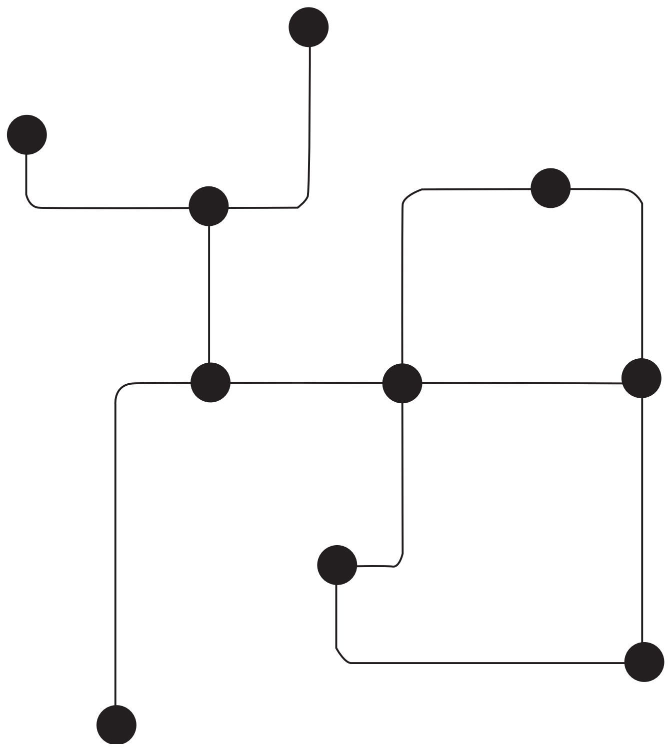
Generating Thiessen Polygons

A Thiessen polygon is a Voronoi Diagram that is also referred to as the Dirichlet Tessellation. Given a set of points, it defines a region around each point. A Thiessen polygon divides a plane such that each point is enclosed within a polygon and assigns the area to a point in the point set. Any location within a particular Thiessen polygon is nearer to that polygon's point than to any other point. Mathematically, a Thiessen is constructed by intersecting perpendicular bisector lines between all points.

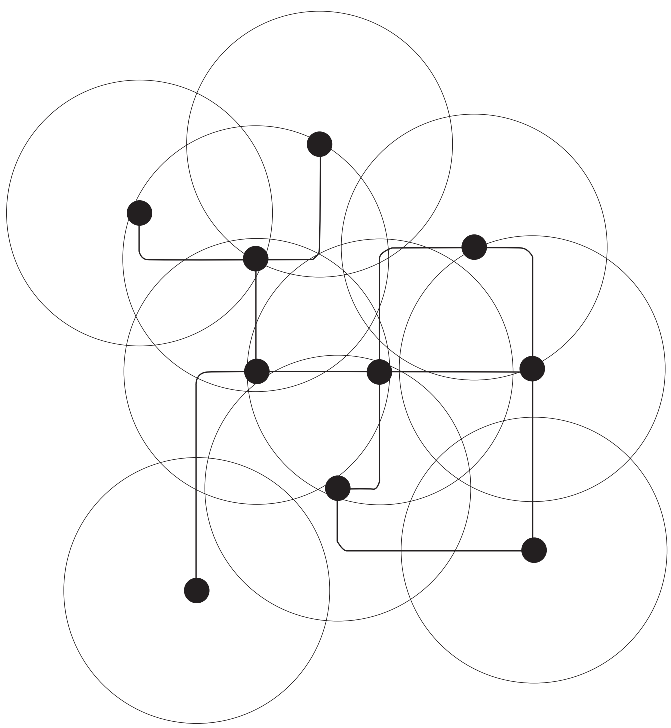
Thiessen polygon has many applications in different location-related disciplines such as business planning, community services, transportation and hydraulic/hydrological modeling. For water distribution modeling, the Thiessen Polygon generation utility was developed to quickly and easily define the service areas of demand nodes. Since each customer within a Thiessen polygon for a junction is nearer to that node than any others, it is assumed that the customers within a particular Thiessen polygon are supplied by the same demand node.

The following diagrams illustrate how Thiessen polygons would be generated manually. The Thiessen polygon generator does not use this method, although the results produced by the generator are consistent with those that would be obtained using this method.

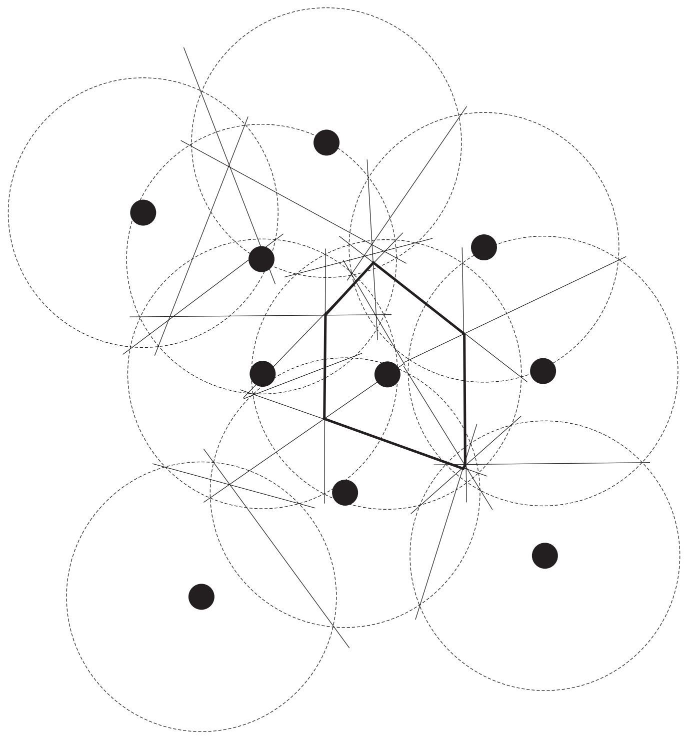
The first diagram shows a simple pipe and junction network.



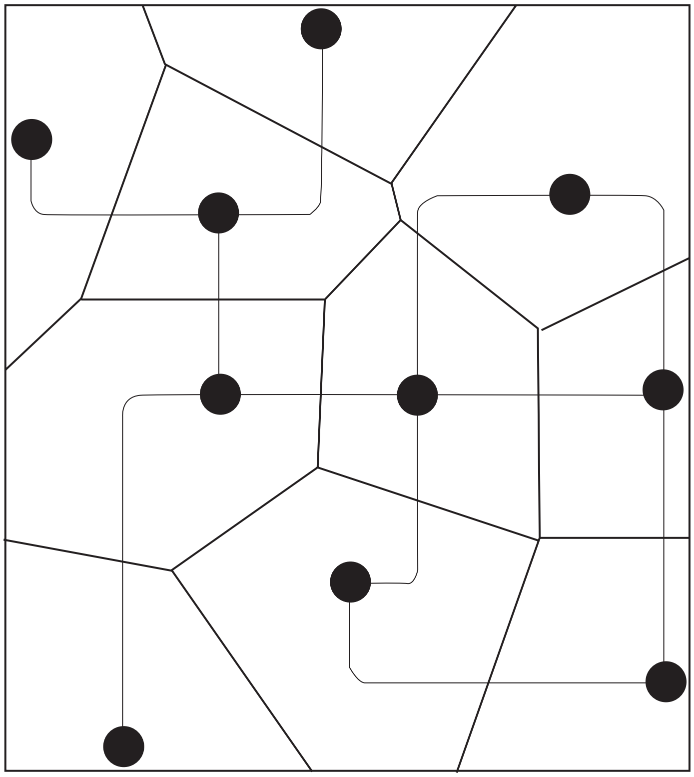
In the second diagram, the circles are drawn around each junction.



In the third diagram, bisector lines are added by drawing a line where the circles interjoin.



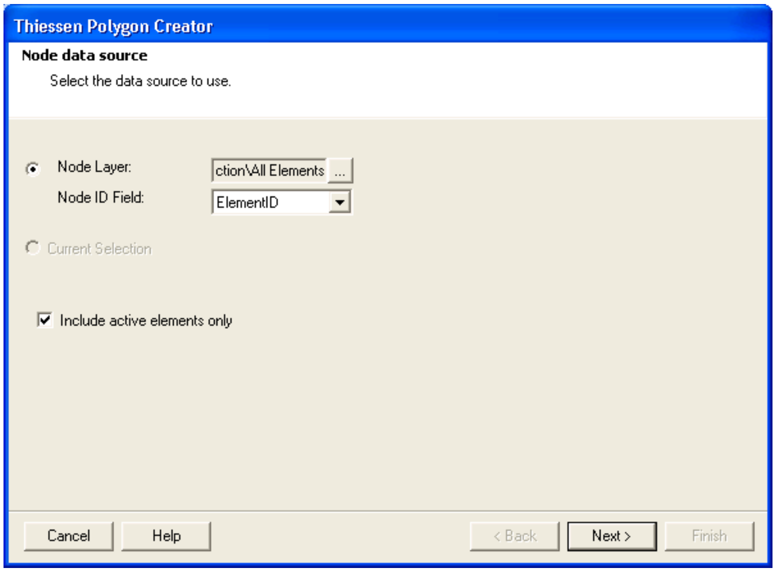
In the final diagram, the network is overlaid with the polygons that are created by connecting the bisector lines.



Thiessen Polygon Input Dialog Box

The Thiessen Polygon Creator allows you to quickly create polygon layers for use with the LoadBuilder demand allocation module. This utility creates polygon layers that can be used as service area layers for the following LoadBuilder loading strategies:

- Billing Meter Aggregation
- Proportional Distribution By Area
- Proportional Distribution By Population
- Projection by Land Use
- Load Estimation by Population.



The Thiessen Polygon Creator dialog box consists of the following steps:

Step 1: Node Data Source

- **Node Data Source**—Select the data source to use.
  - **Node Layer**—This lists the valid point feature classes and shapefiles that Thiessen Polygon Creator can use.
  - **Current Selection**—Click if the current feature data set contains a previously created selection set.
  - **Include active elements only**—Click to activate.
  - **Selection**—This option allows you to create a selection on the fly for use with the Thiessen Polygon Creator. To use this option, use the ArcMap **Select Features** tool to select the point features that you want before opening the Thiessen Polygon Creator.

Step 2: Boundary Layer

- **Buffering Percentage**—This percentage value is used for calculating the boundary for a collection of points. In order to make the buffer boundary big enough to cover all the points, the boundary is enlarged based upon the value entered in this field as it relates to the percentage of the area enclosed by drawing a polygon that connects the outermost nodes of the model.
- **Polygon Boundary Layer**—Select the boundary polygon feature class or shape-file, if one has already been created. A boundary is specified so that the outermost polygons do not extend to infinity.

Note: For more information about boundary layers, see “Creating Boundary Polygon Feature Classes”**.**

Step 3: Output Layer

- **Output File**—Specify the name of the shapefile that will be created.

Note: The Thiessen Polygon Creator is flexible enough to generate Thiessen polygons for unusual boundary shapes, such as borders with cutouts or holes that Thiessen polygons should not be created inside. To accomplish this, the boundary polygon must be created as one complex (multi-part) polygon. For more information about creating boundary polygon feature classes, see your ArcGIS documentation.

Creating Boundary Polygon Feature Classes

The Thiessen polygon generator requires a boundary to be specified around the area in which Thiessen Polygons will be created. This is to prevent the outside edge of the polygons along the perimeter of this area from extending to infinity. The generator can automatically create a boundary using the **Buffering Point Area Percentage** value, or it can use a previously created polygon feature class as the boundary.

A border polygon feature class can be created in ArcCatalog, and edited in ArcMap.

To create a border feature class, you will need a Bentley SewerGEMS V8*i* model that has had at least one scenario published as an ESRI feature dataset. Then, follow these steps:

1. In the directory structure pane of ArcCatalog, right-click the Bentley SewerGEMS V8*i* feature dataset and select **New**...**Feature Class**.
2. A dialog box will open, prompting you to name the new feature class. Enter a name and click **Next**.
3. In the second step, you are prompted to select the database storage configuration. Do so, and click **Next**.
4. In the third step, click the **Shape** cell under the **Field Name** column, and ensure that the **Geometry Type** is **Polygon**. Click **Finish**.
5. In ArcMap, click the **Add Data** button and select your Bentley Sewer-GEMS V8*i* feature dataset. 
6. Click the **Editor** button and select **Start Editing**. Ensure that the border feature class is selected in the **Target** drop-down list.
7. Draw a polygon around the point features (generally manholes) that you wish to be used to generate the polygons. When you are finished drawing the polygon, click **Editor...Stop Editing**. Choose **Yes** when prompted to save your edits.

The polygon feature class you just created can now be used as the boundary during Thiessen polygon generation. For more information about creating and editing feature classes, see your ArcGIS documentation.

ModelBuilder

For detailed information on how to use ModelBuilder, see "Using Modelbuilder" on page 5-171.

Using GeoTables

A GeoTable is a flexible table definition provided by Bentley SewerGEMS V8*i*. Bentley SewerGEMS V8*i* creates feature classes with a very simple schema. The schema consists solely of the Geometry, the Bentley SewerGEMS V8*i* ID and Bentley SewerGEMS V8*i* feature type. what Bentley SewerGEMS V8*i* provides is a dynamic join of this data to our trademarked GeoTable. The join is then managed so that it will be automatically updated on a change to the GeoTable definition for each element type.

GeoTables allow for a dynamic view on the data. The underlying data will represent the data for the current scenario, the current timestep and the unit definition of the GeoTable. By using these GeoTables, Bentley SewerGEMS V8*i* provides ultimate flexibility for using the viewing and rendering tools provided by the ArcMap environment.

Note that the GeoTable settings are not project specific, but are stored on your local machine - any changes you make will carry across all projects. This means that if you have ArcMap display settings based on attributes contained in customized GeoTables, you will have to copy the AttributeFlexTables.xml file (located in the C:\Documents and Settings\All Users\Application Data\Haestad\Bentley SewerGEMS V8*i*\1 folder) for these display settings to work on another computer.

Using GeoTables, you can:

- Apply ArcMap symbology definitions to map elements based on Bentley SewerGEMS V8*i* data
- Use the ArcMap Select By Attributes command to select map elements based on Bentley SewerGEMS V8*i* data
- Generate ArcMap reports and graphs that include Bentley SewerGEMS V8*i* data

To Edit a GeoTable:

1. In the FlexTable Manager list pane, expand the GeoTables node if necessary. Double-click the GeoTable for the desired element.
2. By default, only the ID, Label, and Notes data is included in the GeoTable. To add attributes, click the Edit button.
3. In the Table setup dialog that appears, move attributes from the Available Columns list to the Selected columns list to include them in the GeoTable. This can be accomplished by double-clicking an attribute in the list, or by highlighting attributes and using the arrow buttons (a single arrow button moves the highlighted attribute to the other list; a double arrow moves all of them).
4. When all of the desired attributes have been moved to the selected columns, click OK.

Chapter

12 Features of the MicroStation Version

Bentley SewerGEMS V8*i* features support for MicroStation integration. You run Bentley SewerGEMS V8*i* in both MicroStation and stand-alone mode.

The MicroStation functionality has been implemented in a way that is the same as the Bentley SewerGEMS V8*i* base product. Once you become familiar with the stand-alone mode, you will not have any difficulty using the product in MicroStation mode.

In MicroStation mode, you will have access to the full range of functionality available in the MicroStation design and drafting environment. The standard environment is extended and enhanced by using MicroStation's MDL (MicroStation Development Language) client layer that lets you create, view, and edit the native Bentley SewerGEMS V8*i* network model while in MicroStation.

MDL is a complete development environment that lets applications take full advantage of the power of MicroStation and MicroStation-based vertical applications. MDL can be used to develop simple utilities, customized commands or sophisticated commercial applications for vertical markets.

Some of the advantages of working in MicroStation mode include:

- Lay out network links and structures in fully-scaled mode in the same design and drafting environment that you use to develop your engineering plans.
- You will have access to any other third party applications that you currently use, along with any custom MDL applications.
- Use native MicroStation insertion snaps to precisely position Bentley SewerGEMS V8*i* elements with respect to other entities in the MicroStation drawing.
- Use native MicroStation commands on Bentley SewerGEMS V8*i* model entities with automatic update and synchronization with the model database.
- Control destination levels for model elements and associated label text and annotation, giving you control over styles, line types, and visibility of model elements.

Note: Bentley SewerGEMS V8*i* supports MicroStation 2004 Geographics Edition only.

Additional features of the MicroStation version includes:

- "MicroStation Environment" on page 12-700
- "MicroStation Project Files" on page 12-701
- "Bentley SewerGEMS V8i Element Properties" on page 12-701
- "Working with Elements" on page 12-703
- "MicroStation Commands" on page 12-705
- "Undo/Redo" on page 12-706
- "Special Considerations" on page 12-707

MicroStation Environment

The MicroStation environment includes:

- "MicroStation Mode Graphical Layout" on page 12-700

MicroStation Mode Graphical Layout

In MicroStation mode, our products provide a set of extended options and functionality beyond those available in stand-alone mode. This additional functionality provides enhanced control over general application settings and options and extends the command set, giving you control over the display of model elements within MicroStation.

Key differences between MicroStation and stand-alone mode include:

- Full element symbol editing functionality is available through the use of custom cells. All elements and graphical decorations (flow arrows, control indicators, etc.) are contained within a SewerGEMS V8*i* .cel file.

You can control the appearance and destination of all model elements using the Element Levels command under the View menu. For example, you can assign a specific level for all outlets, as well as assign the label and annotation text style to be applied.

Note: Any MicroStation tool that deletes the target element (such as Trim and IntelliTrim) will also remove the connection of that element to SewerGEMS V8*i*. After the SewerGEMS V8*i* connection is removed, the element is no longer a valid CSD link and will not show properties on the property grid. Storm.

MicroStation Project Files

When using Bentley SewerGEMS V8*i* in MicroStation mode, there are three files that fundamentally define a Bentley SewerGEMS V8*i* model project:

- **Drawing File (.DGN)**—The MicroStation drawing file contains the elements that define the model, in addition to the planimetric base drawing information that serves as the model background.
- **Model File (.CSD)**—The model file contains model data specific to SewerGEMS V8*i*, including project option settings, color-coding and annotation settings, etc. Note that the MicroStation .dgn that is associated with a particular model may not bave the same filename as the model's .csd file.
- **Database File (.MDB)**—The model database file that contains all of the input and output data for the model. Note that the MicroStation .dgn that is associated with a particular model may not bave the same filename as the model's .mdb file.

To send the model to another user, all three files are required.

It is important to understand that archiving the drawing file is not sufficient to reproduce the model. You must also preserve the associated .CSD and .MDB files.

Bentley SewerGEMS V8*i* Element Properties

Bentley SewerGEMS V8*i* element properties includes:

- "Element Properties" on page 12-702
- "Levels" on page 12-702
- "Text Styles" on page 12-703

Element Properties

When working in the MicroStation mode, this feature will display a dialog box containing fields for the currently selected element's associated properties. To modify an attribute, click each associated grid cell.

You can also review or modify MicroStation drawing information about an element(s), such as its type, attributes, and geometry, by using the Element Information dialog. To access the Element Information dialog, click the Element Information button or click the Element menu and select the Information command.

Levels

To control display of elements in the selected levels, use the Level Display dialog box. To access the Level Display dialog, click the Settings menu and select the Level > Display command.

If you want to freeze elements in levels, select Global Freeze from the View Display menu in the Level Display dialog.

You can create new Levels in the Level Manager. To access the Level Manager, click the Settings menu and select the Level > Manager command.

To control the display of levels, use level filters. Within MicroStation, you can also create, edit, and save layer filters to DWG files in the Level Manager. To access the Level Manager, click the Settings menu and select the Level > Manager command. Layer filters are loaded when a DWG file is opened, and changes are written back when the file is saved. To create and edit Level Filters,

Element Levels Dialog

This dialog allows you to assign newly created elements and their associated annotations to specific MicroStation levels.

To assign a level, use the pulldown menu next to an element type (under the Element Level column heading) to choose the desired level for that element. You can choose a seperate level for each element and for each element's associated annotation.

You cannot create new levels from this dialog; to create new levels use the MicroStation Level Manager. To access the Level Manager, click the Settings menu and select the Level > Manager command.

Text Styles

You can view, edit, and create Text Style settings in MicroStation mode by clicking the Element menu and selecting the Text Styles command to open the Text Styles dialog.

Working with Elements

Working with elements includes:

- "Edit Elements" on page 12-704
- "Deleting Elements" on page 12-704
- "Modifying Elements" on page 12-704

Edit Elements

Elements can be edited in one of two ways in MicroStation mode:

Element Properties Dialog: To access the Element Properties dialog, click the SewerGEMS V8*i* View menu and select the Properties command.

FlexTables: To access the FlexTables dialog, click the SewerGEMS V8*i* View menu and select the FlexTables command.

Deleting Elements

In MicroStation mode, you can delete elements by clicking on them using the Delete Element tool, or by highlighting the element to be deleted and clicking your keyboard's Delete key.

Note: Any MicroStation tool that deletes the target element (such as Trim and IntelliTrim) will also remove the connection of that element to SewerGEMS V8*i*. After the SewerGEMS V8*i* connection is removed, the element is no longer a valid CSD link and will not show properties on the property grid. Storm.

Modifying Elements

In MicroStation mode, these commands are selected from the shift-right-click shortcut menu (hold down the Shift key while right-clicking). They are used for scaling and rotating model entities.

Change Pipe Widths

In MicroStation mode, you can change the line width through the Element Information dialog. To access the Element Information dialog, click the Element menu and select the Information command. To change the width of a pipe, select it and open the Element Information dialog. Then change the value in the Weight pulldown menu

Edit Elements

In MicroStation mode, this menu command is used to open a spreadsheet FlexTable editor or a selection of one or more network figures. You are prompted to select figures on which to build a table.

Working with Elements Using MicroStation Commands

Working with elements using MicroStation commands includes:

"Bentley SewerGEMS V8i Custom MicroStation Entities" on page 12-705

"MicroStation Commands" on page 12-705

"Moving Elements" on page 12-705

"Moving Element Labels" on page 12-705

"Snap Menu" on page 12-706

Bentley SewerGEMS V8*i* Custom MicroStation Entities

The primary MicroStation-based Bentley SewerGEMS V8*i* element entities—pipes, channels, gutter links, manholes, catch basins, outfalls, pond outlet structures, and cross section nodes—are all implemented using native MicroStation elements. These elements have feature linkages to define them as SewerGEMS V8*i* objects.

This means that you can perform standard MicroStation commands (see “MicroStation Commands” on page 12-705) as you normally would, and the model database will be updated automatically to reflect these changes.

It also means that the model will enforce the integrity of the network topological state. Therefore, if you delete a nodal element such as a junction, its connecting pipes will also be deleted since their connecting nodes topologically define model pipes.

Using MDL technology ensures the database will be adjusted and maintained during Undo and Redo transactions.

MicroStation Commands

When running in MicroStation mode, Haestad Methods products make use of all the advantages that MicroStation has, such as plotting capabilities and snap features. Additionally, MicroStation commands can be used as you would with any design project. For example, our products’ elements and annotation can be manipulated using common MicroStation commands.

Related Topics

Moving Elements

When using MicroStation mode, the MicroStation commands Move, Scale, Rotate, Mirror, and Array can be used to move elements.

To move a node, execute the MicroStation command by either typing it at the command prompt or selecting it. Follow the MicroStation prompts, and the node and its associated label will move together. The connecting pipes will shrink or stretch depending on the new location of the node.

Related Topics

Moving Element Labels

When using MicroStation mode, the MicroStation commands Move, Scale, Rotate, Mirror, and Array can be used to move element text labels.

To move an element text label separately from the element, click the element label you wish to move. The grips will appear for the label. Execute the MicroStation command either by typing it at the command prompt, by selecting it from the tool palette, or by selecting it from the right-click menu. Follow the MicroStation prompt, and the label will be moved without the element.

Snap Menu

When using MicroStation mode, you can enable the Snaps button bar by clicking the Settings menu and selecting the Snaps > Button Bar command. See the MicroStation documentation for more information about using snaps.

Polygon Element Visibility

By default, polygon elements are sent to the back of the level order when they are drawn. If the level order is modified, polygon elements can interfere with the visibility of other elements. This can be remedied using the MicroStation Level Manager (Click the Settings menua and select the Level > Manager command.).

To access the MicroStation Level Manager, click the Settings menu and select the Level > Manager command.

Undo/Redo

If you use the native MicroStation undo, you are limited to a single redo level.

If you undo using the MicroStation undo/redo and you restore Bentley SewerGEMS V8*i* elements that have been previously deleted, some model state attributes such as diameters or elevations may be lost, even though the locational and topological state is fully consistent. This will only happen in situations where the Bentley SewerGEMS V8*i* command history has been deleted. In such cases, you will be warned to check your data carefully.

In MicroStation mode, you have two types of undo/redo available to you. From the Edit menu, you have access to Bentley SewerGEMS V8*i* undo and redo. Alternatively, you can perform the native MicroStation undo and redo by typing at the MicroStation command line. The implementations of the two different operation types are quite distinct.

The menu-based undo and redo commands operate exclusively on Bentley SewerGEMS V8*i* elements by invoking the commands directly on the model server. The main advantage of using the specialized command is that you will have unlimited undo and redo levels. This is an important difference, since in layout or editing it is quite useful to be able to safely undo and redo an arbitrary number of transactions.

Whenever you use a native MicroStation undo, the server model will be notified when any Bentley SewerGEMS V8*i* entities are affected by the operation. Bentley SewerGEMS V8*i* will then synchronize the model to the drawing state. Wherever possible, the model will seek to map the undo/redo onto the model server's managed command history. If the drawing's state is not consistent with any pending undo or redo transactions held by the server, Bentley SewerGEMS V8*i* will delete the command history. In this case, the model will synchronize the drawing and server models.

Special Considerations

Special considerations include:

- "Import Bentley SewerGEMS V8i" on page 12-707
- "Annotation Display" on page 12-707
- "Use SewerGEMS V8i Z Order Command" on page 12-707

Import Bentley SewerGEMS V8i

When running Bentley SewerGEMS V8*i* in MicroStation mode, this command imports a selected Bentley SewerGEMS V8*i* data (.CSD) file for use in the current drawing. The new project file will now correspond to the drawing name, such as, CurrentDrawingName.CSD. Whenever you save changes to the network model through Bentley SewerGEMS V8*i*, the associated .CSD data file is updated and can be loaded into Bentley SewerGEMS V8*i* 4.0 or higher.

Warning! A SewerGEMS V8*i* Project can only be imported to a new, empty MicroStation design model.

Annotation Display

Some fonts do not correctly display the full range of characters used by SewerGEMS V8*i*'s annotation feature because of a limited character set. If you are having problems with certain characters displaying improperly or not at all, try using another font.

Use SewerGEMS V8i Z Order Command

When this control is toggled on, SewerGEMS V8*i* will parse the elements when the screen is redrawn. It takes significant time to do this parsing, so for larger models performance can be improved by toggling this control off

Chapter

Working in AutoCAD Mode 13

Caution: If you previously installed Bentley ProjectWise and turned on AutoCAD integration, you must add the following key to your system registry using the Windows Registry Editor. Before you edit the registry, make a backup copy.

HKEY_LOCAL_MACHINE\SOFTWARE\Bentley\ProjectWise iDesktop Integration\XX.XX\Configuration\AutoCAD"

String value name: DoNotChangeCommands

Value: 'On'

To access the Registry Editor, click Start > Run, then type regedit. Using the Registry Editor incorrectly can cause serious, system-wide problems that may require you to re-install Windows to correct them. Always make a backup copy of the system registry before modifying it.

Bentley SewerGEMS V8*i* features support for AutoCAD integration. You can determine if you have purchased AutoCAD functionality for your license of Bentley SewerGEMS V8*i* by using the **Help > About** menu option. Click the **Registration** button to view the feature options that have been purchased with your application license. If AutoCAD support is enabled, then you will be able to run your Bentley SewerGEMS V8*i* application in both AutoCAD and stand-alone mode.

The AutoCAD functionality has been implemented in a way that is the same as the SewerGEMS V8*i* base product. Once you become familiar with the stand-alone mode, you will not have any difficulty using the product in AutoCAD mode.

Some of the advantages of working in AutoCAD mode include:

- Layout network links and structures in fully-scaled mode in the same design and drafting environment that you use to develop your engineering plans. You will have access to any other third party applications that you currently use, along with any custom LISP, ARX, or VBA applications that you have developed.

- Use native AutoCAD insertion snaps to precisely position Bentley SewerGEMS V8*i* elements with respect to other entities in the AutoCAD drawing.
- Use native AutoCAD commands such as ERASE, MOVE, and ROTATE on Bentley SewerGEMS V8*i* model entities with automatic update and synchronization with the model database.
- Control destination layers for model elements and associated label text and annotation, giving you control over styles, line types, and visibility of model elements.

- "AutoCAD Project Files" on page 13-712

The AutoCAD Workspace

In AutoCAD mode, you will have access to the full range of functionality available in the AutoCAD design and drafting environment. The standard environment is extended and enhanced by an AutoCAD ObjectARX Bentley SewerGEMS V8*i* client layer that lets you create, view, and edit the native Bentley SewerGEMS V8*i* network model while in AutoCAD.

Click one of the following links to learn more about Bentley SewerGEMS V8*i* AutoCAD environment:

- "AutoCAD Integration with SewerGEMS V8i" on page 13-710

AutoCAD Integration with SewerGEMS V8i

When you install SewerGEMS after you install AutoCAD, integration between the two is automatically configured.

If you install AutoCAD after you install SewerGEMS, you must manually integrate the two by selecting **Start > All Programs > Haestad Methods >SewerGEMS > Integrate SewerGEMS with AutoCAD-ArcGIS**. The integration utility runs automatically. You can then run SewerGEMS in AutoCAD mode.

The Integrate SewerGEMS with AutoCAD-ArcGIS command can also be used to fix problems with the AutoCAD configuration file. For example, if you have SewerGEMS installed on the same system as Bentley SewerGEMS V8*i* and you uninstall or reinstall SewerGEMS, the AutoCAD configuration file becomes unusable. To fix this problem, you can delete the configuration file then run the Integrate SewerGEMS with AutoCAD-ArcGIS command.

AutoCAD Mode Graphical Layout

In AutoCAD mode, our products provide a set of extended options and functionality beyond those available in stand-alone mode. This additional functionality provides enhanced control over general application settings and options and extends the command set, giving you control over the display of model elements within AutoCAD.

Note: In AutoCAD, you must hold down the mouse button to keep the submenu open while selecting an element from the layout toolbar. Alternate layout methods include using the right-click menu to select elements or using the command line.

Menus

In AutoCAD mode, in addition to AutoCAD's menus, the following Bentley Sewer-GEMS V8*i* menus are available:

- Analysis
- View
- Tools
- Report

In addition, Bentley SewerGEMS V8*i* adds its own Help menu commands to AutoCAD's Help menu.

The Bentley SewerGEMS V8*i* menu commands work the same way in AutoCAD and the Stand-Alone Editor. For complete descriptions of Bentley SewerGEMS V8*i* menu commands, see "Menus" on page 2-11.

Toolbars

In AutoCAD mode, in addition to AutoCAD's toolbars, the following Bentley Sewer-GEMS V8*i* toolbars are available:

- Layout
- View
- Compute
- Scenarios
- Analysis
- Links

The Bentley SewerGEMS V8*i* toolbars work the same way in AutoCAD and the Stand-Alone Editor. For complete descriptions of Bentley SewerGEMS V8*i* toolbars, see "Toolbars" on page 2-25.

Drawing Setup

When working in the AutoCAD mode, you may work with our products in many different AutoCAD scales and settings. However, Haestad Methods product elements can only be created and edited in model space.

Symbol Visibility

Note: In AutoCAD, it is possible to delete element label text using the ERASE command. You should not use ERASE to control visibility of labels. If you desire to control the visibility of a selected group of element labels, you should move them to another layer that can be frozen or turned off.

In AutoCAD mode, you can control display of element labels using the check box in the Drawing Options dialog box.

AutoCAD Project Files

When using Bentley SewerGEMS V8*i* in AutoCAD mode, there are three files that fundamentally define a Bentley SewerGEMS V8*i* model project:

- **Drawing File (.dwg)**—The AutoCAD drawing file contains the custom entities that define the model, in addition to the planimetric base drawing information that serves as the model background.
- **Model File (.swg)**—The native Bentley SewerGEMS V8*i* model database file that contains all the element properties, along with other important model data. Bentley SewerGEMS V8*i* .swg files can be loaded and run using the Stand-Alone Editor. These files may be copied and sent to other Bentley SewerGEMS V8*i* users who are interested in running your project. This is the most important file for the Bentley SewerGEMS V8*i* model.
- **SewerCAD Exchange Database (.srw.mdb)**—The intermediate format for SewerCAD project files. When you import a SewerCAD file into Bentley SewerGEMS V8*i*, you first export it from SewerCAD into this format, then import the .srw.mdb file into Bentley SewerGEMS V8*i*. Note that this works the same in the Stand-Alone Editor and in AutoCAD. For more information on importing SewerCAD files, see "Importing Data from SewerCAD" on page 4-163.

The three files have the same base name. It is important to understand that archiving the drawing file is not sufficient to reproduce the model. You must also preserve the associated .swg and srw.mdb file.

Since the .swg file can be run and modified separately from the .dwg file using the Stand-Alone Editor, it is quite possible for the two files to get out of sync. Should you ever modify the model in the Stand-Alone Editor and then later load the AutoCAD .dwg file, the Bentley SewerGEMS V8*i* program compares file dates, and automatically use the built-in AutoCAD synchronization routine.

Click one of the following links to learn more about AutoCAD project files and Bentley SewerGEMS V8*i*:

- "Drawing Synchronization" on page 13-713
- "Saving the Drawing as Drawing*.dwg" on page 13-714

Drawing Synchronization

Whenever you open a Bentley SewerGEMS V8*i*-based drawing file in AutoCAD, the Bentley SewerGEMS V8*i* model server will start. The first thing that the application will do is load the associated Bentley SewerGEMS V8*i* model (.SWG) file. If the time stamps of the drawing and model file are different, Bentley SewerGEMS V8*i* will automatically perform a synchronization. This protects against corruption that might otherwise occur from separately editing the Bentley SewerGEMS V8*i* model file in stand-alone mode, or editing proxy elements at an AutoCAD station where the Bentley SewerGEMS V8*i* application is not loaded.

The synchronization check will occur in two stages:

- First, Bentley SewerGEMS V8*i* will compare the drawing model elements with those in the server model. Any differences will be listed. Bentley SewerGEMS V8*i* enforces network topological consistency between the server and the drawing state. If model elements have been deleted or added in the .SWG file during a SewerGEMS V8*i* session, or if proxy elements have been deleted, Bentley SewerGEMS V8*i* will force the drawing to be consistent with the native database by restoring or removing any missing or excess drawing custom entities.
- After network topology has been synchronized, Bentley SewerGEMS V8*i* will compare other model and drawing states such as location, labels, and flow directions.

You can run the Synchronization check at any time using the following command:

```
SWRGSYNCHRONIZE
```

Or by selecting Tools > Database Utilities > Synchronize Drawing.

Saving the Drawing as Drawing.dwg*

Note: If this situation inadvertently occurs (save on quit for example), restart AutoCAD, use the Open command to open the Drawing*.dwg file from its saved location, and use the Save As command to save the drawing and model data to a different name.

AutoCAD uses Drawing*.dwg as its default drawing name. Saving your drawing as the default AutoCAD drawing name (for instance Drawing1.dwg) should be avoided, as it makes overwriting model data very likely. When you first start AutoCAD, the new empty drawing is titled Drawing*.dwg, regardless of whether one exists in the default directory. Since our modeling products create model databases associated with the AutoCAD drawing, the use of Drawing*.dwg as the saved name puts you at risk of causing synchronization problems between the AutoCAD drawing and the modeling files.

Working with Elements Using AutoCAD Commands

This section describes how to work with elements using AutoCAD commands, including:

SewerGEMS Custom AutoCAD Entities

The primary AutoCAD-based SewerGEMS element entities—conduits, channels, gutters, pressure pipes, manholes, catch basins, outfalls, ponds, catchments, pond outlet structures, cross section nodes, pressure junctions, junction chambers, wet wells, and pumps—are all implemented using ObjectARX custom objects. Thus, they are vested with a specialized model awareness that ensures that any editing actions you perform will result in an appropriate update of the model database.

This means that you can perform standard AutoCAD commands (see "AutoCAD Commands" on page 13-715) as you normally would, and the model database will be updated automatically to reflect these changes.

It also means that the model will enforce the integrity of the network topological state. Therefore, if you delete a nodal element such as a junction, its connecting pipes will also be deleted since their connecting nodes topologically define model pipes.

Using ObjectARX technology ensures the database will be adjusted and maintained during Undo and Redo transactions.

Related Topics

- "AutoCAD Commands" on page 13-715
- "Explode Elements" on page 13-715
- "Moving Elements" on page 13-716
- "Moving Element Labels" on page 13-716
- "Snap Menu" on page 13-716

AutoCAD Commands

When running in AutoCAD mode, Haestad Methods products make use of all the advantages that AutoCAD has, such as plotting capabilities and snap features. Additionally, AutoCAD commands can be used as you would with any design project. For example, our products' elements and annotation can be manipulated using common AutoCAD commands.

- "SewerGEMS Custom AutoCAD Entities" on page 13-714

Explode Elements

In AutoCAD mode, running the AutoCAD Explode command will transform all custom entities into equivalent AutoCAD native entities. When a custom entity is exploded, all associated database information is lost. Be certain to save the exploded drawing under a separate filename.

Use Explode to render a drawing for finalizing exhibits and publishing maps of the model network. You can also deliver exploded drawings to clients or other individuals who do not own a Bentley Systems Product license, since a fully exploded drawing will not be comprised of any ObjectARX proxy objects. For more information, see "Working with Proxies" on page 13-718.

- "SewerGEMS Custom AutoCAD Entities" on page 13-714

Moving Elements

When using AutoCAD mode, the AutoCAD commands Move, Scale, Rotate, Mirror, and Array can be used to move elements.

To move a node, execute the AutoCAD command by either typing it at the command prompt or selecting it. Follow the AutoCAD prompts, and the node and its associated label will move together. The connecting pipes will shrink or stretch depending on the new location of the node.

- "SewerGEMS Custom AutoCAD Entities" on page 13-714

Moving Element Labels

When using AutoCAD mode, the AutoCAD commands Move, Scale, Rotate, Mirror, and Array can be used to move element text labels.

To move an element text label separately from the element, click the element label you wish to move. The grips will appear for the label. Execute the AutoCAD command either by typing it at the command prompt, by selecting it from the tool palette, or by selecting it from the right-click menu. Follow the AutoCAD prompt, and the label will be moved without the element.

- "SewerGEMS Custom AutoCAD Entities" on page 13-714

Snap Menu

When using AutoCAD mode, the Snap menu is a standard AutoCAD menu that provides options for picking an exact location of an object. See the Autodesk AutoCAD documentation for more information.

- "SewerGEMS Custom AutoCAD Entities" on page 13-714

Polygon Element Visibility

By default, polygon elements are sent to the back of the draw order when they are drawn. If the draw order is modified, polygon elements can interfere with the visibility of other elements. This can be remedied using the AutoCAD Draw Order toolbar.

To access the AutoCAD Draw Order toolbar, right-click on the AutoCAD toolbar and click the Draw Order entry in the list of available toolbars.

By default, polygon elements are filled. You can make them unfilled (just borders visible) using the AutoCAD FILL command. After turning fill mode OFF, you must REGEN to redraw the polygons.

Undo/Redo

Note: If you use the native AutoCAD undo, you are limited to a single redo level. The Bentley SewerGEMS V8*i* undo/redo is faster than the native AutoCAD undo/redo. If you are rolling back Bentley SewerGEMS V8*i* model edits, it is recommended that you use the menu-based Bentley SewerGEMS V8*i* undo/redo.

If you undo using the AutoCAD undo/redo and you restore Bentley SewerGEMS V8*i* elements that have been previously deleted, morphed, or split, some model state attributes such as diameters or elevations may be lost, even though the locational and topological state is fully consistent. This will only happen in situations where the Bentley SewerGEMS V8*i* command history has been deleted. In such cases, you will be warned to check your data carefully.

In AutoCAD mode, you have two types of undo/redo available to you. From the Edit menu, you have access to Bentley SewerGEMS V8*i* undo and redo. Alternatively, you can perform the native AutoCAD undo and redo by typing at the AutoCAD command line. The implementations of the two different operation types are quite distinct.

The menu-based undo and redo commands operate exclusively on Bentley SewerGEMS V8*i* elements by invoking the commands directly on the model server. The main advantage of using the specialized command is that you will have unlimited undo and redo levels. This is an important difference, since in layout or editing it is quite useful to be able to safely undo and redo an arbitrary number of transactions.

Whenever you use a native AutoCAD undo, the server model will be notified when any Bentley SewerGEMS V8*i* entities are affected by the operation. Bentley SewerGEMS V8*i* will then synchronize the model to the drawing state. Wherever possible, the model will seek to map the undo/redo onto the model server's managed command history. If the drawing's state is not consistent with any pending undo or redo transactions held by the server, Bentley SewerGEMS V8*i* will delete the command history. In this case, the model will synchronize the drawing and server models.

Special Considerations

There are special considerations to remember when you perform the following tasks in AutoCAD mode:

- "Importing SewerGEMS Data" on page 13-718

- "Working with Proxies" on page 13-718

Importing SewerGEMS Data

When running SewerGEMSin AutoCAD mode, this command imports a selected SewerGEMS data (.swg) file for use in the current drawing. The new project file will now correspond to the drawing name, such as, CurrentDrawingName.swg. Whenever you save changes to the network model through SewerGEMS, the associated .swg data file is updated and can be loaded into SewerGEMS.

Warning! A SewerGEMS Project can only be imported to a new, empty AutoCAD drawing.

Related Topic

- "Working with Proxies" on page 13-718

Working with Proxies

If you open a Bentley SewerGEMS V8*i* drawing file on an AutoCAD workstation that does not have the Bentley SewerGEMS V8*i* application installed, you will get an AutoCAD Proxy Information message box. This is because the executable logic for managing the AutoCAD entities is not available, and the Bentley SewerGEMS V8*i* modeling elements are not associated with the Bentley SewerGEMS V8*i* native database.

Bentley SewerGEMS V8*i* proxy objects can be moved and erased. However, doing so will put the drawing state out of sync with the model database if the drawing is saved with its original name. If this happens, and you later reload the drawing on an AutoCAD station that is running a Bentley SewerGEMS V8*i* application, the application will automatically load and will attempt to reconcile any differences it finds by automatically loading its Database Synchronization routine. (For more information, see "Drawing Synchronization" on page 13-713).

- "Importing SewerGEMS Data" on page 13-718

Chapter

Theory 14

Fundamental Solution of the Gravity Flow System

With increasing urbanization and urban renewal impacts driving the drainage and water quality regulatory framework, the design and analysis of storm water systems are becoming increasingly complex. The hydraulics characteristics of a drainage system often exhibit many complicated features, such as tidal or other hydraulic obstructions influencing backwater at the downstream discharge location, confluence interactions at junctions of a pipe network, interchanges between surcharged pressure flow and gravity flow conditions, street-flooding from over-loaded pipes, integrated detention storage, bifurcated pipe networks, and various inline and offline hydraulic structures. The time variations of the storm drainage design flow event are increasingly important in verifying total performance and achieving a measure of regulatory or design policy compliance. To better understand these complicated hydraulic features and accurately simulate flows in a complicated storm water handling system hydrodynamic flow models are necessary.

To simulate unsteady flows in storm water collection systems, numerical computational techniques have been the primary tools, and the results from numerical models are widely used for planning, designing and operational purposes. Since an urban drainage system can be composed of hundreds of pipes and many hydraulic control structures, the hydraulics in storm system can exhibit very complicated flow conditions. Consequently the numerical stability, computational performance, capabilities and robustness in handling complicated hydraulic conditions and computational accuracy are the major factors when deciding which approach to use to solve the hydraulic system.

Although many numerical methods have been developed to simulate the unsteady flows in sewer and storm water systems, including those based on explicit numerical schemes and those based on implicit schemes, limitations in most of models exist. SewerGEMS V8*i* features engines capable of solving the dynamic solution using both schemes. Users may select to either user EPA SWMM's native explicit solver or a custom implicit solver as more fully described in this section. The implicit solver is the default solver used in SewerGEMS V8*i*.

Flows in sewers are usually free surface open-channel flows, therefore the Saint-Venant equations of one-dimensional unsteady flow in non-prismatic channels or conduits are the basic equations for unsteady sewer flows. The dynamic model solution uses the following complete and extended equations:

$$\frac{\partial Q}{\partial x} + \frac{\partial (A + A_0)}{\partial t} - q = 0 \tag{14.1}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial (\beta Q^2 / A)}{\partial x} + gA\left(\frac{\partial y}{\partial x} - S_o + S_f + S_e\right) + L = 0 \tag{14.2}$$

Where	t	=	time
	x	=	the distance along the longitudinal axis of the sewer reach
	y	=	flow-depth
	A	=	the active cross-sectional area of flow
	A_0	=	the inactive (off-channel storage) cross-sectional area of flow
	q	=	lateral inflow or outflow
	$\hat{a}$	=	the coefficient for nonuniform velocity distribution within the cross section
	g	=	gravity constant
	S_o	=	sewer or channel slope
	S_f	=	friction slope due to boundary turbulent shear stress and determined by Manning's equation
	S_e	=	slope due to local severe expansion-contraction effects (large eddy loss)
	L	=	the momentum effect of lateral flow

A weighted four-point implicit scheme is used to solve the Saint-Venant equations. An implicit method is preferred over explicit since these methods have the advantage of maintaining good stability for large computational time steps and exhibit robustness in modeling systems that integrate the complex hydraulic interactions encountered in gravity sewer systems. The scheme was adopted since it handles unequal distance steps, its stability-convergence properties can be conveniently modified, and the internal (any hydraulic structures, such as dams, weirs, pumps, manholes etc) and external boundary conditions can be easily applied. The dynamic model is developed using the following four-point finite-difference scheme:

$$\frac{\partial f}{\partial t} = \frac{f_i^{j+1} + f_{i+1}^{j+1} - f_i^j - f_{i+1}^j}{2\Delta t_j} \tag{14.3}$$

$$\frac{\partial f}{\partial x} = \frac{\theta(f_{i+1}^{j+1} - f_i^{j+1}) + (1-\theta)(f_{i+1}^j - f_i^j)}{\Delta x_i} \tag{14.4}$$

$$f = \frac{\theta(f_i^{j+1} + f_{i+1}^{j+1}) + (1-\theta)(f_i^j + f_{i+1}^j)}{2} \tag{14.5}$$

in which θ is a weighting factor and the weighted four-point implicit scheme is unconditionally stable for >0.5. The value of θ of 0.6-0.8 is found to be optimal in maintaining stability and accuracy for large computational time steps.

The Newton-Raphson iteration method is used to solve the finite-difference equations derived from applying Equations 3 to 4 to Equations 1 and 2. Exceptional computational efficiency is achieved by special algorithm to iterate banded matrixes.

Application of the St. Venant Equation in Branched and Looped Networks

A model network could comprise only one or two branches or it could be as complicated as a system of hundreds of branches with cross-connections and various junctions containing different hydraulic structures and facilities, such as weirs and pumps. In many situations, the mutual flow interaction must be accounted for to achieve realistic results, particularly for unsteady flows since those confluence junctions can have significant effects due to the traveling dynamic waves traveling in a sewer system.

An extended relaxation technique is used in SewerGEMS V8*i* to achieve a balanced solution. This extended relaxation algorithm decomposes the storm network into individual branches and loops and solves each component using a four-point implicit scheme. In doing so, the algorithm treats the influences of interconnecting branches as a combination lateral flows and stage and discharge boundary condition.

Branches are ranked and ordered by a network parsing heuristic that traverses in depth-first fashion that sequences and weights according to conduit and channel conveyance characteristics (larger is ranked lower) and confluence angles (smaller is ranked lower). Loop-forming branches are isolated as part of a pre-traversal phase and are identified principally on the basis of localized slope and connectivity considerations. Generally, the lower the rank or branch number, the greater the branches hydraulic contribution to the total network response. Loops, if any, will typically have higher branch numbers.

Each branch confluence has two cross sections, which are located, just upstream and downstream of each of its junctions with connecting branches. During the numerical solution process, as each branch is solved by the Newton-Raphson iteration, an assumed lateral inflow or outflow is added at each junction reach to replace the confluence branch. The branches are automatically ranked such that the dendritic branches are always treated before those branches that form interconnecting loops. Also, a connecting branch ranked after the branch into which it is connected. This numbering scheme enables a stage boundary condition at the downstream of a branch to be determined using the average computed stages at the two confluence cross sections at the junction which the branch joins. In this way, each branch is independently solved one by one using the estimated lateral flows at each of the branch junctions. If the system has a total of J junctions, the relaxation is to iterate these J junction-related lateral flows. The relaxation equation for the lateral flows is:

$$q^{*} = (1-\alpha)q^{**} + \alpha Q \tag{14.6}$$

Where q^{*} = the estimated confluence lateral flow for the next iteration

q^{**} = the previous estimated lateral flow

Q = the computed discharge at the downstream end of the connecting branch in the previous iteration

$\acute{a}$ = a weighting factor ($0<\alpha<1.0$)

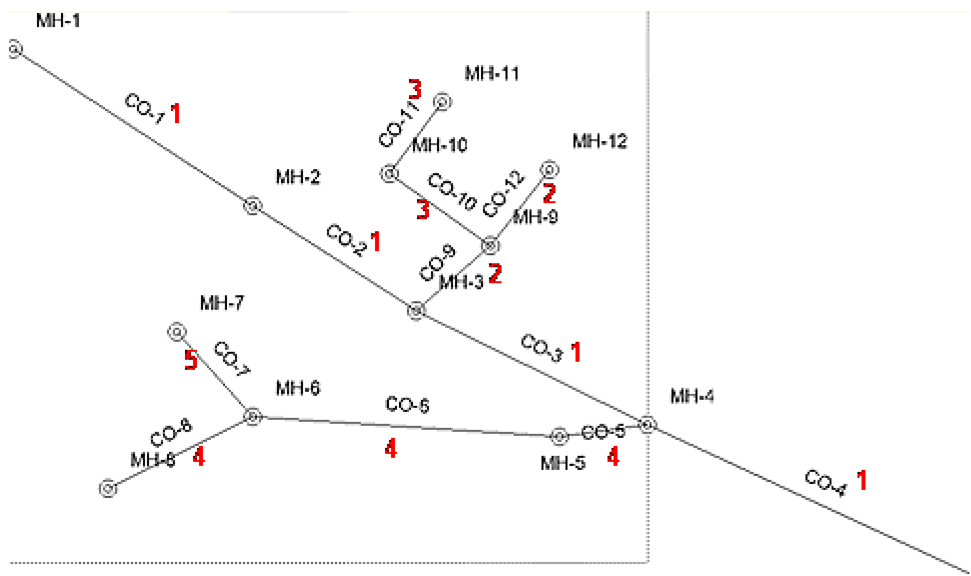
Values of α between 0.8 and 0.9 provide the most efficient convergence for the relaxation iteration. Extensive tests have shown that the relaxation iteration convergence is achieved within one to three iterations for almost all situations using $\alpha = 0.6$.

Branches

The implicit dynamic engine solves the St. Venant equations along straight branches of conduit or channel starting at the most downstream outfall. Branch 1 starts at the outfall and upstream until it reaches the first junction. There it follows the junction with the largest conduit and/or the conduit with the alignment that matches the outlet pipe alignment. This continues until the branch reaches the most upstream node. At this point a second branch starts from the largest pipe from the first junction that was not in branch 1. This branch continues to its most upstream point. Once these branches are numbered, branches that start at pump station wet wells are traced out to their source.

An example of the branch labeling is shown in the figure below. The red numbers indicate branches. In the figure, branch 1 is made up of 30 in. pipes; branch 2 is made up of 24 in. pipes while the other branches consist of 18 in. pipes.

Figure 14-1: Branch Labeling



Section Count

The element property Section Count refers to the number of spatial sections into which the element is divided along its length by the implicit numerical engine. For any element there will be a minimum of five sections. Depending on the value of the Computational Distance property, which you set in the Property Editor for Calculation Options, additional sections are added for longer pipes. The default computational distance is 50 feet so that there will be five sections for any element up to 250 ft. Beyond that length, a section is added for each 50 ft of length. You can control the number of sections by increasing or decreasing the computational distance, which will decrease or increase the number of sections accordingly.

Special Considerations

Gravity sewer collection systems are subject to a number of special hydraulic conditions that must be considered in developing a full and complete solution scheme. These special conditions challenge the basic algorithm since they are not explicitly accounted for in the basic solution for 1-D gravity flow. The hydraulics engine has been extended to account for these conditions and the numerical adaptations are described in this section, which includes the following topics:

- "Pressurized Flow" on page 14-724
- "Mixed (Transcritical) Flow" on page 14-726
- "Dry Bed (Low Flow)" on page 14-727
- "Steep Reaches" on page 14-728
- "Flooding" on page 14-728

Related Topics:

- "Fundamental Solution of the Gravity Flow System" on page 14-719
- "Application of the St. Venant Equation in Branched and Looped Networks" on page 14-721
- "Section Hydraulics" on page 14-729

Pressurized Flow

The typical gravity sewer network is dominated by circular pipe segments. These pipes are all closed and characterized by a converging top where the hydraulic top width approaches zero as flow transitions from free surface to pressure. The Preissmann slot method is used for simulating pressure or surcharged flows by adapting the conceptualization of pressurized flow to fit a free surface model. The slot extends vertically from pipe crown to infinity and over the entire length the pipe, and the width of the slot is usually 1% of the characteristic pipe dimension (diameter for a circular pipe) but not large than 0.02 ft.

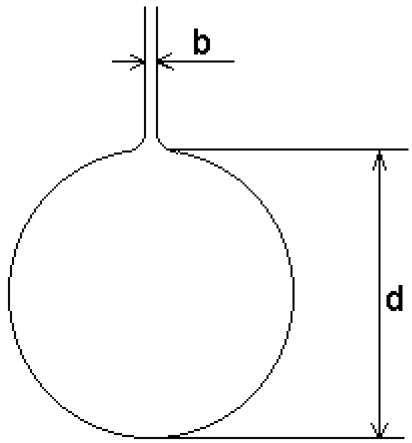


Figure 14-2: Transition of a Circular Pipe to the Slot

Since a circular conduit width changes dramatically near the crown and in order to maintain a smooth transition between conduit width and the slot width, the

$$\frac{b}{d} = 0.28\left(\frac{0.98}{y/d}\right)^{32}$$

s
where $0.98 < y/d < 1.2$
$b/d = 0.001$ and
$y/d > 1.2$, and

Where	b	=	conduit and slot width
	d	=	circular conduit diameter
	y	=	flow depth

The maximum width allowed in the slot is 0.01 ft. Also, when the flow depth is above the diameter d the area remains the full circular section area therefore the slot will have no impact on the flow continuity.

The significant advantages in using this hypothetical slot are apparent in simulating the moving transitional interface between open-channel flow and pressure flow, which can happen anywhere at any time in a sewer system. Since the model applies a unified set of consistent equations and numerical schemes, it makes no special switches between open-channel flows and pressure flows, giving rise to a robust solution.

Related Topics:

- "Mixed (Transcritical) Flow" on page 14-726
- "Dry Bed (Low Flow)" on page 14-727
- "Steep Reaches" on page 14-728
- "Flooding" on page 14-728

Mixed (Transcritical) Flow

One of the challenging features in the unsteady flows in a sewer or storm water drainage system is the interchanging or moving interface of different flow regimes between subcritical and supercritical flows. This is largely due to the fact that an urban hydraulic system can experience a large range of slopes of conduits and it is common to have significant slope changes at many pipe junctions. A good numerical model for sewer and storm water system has to be able to handle the mixed flow regimes and interchanges with great robustness.

When modeling unsteady flows, the dynamic routing technique using the four-point implicit numerical scheme tends to be less numerically stable than the diffusion (zero inertia) routing technique for certain mixed flows, especially in the near critical range of the Froude number (F_r) or mixed flows with moving supercritical/subcritical interfaces. It has been observed that the diffusion technique, which eliminates the two inertial terms in the momentum equation, produces stable numerical solutions for flows where F_r is in the range of critical flow (F_r=1.0) and for supercritical flows. To take advantage of the diffusion method's stability and retain the accuracy of the fully dynamic method, the Local Partial Inertia modification (LPI) technique is used in the dynamic sewer model. In the LPI technique, the momentum equation, Equation 11.2, is modified by a numerical filter, σ, so that the inertial terms are partially or totally omitted based on the time-dependent local hydraulic conditions.

The modified equation and numerical filter are:

$$\sigma[\frac{\partial Q}{\partial t} + \frac{\partial(\beta Q^2/A)}{\partial x}] + gA(\frac{\partial y}{\partial x} - S_0 + S_f + S_i) + L = 0 \tag{14.7}$$

in which σ is a numerical modifier and its value for every finite-difference box (between x_i and x_{i+1}) will be determined at each time step by the following equation:

$$\sigma = \begin{cases} 1.0 - F_r^{\,m} & F_r \le 1.0 \\ 0 & F_r > 1.0 \end{cases} \tag{14.8}$$

in which m is a user specified constant and $m \geq 1.0$. It is found that smaller values of m tend to stabilize the solution in some cases while larger values of m provide more accuracy.

The LPI technique was developed by Dr. Ming Jin and Dr. Danny Fread and this technique has been adapted by Federal dynamic models such as NWS Fldwav model, USACE HEC-RAS unsteady flow model and EPA-SWMM model.

Related Topics:

- "Pressurized Flow" on page 14-724
- "Dry Bed (Low Flow)" on page 14-727
- "Steep Reaches" on page 14-728
- "Flooding" on page 14-728

Dry Bed (Low Flow)

For the dry flow condition, the numerical model applies a very small initial steady flow (virtual flow) at the start the simulation. This virtual flow is applied system-wide and has negligible effect on the computational results over the full simulation. The engine distributes and manages the virtual flow allocation and de-allocation dynamically across all the network branches and loops over the full course of the simulation, and sophisticated algorithms are developed to distribute the virtual flows in the way that they will not be accumulative and they have only local impacts on the very low flow conditions. These virtual flow assignments are based on a tiny threshold value that is dynamically adjusted over the duration of the analysis. A virtual flow filter algorithm adjusts the results for the virtual flow quantities and depths by subtracting the virtual flow effects from the hydraulic results at each time step at each solution point over the network.

To users, these virtual flows are invisible and there is no practical impact on the computational results.

Related Topics:

- "Pressurized Flow" on page 14-724
- "Mixed (Transcritical) Flow" on page 14-726
- "Steep Reaches" on page 14-728
- "Flooding" on page 14-728

Steep Reaches

Unlike a natural river, a storm drainage system can often have pipes of very steep slopes, sometimes more than 30%. The flows in such steep pipes are overwhelmingly very supercritical. A kinematical treatment is applied on such very steep pipes in which the Manning equation is used to replace the momentum equation during the solution process.

Related Topics:

- "Pressurized Flow" on page 14-724
- "Mixed (Transcritical) Flow" on page 14-726
- "Dry Bed (Low Flow)" on page 14-727
- "Flooding" on page 14-728

Flooding

A unique hydraulic condition in the storm sewer modeling is the overcharged-flow-resulted street surface flooding. This is the condition in which the drainage flow into the sewer pipe is much larger than the sewer capacity and the depth is built higher than the ground surface elevation. In addition, at the sewer junctions (manholes) where there may be open access to the ground, the flow starts to go upward through the manhole openings, overtop the manhole rims.

There are two scenarios after the street flooding occurs:

- If there is a surface gutter or channel connected to the manhole, the overflowing water will join the surface gutter or channel and will be accounted for and simulated as part of the flows in the gutter subsystem. These flows may drain back to the sewer subsystem somewhere downstream.
- If there is no surface gutter or channel connected to the overflowed manhole, the overtopped flows leave the sewer system and these flows are lost to the system; this will be reflected by a flow volume loss. In this condition, there may also be a storage area above the ground elevation and below the user-specified overtop elevation. The water stored in the storage area will drain back to the manhole when the water elevation recesses. Users can specify the storage areas and the street-flooding-overtop elevation. A default overtop elevation is the ground rim elevation, assuming there is no storage effects.

The implicit hydraulic engine treats the street overtopping overflow as weir flow and uses a weir equation to determine the overflow. The weir crest elevation is the user-specified street overtop elevation and the weir length is determined by an empirical equation:

$$W_L = 6.0(1 + dh) \tag{14.9}$$

Where W_L = overflow weir crest length

dh = the head over the overflow weir

Related Topics:

- "Pressurized Flow" on page 14-724
- "Mixed (Transcritical) Flow" on page 14-726
- "Dry Bed (Low Flow)" on page 14-727
- "Steep Reaches" on page 14-728

Section Hydraulics

Within the hydraulics solver the decomposed network branches and loops comprise a series of reach segments and/or structures that are logically ordered from upstream to downstream by the numerical engine. Each reach segment consists of either a prismatic conduit section or a natural channel segment described by separately defined upstream and downstream open channel sections.

This section includes the following topics:

- "Conduit Shapes" on page 14-729
- "Natural Reach Shapes" on page 14-741
- "Virtual Link Types" on page 14-742
- "Roughness Models" on page 14-742

Conduit Shapes

The supported conduit shapes are shown in Figures 11-2 to 11-21. Each shape is parameterized by one, two, or more characteristic dimensions as shown in the reference figure. In this model, a conduit is taken to be a prismatic (constant-shaped) conveyance segment that is defined by a single shape. Conduits do not have to be closed sections, so prismatic design channels can be modeled using conduit elements.

The allowable conduit shapes include:

- “Circular Channel” on page 14-731
- “Trapezoidal Channel” on page 14-731
- “Basket Handle” on page 14-732
- “Ellipse” on page 14-732
- “Horseshoe” on page 14-733
- “Egg” on page 14-733
- “Semi-ellipse” on page 14-734
- “Pipe-Arch” on page 14-735
- “Semi-Circle” on page 14-736
- “Catenary” on page 14-736
- “Gothic” on page 14-737
- “Modified Basket Handle” on page 14-737
- “Triangle” on page 14-738
- “Rectangular Channel” on page 14-738
- “Irregular Open Channel” on page 14-739
- “Irregular Closed Section” on page 14-739
- “Rectangular-Rounded” on page 14-740
- “Rectangular-Triangular” on page 14-740
- “Power” on page 14-741
- “Parabola” on page 14-741

In addition, SewerGEMS V8*i* supports the following:

- “Natural Reach Shapes” on page 14-741
- “Virtual Link Types” on page 14-742

Circular Channel

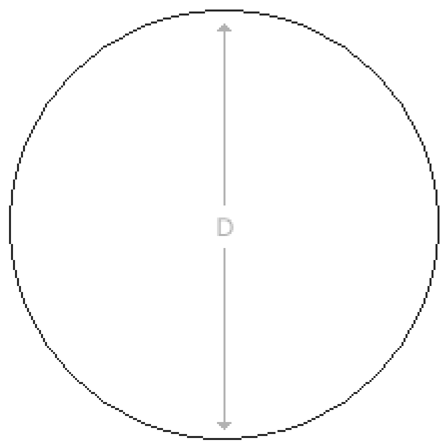


Figure 14-3: Circular Channel Shape

Trapezoidal Channel

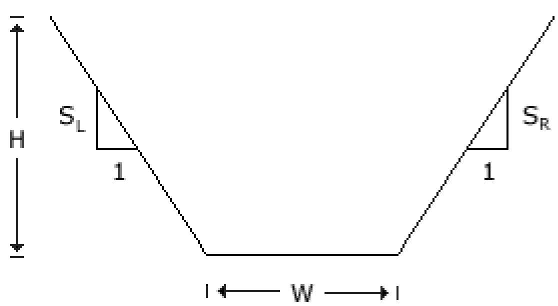


Figure 14-4: Trapezoidal Channel Shape

Basket Handle

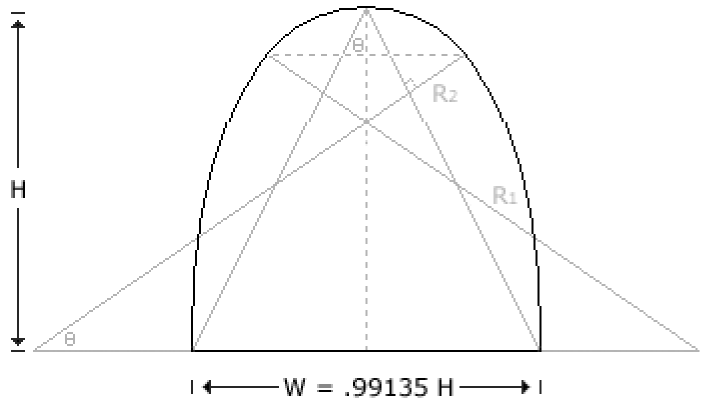


Figure 14-5: Basket-Handle Shape

Ellipse

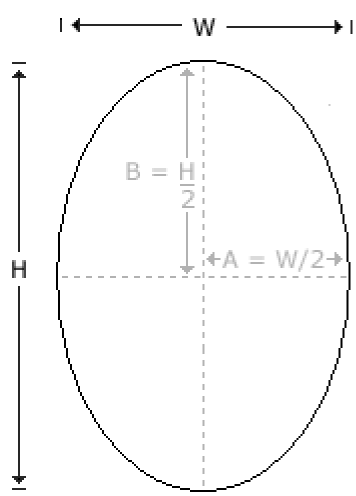


Figure 14-6: Ellipse Shape

Horseshoe

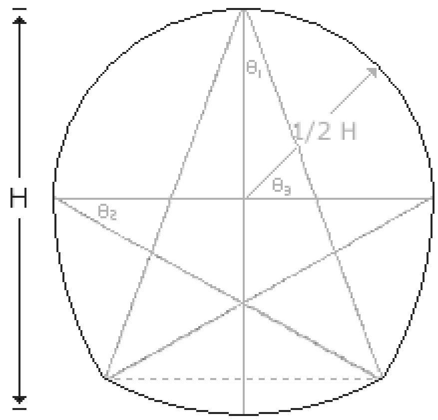


Figure 14-7: Horseshoe Shape

Egg

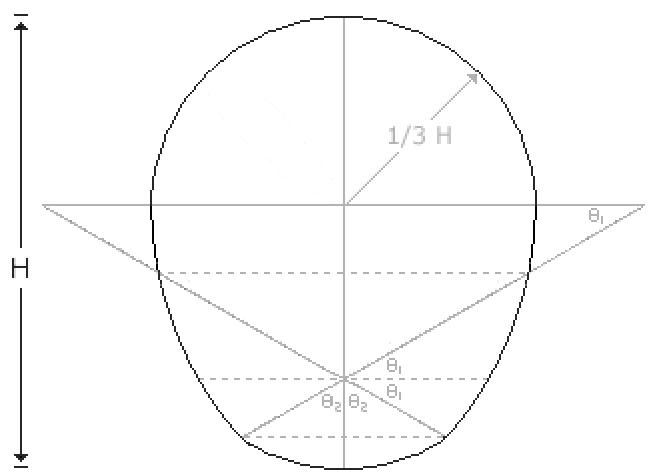


Figure 14-8: Egg Shape

Semi-ellipse

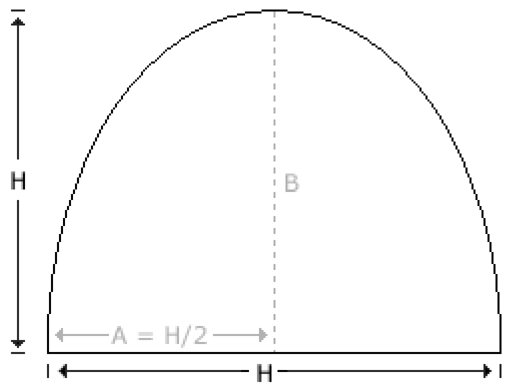


Figure 14-9: Semi-Ellipse Shape

Pipe-Arch

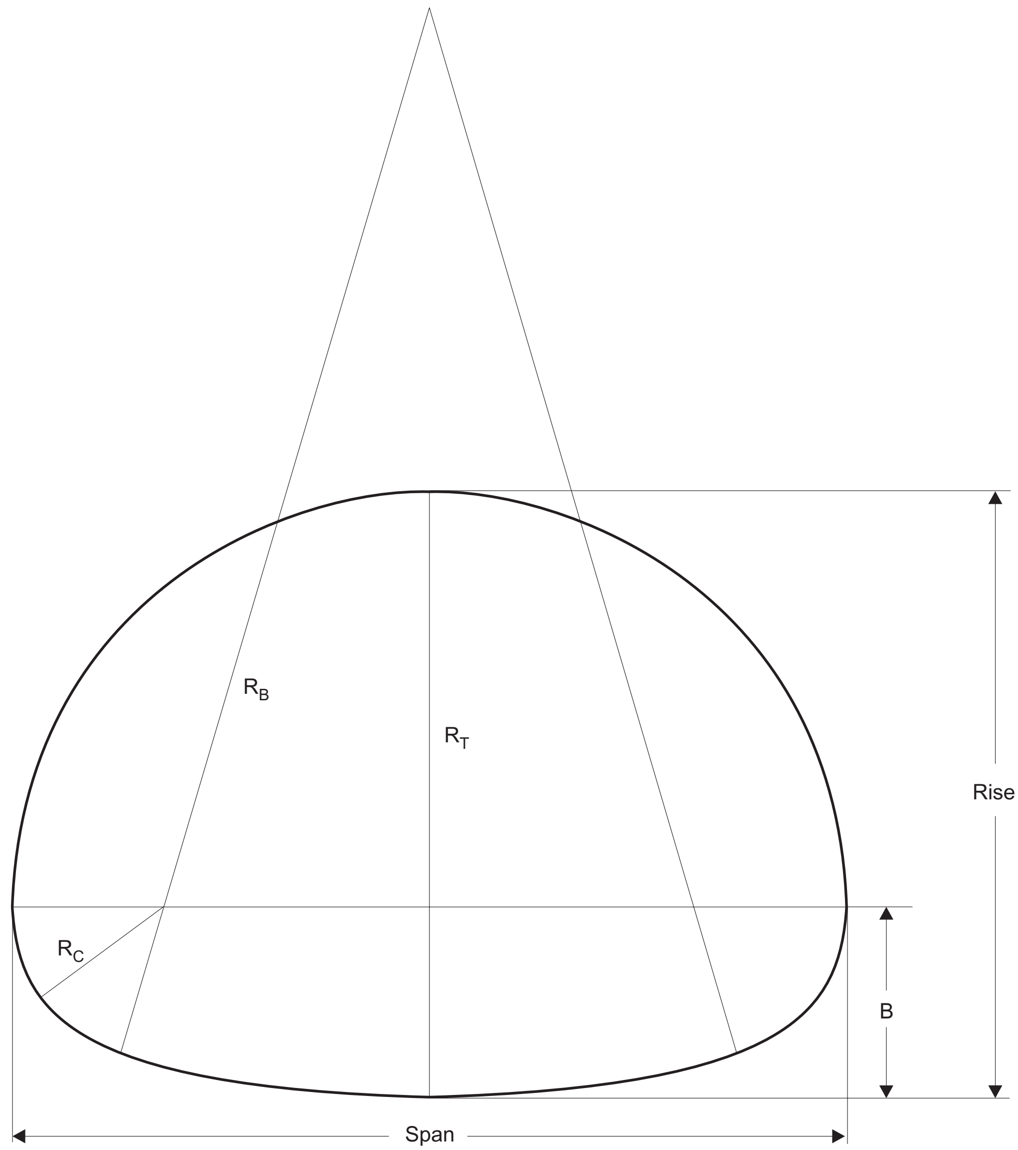


Figure 14-10: Pipe-Arch Shape

Semi-Circle

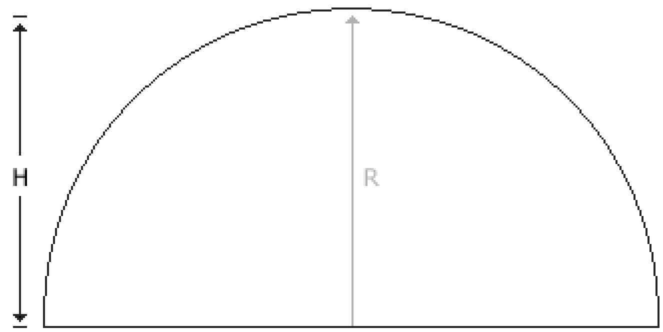


Figure 14-11: Semi-Circular Shape

Catenary

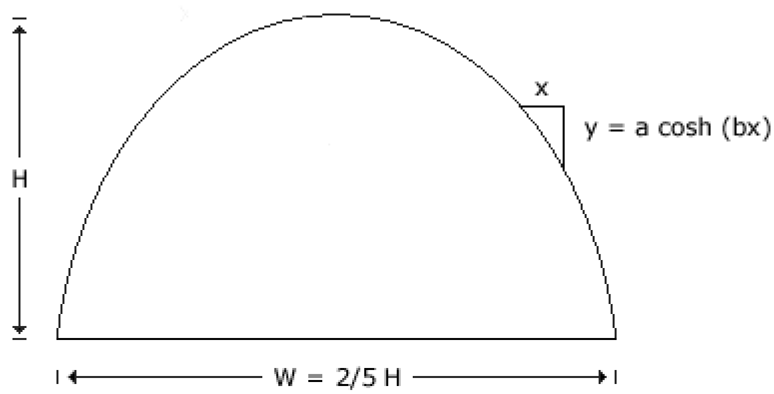


Figure 14-12: Catenary Shape

Gothic

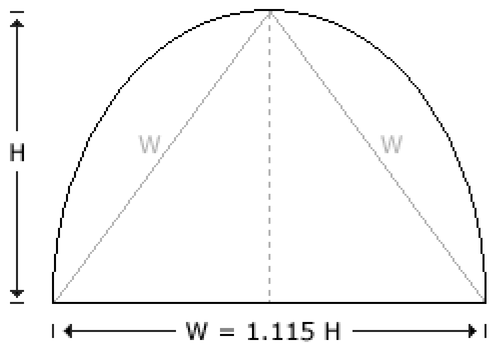


Figure 14-13: Gothic Shape

Modified Basket Handle

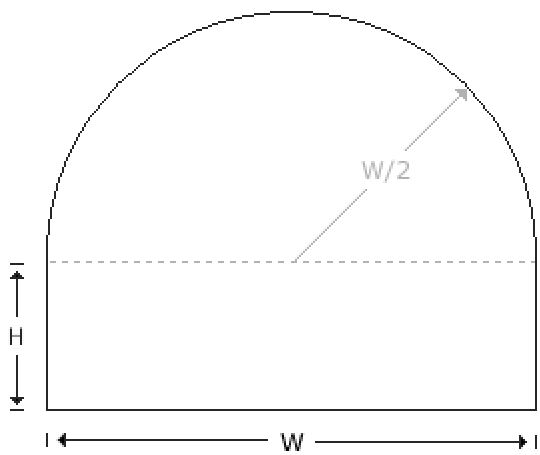


Figure 14-14: Modified Basket-Handle Shape

Triangle

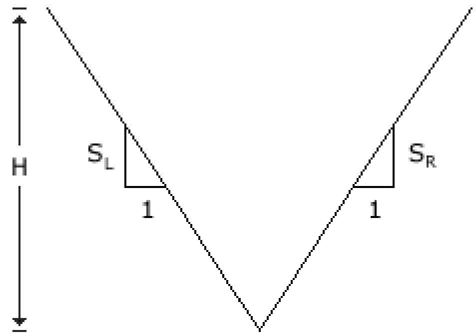


Figure 14-15: Triangle Shape

Rectangular Channel

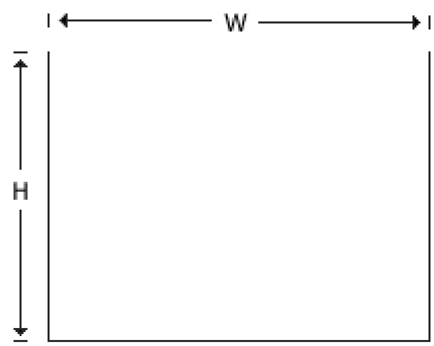


Figure 14-16: Rectangular Channel Shape

Irregular Open Channel

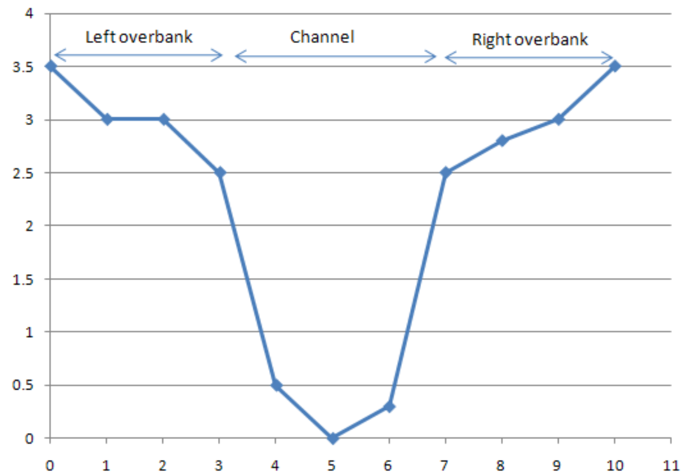


Figure 14-17: Irregular Open Channel Shape

Irregular Closed Section

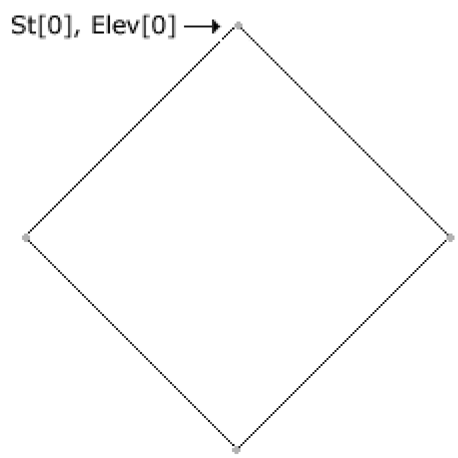


Figure 14-18: Irregular Closed Section Shape

Rectangular-Rounded

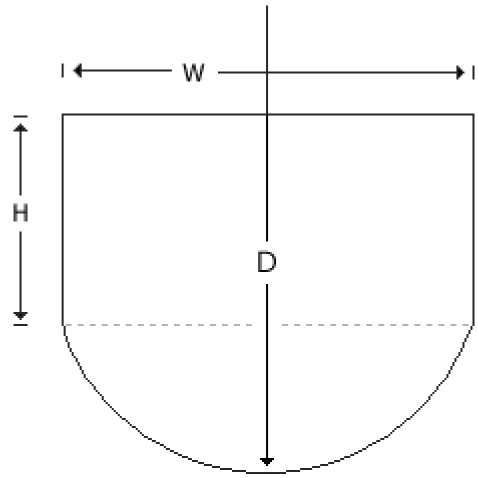


Figure 14-19: Rectangular-Rounded Shape

Rectangular-Triangular

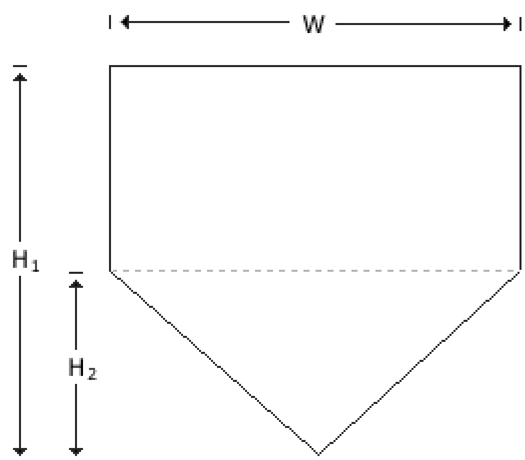


Figure 14-20: Rectangular-Triangular Shape

Power

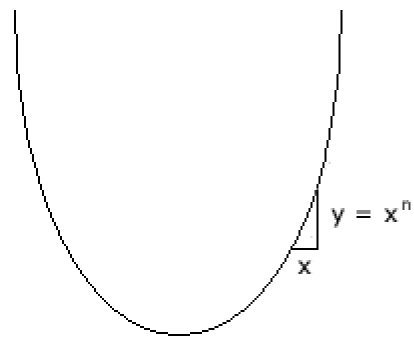


Figure 14-21: Power Shape

Parabola

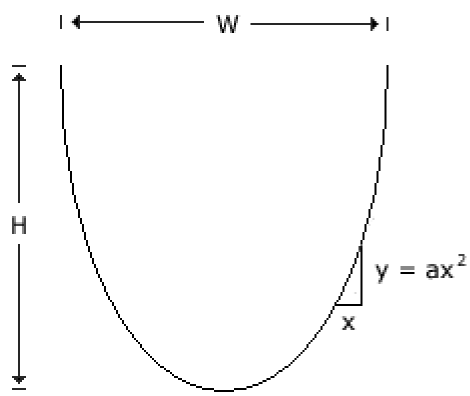


Figure 14-22: Parabola Shape

Natural Reach Shapes

As in most river models, a natural channel branch can be taken as a series of gradually varying sections. Natural channels segments that describe a branch are defined using an upstream and downstream open channel section element. The following section shapes may be used to define natural channels:

- Irregular Closed Section
- Trapezoidal

For natural sections, the engine will automatically insert the required computational sections along the reach by interpolating a top width versus elevation table that is dynamically built according to the maximum number of input points that describes either end-section.

Note: The SWMM engine does not support the notion of a natural channel described between two open channels cross-sections. So, when solving using the SWMM engine, each open channel reach will be modeled using the upstream section shape. Using a conduit element with an irregular shape will provide computational consistency between the SWMM and implicit engines.

Virtual Link Types

The model includes a virtual link type that can be used to achieve fidelity with the SWMM modeling abstraction as applied to certain hydraulic element types such as weirs, orifices, and pumps. Since SWMM treats each of these elements as a logical topological link that is contrary to spatial reality, a virtual section can be supplied to treat these nodal elements in a way that matches the SWMM model. In the model, a virtual link is a placeholder element that conveys flow directly while suppressing all hydraulic effects on the network.

Roughness Models

The SewerGEMS V8*i* solver uses the Manning's equation to evaluate the friction slope term, S_f, in Equation 11-3:

$$S_f = \frac{n^2 |Q| Q}{\mu^2 A^2 R^{4/3}} = \frac{|Q| Q}{K^2} \tag{14.10}$$

Where	n	=	Manning coefficient for friction
	μ	=	unit conversion factor (1.49 for US Customary and 1.0 for SI units)
	R	=	hydraulic radius
	K	=	flow conveyance factor

The Manning's n is a user-defined value that introduces the effects of conduit, channel, or gutter roughness. The K that is actually applied for a segment between two interpolated locations along the computational stream is evaluated by averaging the K values computed for the two locations.

For more information on the application of this roughness model, see "Implementations" on page 14-743.

Implementations

This section describes the applications of the roughness model available in Bentley SewerGEMS V8*i*.

Single Roughness

The simplest application of the Manning's model is to supply a single roughness value to the segment being modeled.

Horizontal Variation

The modeler can describe the horizontal variation of roughness across a natural section. Horizontally varied roughness is automatically pre-processed and described to the engine as a vertical variation using Pavloskii's Method:

$$n = \sqrt{\frac{\sum_{1}^{N}(P_N n_N^2)}{P}} = \sqrt{\frac{P_1 n_1^2 + P_2 n_2^2 + \cdots + P_N n_N^2}{P}} \tag{14.11}$$

Where	n	=	Roughness coefficient
	P	=	Weighted perimeter
			Subscripts represents subdivisions of one given section

Overbank Segments

This roughness model is widely applied in floodplain analysis and is a useful way to describe the overbank and channel components of a river reach. In these circumstances the conveyance factor for the section is computed as follows:

$$K_L = \frac{\mu}{n_L} A_L R_L^{\ 2/3}$$

$$K_C = \frac{\mu}{n_C} A_C R_C^{\ 2/3}$$

$$K_R = \frac{\mu}{n_R} A_R R_R^{2/3}$$

$$K = K_L + K_C + K_R$$

Where	L	=	left floodplain
	C	=	channel
	R	=	right floodplain

Overbank segment manning types are converted to horizontal variation roughness types, which can be converted to vertical variation types using Pavlovskii's Method.

Vertical Variation

A Manning's n versus depth relationship can be supplied. In the case of irregular sections, the engine simply interpolates the roughness value to apply for each interpolated internal top width value that is developed by the algorithm for the section.

Flow Roughness

Describing Manning's n versus flow is a roughness model that can be used in natural river applications where the flow record can be used to calibrate the n.

Hydraulic Boundaries

In order to numerically solve the Saint Venant equations, boundary conditions are needed in the model to provide the necessary additional equations to form a complete set of equations.

There are two types of hydraulic boundaries:

- "External Boundaries" on page 14-745
- "Internal Boundaries" on page 14-745

External Boundaries

External boundaries in a sewer system include outfalls at the downstream ends and very first section at the upstream ends. For the upstream end boundaries, usually a simple zero flow is used as upstream boundary condition or a flow time series can be used as upstream boundary condition.

There are a few different boundary conditions users can select for the outfall at the downstream end:

- A constant user-defined tail water elevation.
- A user-defined water elevation time series (time-elevation curve), such as a tide surface elevation time series.
- A user-defined tabular relation between the outfall water elevation and outflow discharge (elevation-flow curve), often called as single-valued rating curve or simply rating curve. Sometimes more than one outfall discharges to one receiving point; in this situation, the discharge in the rating curve would be the summation of all the flows from these discharging pipes.
- A free outfall, which means that the outflow is freely discharged without any anticipated backwater effects. In this case, the model automatically applies the proper boundary equation, either a normal flow equation or a critical flow equation, to the outfall boundary based on the dynamic hydraulic condition at the boundary. The normal flow equation will be used if the flow is in supercritical condition and the critical flow equation will be used if the flow is subcritical.

In the first three cases, the control elevation h at the downstream boundary (outfall) is determined from the curves at each time step. It can be replaced by normal or critical flow elevations if it falls below those normal or critical elevations.

The dynamic model also supports boundary elements, such as ponds or storage nodes, as downstream boundaries even when there are no further outflow outlets from there. In this case, a storage equation is used as a boundary condition. If there are no outlets from these boundary elements, then these elements are treated as internal regular elements.

Internal Boundaries

Along a sewer pipeline, there are hydraulic structures and control devices, such as manholes, weirs, and orifices where the flow is often rapidly varied rather then gradually varied in space. The Saint-Venant equations are not applicable at these locations since the gradually varied flow assumption in the Saint-Venant equations derivation is no longer valid. Instead these locations are treated as hydraulic internal boundaries; usually alternative empirical internal boundary equations are used for these internal local computational reaches (a computational reach is a link between two computational sections).

Typical internal boundaries are:

- "Manholes and Sewer Junctions" on page 14-746
- "Flow Control Structures" on page 14-748
- "Culverts" on page 14-754

Manholes and Sewer Junctions

Manholes and sewer junctions are the most common internal boundaries. Hydraulically they represent significant changes in many properties such as bottom slope, boundary roughness, and cross section shape. They may have different vertical and horizontal alignments, such as drop manhole or perched manhole. As a consequence of these significant hydraulic property changes, the dynamic hydraulic conditions in manholes and junctions are very complicated and modeling these conditions is one of the most challenging aspects in sewer dynamic modeling.

Usually a manhole and a junction has a storage area and may have open access to ground surface (the user would be able to set a manhole as bolted so that the access to the ground is turned off). SewerGEMS' dynamic model applies a manhole storage equation (a form of continuity equation) as one of the internal boundary equations. When the water elevation is above the ground rim elevation, additional street storage and street flooding may occur. For more information about flooding, see "Flooding" on page 14-728.

Related Topics:

- "Junction Headloss Methods" on page 14-746
- "Minor Losses" on page 14-747
- "Flow Control Structures" on page 14-748
- "Culverts" on page 14-754

Junction Headloss Methods

Another internal boundary equation is the energy equation, in which a user selects different head loss methods to calculate the head loss in a manhole and a junction:

- **Standard loss method** - a user-defined loss coefficient is used to calculate the head loss based on the velocity head of the exit conduit.

- **Absolute loss method** - a user-defined loss amount (in feet) is used as the head loss.
- **HEC-22 loss method** - a procedure of calculating the junction head loss specified in DOT HEC-22 manual is used to calculate the head loss.
- **Generic loss method** - a user defined loss coefficient is used to calculate the head loss based on the velocity head difference between entry and exit conduits.
- "Manholes and Sewer Junctions" on page 14-746

Minor Losses

Minor losses in pressure pipes are caused by localized areas of increased turbulence that create a drop in the energy and hydraulic grades at that point in the system. The magnitude of these losses is dependent primarily upon the shape of the fitting, which directly affects the flow lines in the pipe.

The equation most commonly used for determining the loss in a fitting, valve, meter, or other localized component is:

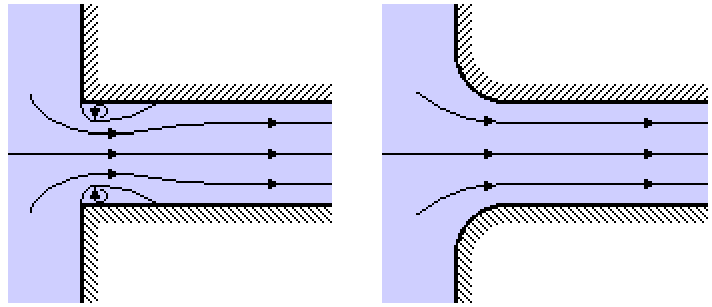
$$h_m = K\frac{V^2}{2g} \tag{14.12}$$

Where	h_m	=	Loss due to the minor loss element (m, ft)
	V	=	Velocity (m/s, ft/s)
	g	=	Gravitational acceleration constant (m/s2, ft/s2)
	K	=	Loss coefficient for the specific fitting

Typical values for the fitting loss coefficient are included in the Fittings Table at the end of this chapter.

Generally speaking, more gradual transitions create smoother flow lines and smaller headlosses. For example, the figure below shows the effects of a radius on typical pipe entrance flow lines.

Flow Lines at Entrance



- "Manholes and Sewer Junctions" on page 14-746

Flow Control Structures

Flow regulating structures, also known as control structures, are very common in storm water drainage systems and in combined sewer systems. The most common control structures are weirs and orifices.

In SewerGEMS V8*i*, you can attach a weir or orifice at either the upstream end or the downstream end of a conduit, or at both ends of the conduit. A control can also have a flap gate which allows flow to travel in only one direction. Hydraulically these controls are treated as internal boundaries, i.e., the empirical weir or orifice equations are used to replace the momentum equations in the Saint Venant equations and the continuity equation is simply that the flow is the same between the upstream face and the downstream face of the internal boundary (control structure).

For more information on flow control structures, see:

- "Weirs" on page 14-748
- "Orifices" on page 14-753
- "Rating Curves" on page 14-754
- "Manholes and Sewer Junctions" on page 14-746

Weirs

Weirs are classified by their flow-diversion purpose as either a side weir or a transverse weir, as described in the following definitions:

- Side weirs or overflow weirs are used to divert extra high flows to overflow waterways. Typically a side weir is a weir parallel to the main sewer pipe and with enough high crest elevation to prevent any discharge of dry-weather flow, but it is also low and long enough to discharge required excess of wet weather flow. Weirs in an outlet of a detention pond can be treated as one of the control elements in the composite outlet control structure. Another example of a side weir is the emergency overflow weir or spillway at the top of a detention pond.
- Transverse weirs or inline weirs are typically placed directly cross the sewer pipe, perpendicular to the sewer flow and act like a small dam, to direct the low flow, usually dry weather flow, to diversion pipe such as dry weather flow interceptor sewer pipe.

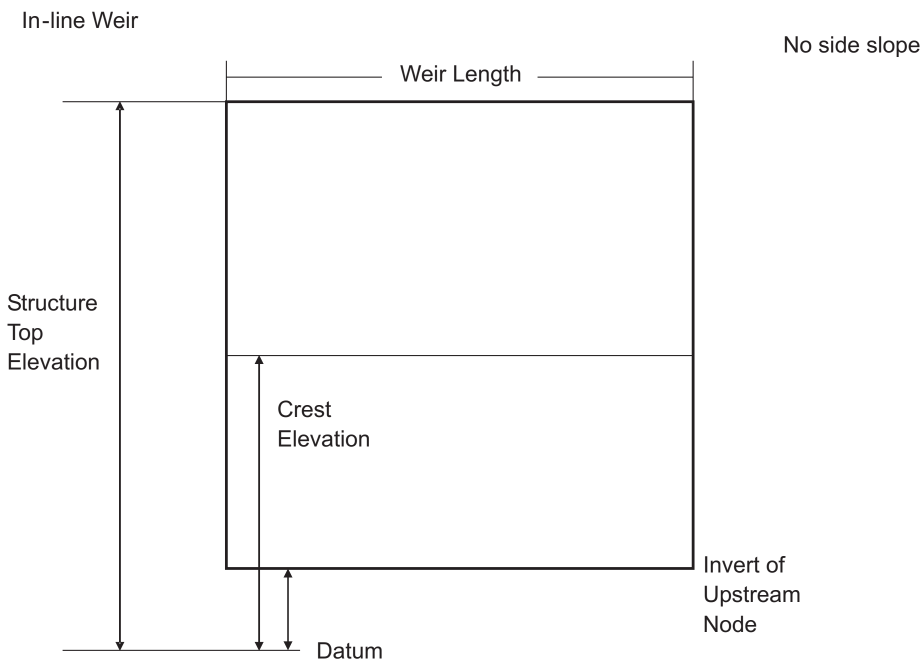
Weirs are also classified by their cross section shapes, such as rectangular, V-notch, trapezoidal, and irregular. Accordingly the computational equations for the weirs are different, the discharge through a rectangular weir is proportional to the 1.5 exponent of the head above the weir crest, and the exponent for the V-notch weir becomes 2.5.

Bentley SewerGEMS V8*i* users need to specify the weir discharge coefficient. Typically a weir discharge coefficient ranges between 2.65 and 3.10 (English units). Since the weirs in a sewer system are mostly sharp-crested weirs, a value of 3.0 is a common default assumption without knowing the weir specifics and hydraulic conditions.

There are three types of in-line weirs:

- "In-Line (Rectangular) Weir" on page 14-750
- "Trapezoidal Weir" on page 14-751
- "V-Notch (Triangular) Weir" on page 14-752

In-Line (Rectangular) Weir



The flow is given by:

$$Q = C(B - n(0.1h))h^{1.5} \tag{14.13}$$

Where	B	=	weir length, L
	h	=	effective head, L
	C	=	weir coefficient
	n	=	number (0,1,2) of end contractions

The weir coefficient can be further given (for weirs stretching across the channel) by:

$$Q = \frac{2}{3} C_v \sqrt{2g}\, b_e\, H_e^{1.5} \tag{14.14}$$

Where g = gravity

b_e = effective width (essentially the width)

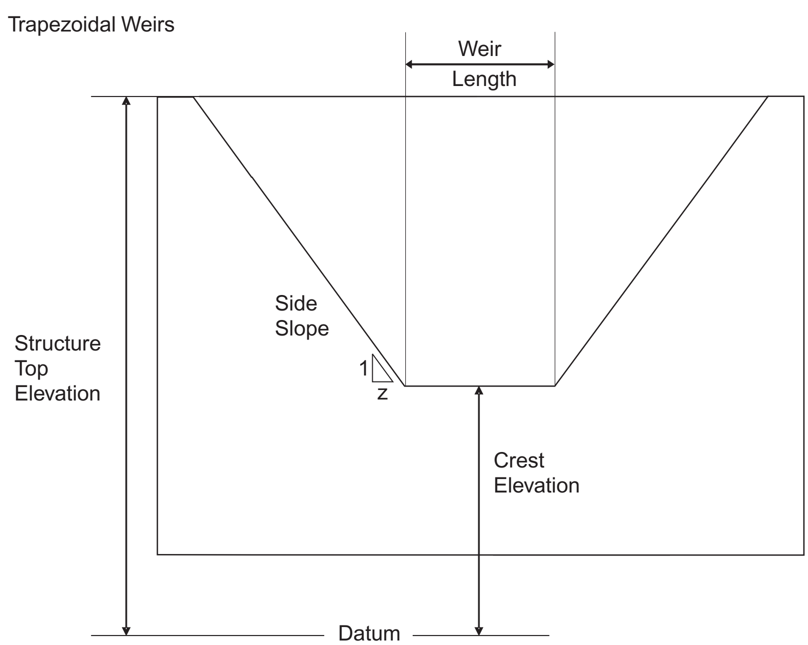
H_e = effective head (essentially the head)

C_v = 0.602 + 0.075 h/p for full width weir
0.587 - 0.023 h/p for fully contracted weir

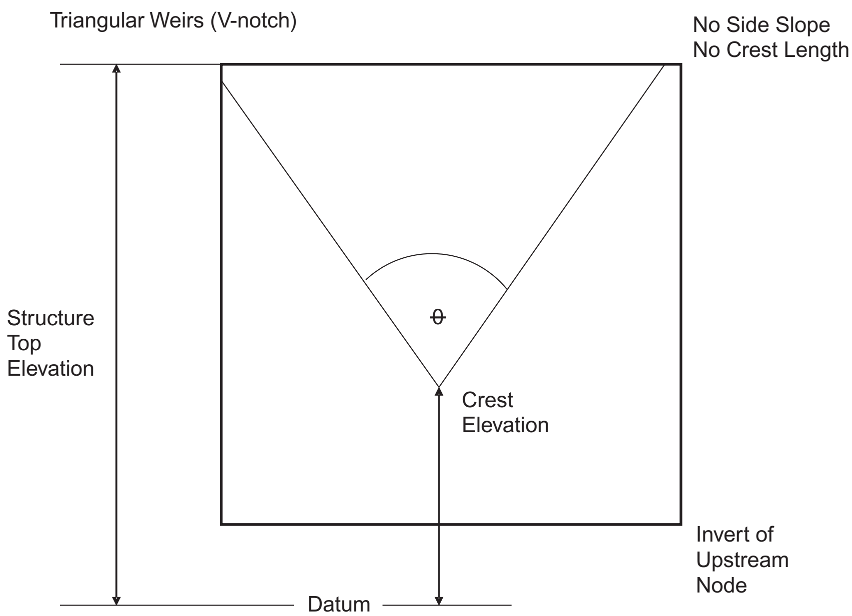
where
p = depth of weir crest above channel bottom
h = head

Trapezoidal Weir

The following illustration assumes that the trapezoidal weir is equivalent to a rectangular channel and a V-notch weir.



V-Notch (Triangular) Weir



The parameter, Θ, must be given in degrees (not radians). The flow for a V-notch weir is given by:

$$Q = C\tan(\theta/2)\,h^{2.5} \qquad (14.15)$$

Where h = head above weir crest, L

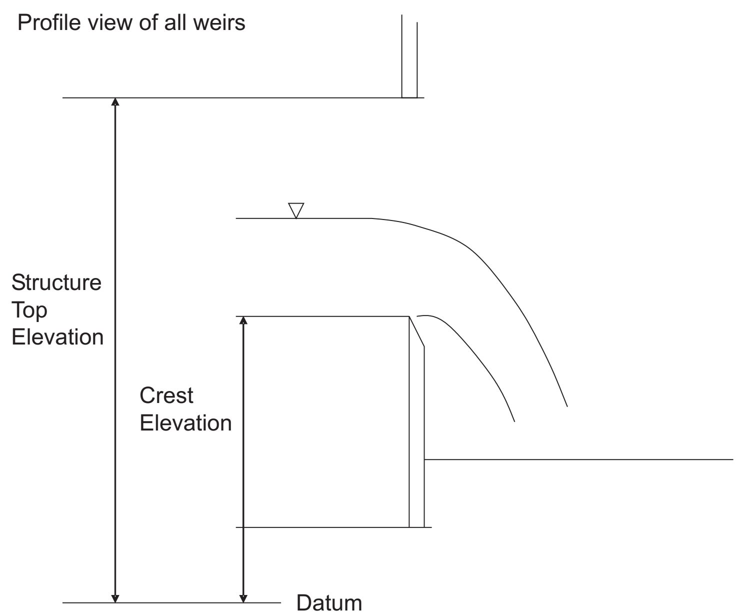
$$C = \left(\frac{8}{15}\right)\sqrt{2g}\,C_v \qquad (14.16)$$

Where C_v = 0.58 for fully contracted weirs with h > 1.2p

C = approximately 1.4 for SI units and 2.5 for U.S. customary units

Elevations for weirs must be specified relative to the datum for the problem, not the invert of the channel. In general:

Invert elevation < Crest Elevation < Structure Top Elevation



Orifices

Orifices are usually circular or rectangular openings in the wall of a tank or in a plate normal to the axis of the conduit. Orifices can be oriented in a variety of ways, such as side outlet or bottom outlet. SewerGEMS V8*i* can also treat an orifice as one of the controlling elements in a detention pond composite control structure; other controlling elements within a composite control structure include weirs, risers and culverts.

Orifices are treated the same as weirs to be internal boundaries except that the flow equation of an orifice is used to calculate the discharge. There are different flow conditions in an orifice and the calculation of the discharge through the orifice is different:

- The discharge through the orifice is proportional to the 0.5 exponent of the head if the orifice is fully submerged.
- A weir equation is usually used for unnumbered conditions of the orifice.
- Special treatment is necessary for a smooth transition between unsubmerged and submerged conditions due to the calculating equation switch.

The orifice discharge coefficients typically range between 0.6 and 0.7 (English units). Without knowing the orifice specifics, a default value of 0.65 is commonly used.

Rating Curves

Another generic control structure can be a rating curve in which a tabular relationship of discharge and head (or elevation) for the structure is prepared offline in advance by the user, then assigned to a weir or orifice by simply specifying that a rating curve is used. In this case, the model uses this rating curve to calculate the discharge at any time base on the dynamic head.

In general, a rating curve table can be used for any internal control structure to represent its flow-head relationship if there are no anticipated backwater effects. A single-valued-rating-curve can not be used in cases where there are backwater effects since the rating curves assumes no such backwater effects.

Culverts

Culverts are common hydraulic elements in a sewer system. There can be stand-alone culverts under highway embankments or conduit vaults in detention pond outlet structures. In SewerGEMS V8*i*, a culvert can be a conduit specified as a culvert or one of controlling elements in a composite control structure. Since a culvert is a type of hydraulic structure that transports water as full or partially full, culvert hydraulics is more complicated than other control structures.

Hydraulically a culvert can be under inlet control or outlet control conditions. The computational procedures for these conditions are very different:

- **Inlet control** - A culvert is under inlet control if the culvert barrel hydraulic capacity is higher than that of the inlet (entrance) and there is no backwater from downstream. In this condition, the relationship of flow and headwater is mainly dependent on the inlet configurations.
- **Outlet control** - A culvert is under outlet control when the culvert barrel is not capable of conveying as much flow as the inlet opening will accept. When the culvert is under outlet control, the flow will depend not only on the headwater but also the tailwater.
- **EQT curves** - Dynamic culvert conditions are complicated in that the flow can change from inlet control to outlet control or vice versa. As a result of this complexity, the computation of culverts can be tedious. In SewerGEMS V8*i*, a sophisticated procedure has been developed to build up a comprehensive EQT data set for any culvert configuration. The EQT represents the headwater (E), flow (Q), and tailwater (T) tabular curves in the way it covers all possible operating ranges of the headwater and tailwater so that any hydraulic conditions are accounted for by the EQT. The SewerGEMS V8*i* dynamic engine builds the EQT for every culvert and uses the EQT for culvert computation dynamically at any time step.

- “Manholes and Sewer Junctions” on page 14-746

Dynamic Storage Routing

Pond outflow is determined by the types of pond outlet structures which are associated with a given pond. The pond outlet can simply represent a connection point between the pond and a downstream conduit, or a more complex composite outlet structure.

The composite outlet structure at a pond outlet is a parallel set of outlet components which empty into the pond outlet's downstream link.

Composite outlets consist of the following types of structures:

- “Riser Structures” on page 14-755
- “Orifices” on page 14-757
- “Weirs” on page 14-759

Riser Structures

Risers are represented as a single opening at some elevation above the invert of the pond. The flow from the riser is then controlled by the flow through the downstream conduit of the pond outlet with which the riser is associated.

A riser can be represented as either a stand pipe or a inlet box. The only distinction between the two is essentially the open area and perimeter of the opening. In other words, the area and perimeter for a stand pipe are determined from the input diameter, while the area and perimeter for an inlet box are input directly.

Flow Stages on a Riser

As water rises in a pond the riser structure will exhibit three distinct flow stages:

- “Weir Stage” on page 14-755
- “Orifice Stage” on page 14-756
- “Full Riser Barrel Flow Stage” on page 14-756

Weir Stage

As the pond stage begins to go over the riser crest elevation, flow into the riser acts like a weir with the perimeter of the opening being the weir length. The following equation dictates the flow into the riser for low pond stages relative to the crest elevation.

$$Q = C * L * H^{3/2} \tag{14.17}$$

Where	C	=	weir coefficient (US, SI forms)
	L	=	effective weir length
	H	=	depth of flow at the standpipe crest (ft, m)

Orifice Stage

As the pond stage rises relative the crest elevation, the riser will then act like an orifice, and the flow is defined by the following equation.

$$Q = C * A * \sqrt{(2gH)} \tag{14.18}$$

Where	C	=	contraction and energy loss coefficient
	A	=	effective orifice area (sq. ft, sq. m)
	g	=	acceleration due to gravity
	H	=	orifice head (ft, m)

Note: For the type of orifices found in most ponds, C = 0.6.

A Note on Weir to Orifice Transition

The transition between the weir and orifice flow hydraulics is a turbulent transition which is computationally abrupt. To enhance convergence characteristics, Bentley SewerGEMS V8*i* supports the formulation of a weir to orifice transition zone. You can specify a hydraulic transition zone height which is (by default) centered about the theoretical transition point. Over this transition range, Bentley SewerGEMS V8*i* will linearly interpolate the stand pipe flow between the lower transition elevation at which weir flow governs to the upper transition elevation at which orifice flow governs.

Full Riser Barrel Flow Stage

Whenever the downstream conduit is undersized with respect to the standpipe capacity, full riser barrel flow will occur if the pond water surface elevation rises high enough. In these cases, the program assumes a negligible loss through the riser barrel and sets the riser flow equal to the downstream conduit flow rate.

If there are other orifices (perforations), slots (weirs), etc, flowing into the riser or inlet box, their flow rates are set equal to zero since the upstream elevation (pond water surface) and downstream elevation (inlet box headwater elevation) are identical (i.e., drop in head equals zero across these elements).

A Note on Perforations and Slots in Risers

If components of a particular composite outlet structure contains a riser structure in addition to orifice and weirs with elevations lower then the crest elevation riser, then Bentley SewerGEMS V8*i* treats the orifices and weirs as perforations and slots in the riser structure, and calculates the overall composite structure accordingly.

Orifices

There are two types of orifices that are associated with a pond outlet's complex outlet structure:

- Circular
- Orifice area

Both structures are defined by behaviors when submerged and unsubmerged.

Submerged Orifice Hydraulics

When the orifice is submerged, the flow is defined by the following equation for both orifice types:

$$Q = C * A * \sqrt{(2gH)} \tag{14.19}$$

Where	C	=	contraction and energy loss coefficient
	A	=	effective orifice area (sq. ft, sq. m)
	g	=	acceleration due to gravity
	H	=	orifice head (ft, m)

The orifice head, *H*, is measured as the difference between the water surface elevation and the greater of the center elevation of the circular orifice or the controlling tail-water elevation.

By inspection it can be seen that the equation is mathematically invalid whenever *H* is less than zero (i.e., the water surface is below the centroid during unsubmerged conditions).

Also, for the equation to be applied correctly, assume that the flow area must be fully submerged.

Circular Unsubmerged Hydraulics

To develop a continuous discharge rating relation for an orifice structure, it is necessary to handle flow situations in which the orifice opening is not fully submerged.

For circular orifices, Bentley SewerGEMS V8*i* models partially submerged orifices by balancing specific energy across the culvert opening. This is implemented in the program by assuming a thin culvert (L = 0.002 ft). This approach makes friction conditions negligible. The inlet loss, Ke, is calibrated so that it matches the results of the orifice flow at the "just submerged" elevation.

Orifice Area Unsubmerged Hydraulics

To develop a continuous discharge rating relation for an orifice structure, it is necessary to handle flow situations in which the orifice opening is not fully submerged. For area-based orifice calculations, Bentley SewerGEMS V8*i* performs a straightline interpolation, setting the flow by multiplying the full flow, Q_t, (at unsubmerged head, H_t) by the ratio of actual H to H_t.

$$Q_u = Q_t * H_u / H_t \tag{14.20}$$

Where	Q_u	=	unsubmerged discharge
	Q_t	=	full discharge at H_t
	H_u	=	unsubmerged head
	H_t	=	height of the orifice opening

Heads are measured from the opening invert or from the controlling tailwater, whichever is greater.

Orifice Orientation

Bentley SewerGEMS V8*i* supports modeling area-based orifice openings which are aligned horizontally and vertically, expressed as oriented parallel or perpendicular to flow direction, respectively. Orifices which are oriented parallel to flow do not require a datum input (since it is assumed to be equal to the opening invert).

In Bentley SewerGEMS V8*i*, circular orifices are all oriented perpendicular to flow. To model an opening oriented parallel with flow, use the Orifice-Area option, or a Stand Pipe.

Weirs

Weirs associated with pond outlets can be one of three types:

- "Rectangular Weirs" on page 14-759
- "V-Notch Weirs" on page 14-760
- "Irregular Weirs" on page 14-760

Rectangular Weirs

In Bentley SewerGEMS V8*i*, a rectangular weir is characterized by two equations: suppressed and contracted.

Suppressed weirs prevent the contraction of the flow through the weir and hence the associated losses. These types of weirs are usually, but not solely, associated with broad crested weirs, and are defined by the following equation:

$$Q = C * L * H^{3/2} \tag{14.21}$$

Where	Q	=	flow (cfs, cms)
	C	=	weir coefficient (US, SI forms)
	L	=	length (ft, m)
	H	=	head (ft, m)

Flow over a contracted weir does contract as it goes over the crest of the weir. These types of weirs are often associated with the sharp crested types of weirs, and are defined by the following equation:

$$Q = C * (L - 0.2H) * H^{3/2} \tag{14.22}$$

Where	Q	=	flow (cfs, cms)
	C	=	weir coefficient (US, SI forms)
	L	=	length (ft, m)
	H	=	head (ft, m)

V-Notch Weirs

V-Notch weirs are defined in Bentley SewerGEMS V8*i* by the following equation:

$$Q = C*(8/15)*\sqrt{(2g)}*\tan(\theta/2)*H^{5/2} \tag{14.23}$$

Where	Q	=	flow (cfs, cms)
	C	=	coefficient of discharge (US, SI forms)
	g	=	gravitational constant
	è	=	angle of notch (degrees)
	H	=	head above the bottom of the notch (ft, m)

H is measured from the water level to the bottom crest of the weir.

Irregular Weirs

Whenever the culvert headwater begins to rise above the minimum elevation of the roadway, overtopping will occur. The weir x-y structure can be used to model overtopping.

Overtopping flow is modeled as a special type of weir flow expressed by the general broad-crested weir equation.

Note: Do not use the irregular structure to model an overflow channel. The equations which define the irregular weir are different then channel equations and would result in significantly different flows.

Broad-Crested Weir

A broad-crested weir has a crest that extends horizontally in the direction of flow far enough to support the nappe (sheet of water flowing over the crest of the weir) so that hydrostatic pressures are fully developed for at least some short distance.

In order to model Embankment or Roadway overtopping, the Federal Highway Administration (FHWA) has developed a methodology that can be found in the manual FHWA, HDS No. 5, Hydraulic Design of Highway Culverts, 1985, which uses the general broad-crested weir equation.

$$Q = C_d L H_r^{3/2} \tag{14.24}$$

Where Q = Discharge over weir (m^3/sec., ft^3/sec.)

C_d = Weir coefficient

L = Length of roadway crest (m, ft)

H_r = Overtopping depth (m, ft)

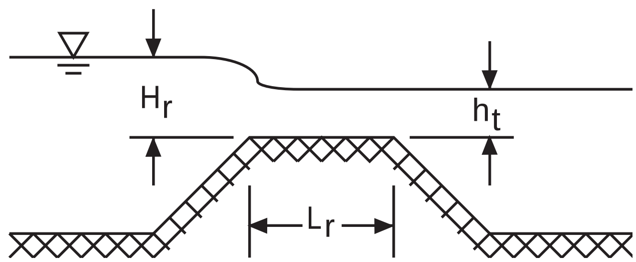


Figure 14-23: Broad-Crested Weir

The overtopping discharge coefficient C_d is a function of the submergence using the equation:

$$C_d = K_t C_r \tag{14.25}$$

The variables K_t and C_r are defined in the following figures, reproduced from the manual FHWA, HDS No.5, Hydraulic Design of Highway Culverts, 1985. The first two figures are used by Bentley SewerGEMS V8*i* to derive the base weir coefficient C_r resulting from deep and shallow overtopping, respectively. The submergence correction K_t is determined implicitly using the third figure.

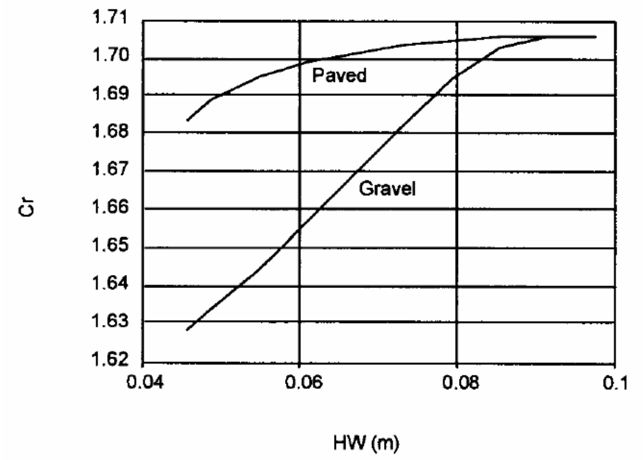


Figure 14-24: Discharge Coefficient C_r, for H_r/L > 0.15

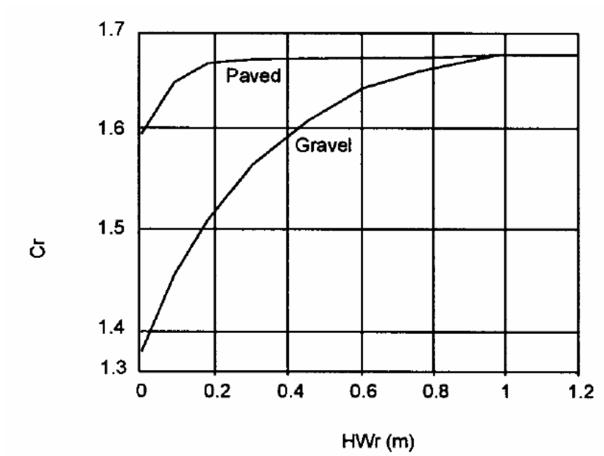


Figure 14-25: Discharge Coefficient C_r, for $H_r/L \leq 0.15$

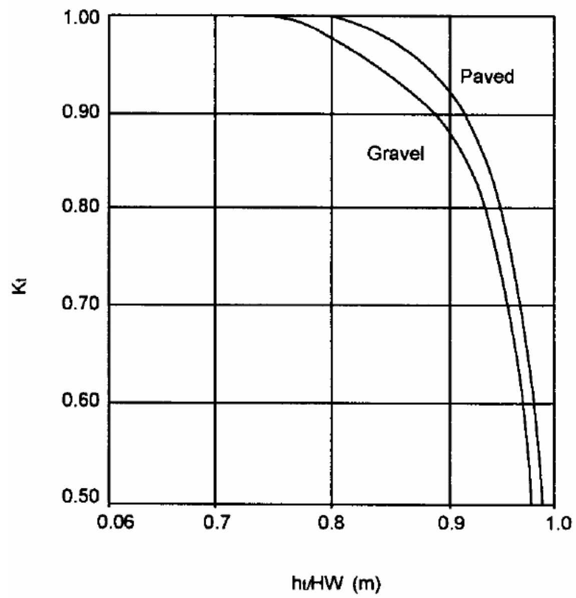


Figure 14-26: Submergence Factor, *k*

Pumps

Pump Station Configuration

A single pump station icon in the program represents a collection of individual pumps arranged in parallel with a single source element, which is defined at the pump. The pumps then can be staggered on and off based on the HGL/water surface elevation at the source node.

The source element can either be a pressure pipe or a source node such as a wet well, manhole, or pond, etc. Intuitively, if the source element for the pump station is a pipe, then the HGL of the node on the upstream end of the pipe will dictate the behavior of the pump.

Each pump in the pump station is defined by a pump definition (the pump curve), an initial status (on or off), and the conditions by which the pump turns on and off during the simulation.

Pump Definition Types

There are four types of pump definitions in Bentley SewerGEMS V8*i*. These are described below.

Type I - Volume Discharge Rating

This pump definition type is best suited for pumps which have either wet wells or ponds as the source element. The curve relates the volume of the source element to the outflow of the pump station. As the volume increases, the discharge increases.

Type II - Elevation Discharge Rating

This pump definition type simply relates the depth of flow of the source element to the outflow of the pump. As the depth increase, the discharge increases incrementally.

Type III -Declining Head Curve (Depth Discharge Rating

This is the most standard pump definition type. It relates the head difference between the upstream and downstream nodes to the discharge of the pump. As the head difference increases, the amount of discharge decreases.

Type IV - Variable Speed Pump

This pump definition type also relates the depth of the source node to the discharge of the pump. As the depth increases, the discharge increases continuously.

Storage Elements

This section describes how the following volume/storage elements in Bentley SewerGEMS V8*i* are defined:

- “Wet Wells” on page 14-764
- “Ponds” on page 14-765
- “Catch Basins, Manholes, and Surface Storage” on page 14-766

Wet Wells

The Wet Well volume can be determined by one of three ways:

- Depth-Area curve
- Constant area
- Area function

Depth-Area Curve

This option allows for the modeling of an irregular shaped volume associated with the wet well. The curve is then translated to volumes using conic sections.

Constant Area

Sets up the volume using with a constant cross sectional area. The volume is analogous to a cylinder.

Area Function

The Area is determined based on the following function which calculates the surface area for a given depth.

$$Area = Coeff * DepthExp + Constant \tag{14.26}$$

Where	*Area*	=	surface Area at given depth
	Coeff	=	user input value which is derived from existing area data
	Depth	=	distance from the invert of the pond
	Exp	=	user input value which is derived from existing area data
	Constant	=	the area at the bottom of the pond and is a user input value

Ponds

Pond volumes are defined one of four ways:

- Elevation-Area Curve
- Elevation-Volume Curve
- Functional
- Pipe Volume

Elevation-Area Curve

Volumes are typically defined as a series of Elevation-Area points, which are easily pulled from the contour map. The simulation then computes the volumes based on the changes in area between two elevations.

Elevation-Volume Curve

This option defines the volume directly by a series of elevation volume points. This allows for more complex storage structures that don't lend themselves to an Elevation-Area curve. If for example you have a fill, or obstructions in the pond you can enter the volume directly without having to work out adjustments to the areas.

Functional

The volume is determined based on the following function which calculates the surface area for a given depth.

$$Area = Coeff * DepthExp + Constant \tag{14.27}$$

Where	*Area*	=	Surface Area at given depth
	Coeff	=	User input value which is derived from existing volume data
	Depth	=	Distance from the invert of the pond
	Exp	=	User input value which is derived from existing volume data
	Constant	=	The area at the bottom of the pond and is a user input value

Pipe Volume

The Pipe Volume option supports modeling horizontal, vertical, or sloped pipes. Typically, the upsized pipes are significantly larger than would be required to simply convey the runoff from the site. For this reason upsized pipes will be terminated by an orifice or small diameter pipe stub which will provide the necessary peak discharge control.

The Pipe option automatically generates the cumulative volume rating table needed for the simulation. It should be emphasized that in upsized pipe systems the assumption is that the water surface elevation in the upsized pipe is taken to be level. This means that inflow into the upstream end of the pipe is immediately translated to the downstream end of the pipe - the standard detention routing assumption.

Catch Basins, Manholes, and Surface Storage

In addition to the negligible volumes assigned to manholes and catch basin structures, you can define the surface storage volumes above the structure's rim elevation.

The surface storage can be defined one of four ways:

- **No Storage -** When this option is selected the HGL at the element is determined based solely on hydraulics.
- **Default Storage Equation -** The surface volume above the rim is automatically established by the engine by extrapolating from the rim elevation.
- **Area (Constant Surface) -** The volume above the rim elevation is based on a volume with a constant surface area.
- **Surface Depth-Area Curve -** The surface volume is defined by a depth-area curve where the volume is determined with conic sections.

Surface (Gutter) System

Storm sewer systems are typically designed and constructed for smaller, more frequent storms. Runoff from large, less frequent events is usually not entirely conveyed by storm sewers; rather, it flows over the land surface in roadways and in natural and constructed open channels. Therefore storm sewer conveyance networks and surface gutter drainage and conveyance networks are integrated into a whole urban storm sewer infrastructure system. SewerGEMS V8*i* is capable of modeling a complete integrated subsurface storm sewer and surface gutter (channel) drainage system.

Gutter System Hydraulics

Stormwater from runoff enters the subsurface sewer conveyance system through catch basin inlets in roadway gutters, parking lots, depressions, ditches, and other locations, and often not all runoff water from the catch basin enters the inlet and additional water flows in gutters further downstream. There are a few hydraulic aspects to be considered in order to properly model the catch basin-inlet-gutter subsystem:

- Inlets are designed to have certain drainage capacities, and these capacities play an important role in the interaction between sewer subsystems and gutter subsystems. There are well-established design procedures to design inlets based on the design storm event. Once an inlet is set with specific dimensions, its capacity or hydraulic performance is known. In a SewerGEMS V8*i* model, this would be a user input. It can be an inlet capacity rating curve, in which a tabular relationship between total catch basin drainage flow and the inlet captured flow is presented, or a maximum inlet capacity flow amount. The model dynamically determines the inlet flow.

- When the inlet capacity is set, the excess water above its capacity will flow in the gutter to a downstream point. The gutter can also represent an open channel. SewerGEMS V8*i* lets the user specify the gutter cross section just like an open channel; it can be a trapezoidal or generic irregular section, and the user would also specify its Manning's friction coefficient.

Fundamental Solution of the Gutter System

The SewerGEMS V8*i* model simulates the gutter subsurface flows using diffusion routing algorithms. A nonlinear Muskingum-Cunge routing method is used to route the flows in gutters and the Manning's equation is used to compute water depths in the gutters.

An inlet receives both runoff flow from the catch basin and flows from gutters. Since it is an open pathway to subsurface sewers, it is possible that the subsurface sewers can become pressurized and as the overloaded flow increases and sewer water elevation rises above the inlet elevation so that "street flood" or "overflow" occurs in which water flows from the subsurface sewer to the ground through the manhole and the inlet. Under this condition, the water also finds its way to gutters and flows downstream if there is a gutter connected to the catch basin. This reverse interaction between subsurface sewer and surface gutter is also properly modeled by SewerGEMS V8*i* model. Therefore a gutter can carry excessive flow from an inlet or overflow from a catch basin.

There is a difference between a gutter as a surface drainage network and an open channel as part of a sewer network in a SewerGEMS V8*i* model. A gutter (or channel) in a surface network is always associated with a catch basin inlet and the main source of its flow comes from the excess water of the inlet or the overflow from the overcharged sewer catch basin. On the other hand, an open channel can be a part of the subsurface sewer system and a channel can be directly linked to a conduit.

Hydrology

Hydrologic theory gives you a behind-the-scenes look at what the SewerGEMS V8*i* software is using to manipulate the data you enter.

- "Rainfall" on page 14-769
- "Time of Concentration" on page 14-787
- "Rational Method" on page 14-794

- "SCS CN Runoff Equation" on page 14-797
- "SCS Peak Discharge" on page 14-803
- "Hydrograph Methods" on page 14-807

Rainfall

SewerGEMS V8*i* considers rainfall in terms of:

- "Design Storms" on page 14-769
- "I-D-F Data" on page 14-770
- "Rainfall Curves" on page 14-773

Design Storms

SewerGEMS V8*i* design storms include:

- Rational design storms
- Cumulative rainfall curve storms

Rational Design Storms

Design storms for use with the Rational method can be created with one of two methods.

- The I-D-F table method uses a table of duration versus intensity values to describe rainfall events of a particular frequency (return period).
- The e, b, d coefficients method uses a collection of three coefficients (e, b, d) to define a mathematical relationship between the rainfall intensity and the duration of the rainfall event for a given frequency.

Both methods yield the equivalent of a rainfall I-D-F curve, and therefore must be created for use in a particular geographic location.

Cumulative Rainfall Curve Storms

Hydrograph methods, such as the SCS Unit Hydrograph procedure, cannot use I-D-F curves for rainfall data (as used in the Rational method). Instead, complex hydrograph methods require time-based rainfall curves. Design storms for use with the hydrograph methods (e.g., SCS Unit Hydrograph) can be created with one of two methods: time-depth or synthetic.

Time-depth:	The time-depth curve method uses a table of time versus rainfall depth values to describe the rainfall event. This method is typically used when gauged data from actual storm events is available.
Synthetic:	The synthetic curve method uses a table of time versus rainfall depth fraction values, a duration multiplier, and a total rainfall depth to describe the rainfall event. This arrangement is very flexible because the same rainfall event shape can be used for storms of various durations and total depth.

I-D-F Data

Intensity-duration-frequency (I-D-F) data includes:

- "I-D-F Curves" on page 14-770
- "I-D-F Tables" on page 14-772
- "I-D-F e, b, d Equation" on page 14-772

I-D-F Curves

Note: The rainfall intensities that are used with the Rational method are generally determined by regulatory agencies. Historical rainfall information is analyzed and compiled into I-D-F curves based on the frequency of the storm events. These curves give the engineer a quick reference to determine the intensity of rainfall that occurs at given return periods.

I-D-F (Intensity-Duration-Frequency) curves provide the engineer with a way of determining the rainfall intensity for a given storm frequency and duration.

Reading an I-D-F Curve

For example, a 5-year frequency, the resulting average intensity is 5 inches an hour for 12 minutes. In other words, if an average intensity of 5 inches/hour falls for a period lasting 12 minutes, it would be considered a 5-year event.

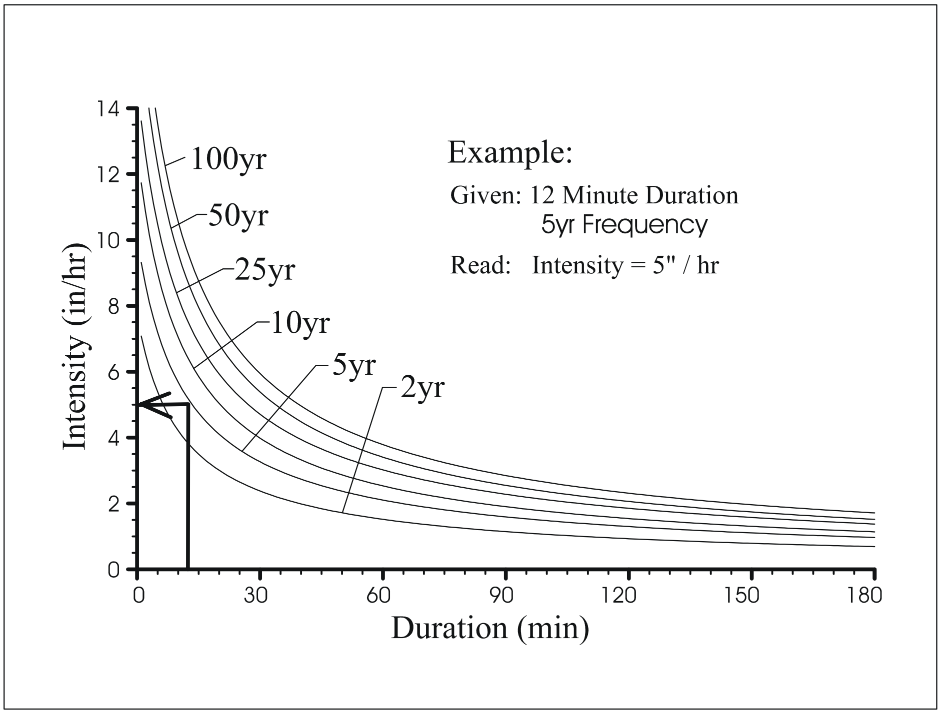
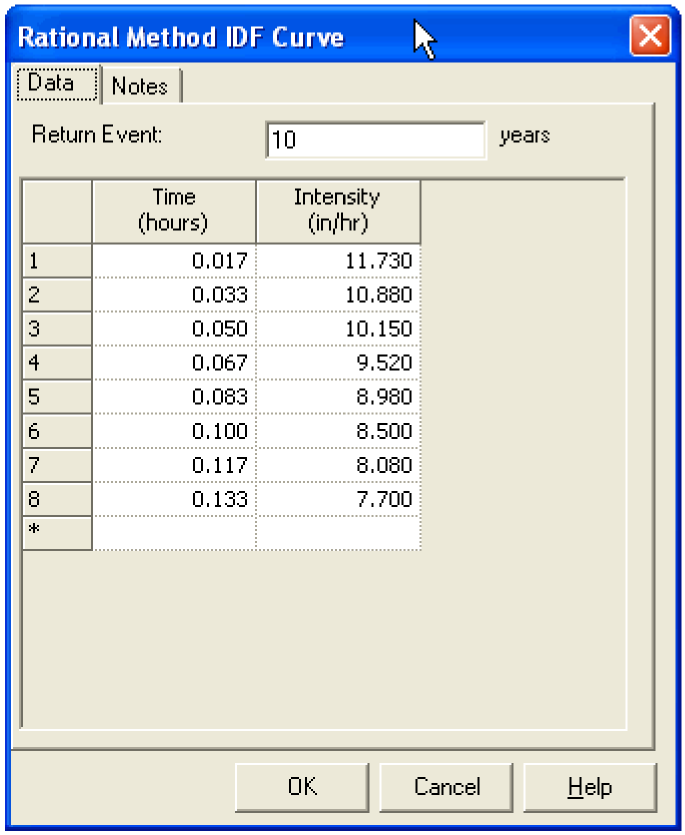


Figure 14-27: I-D-F Curve

I-D-F Tables

SewerGEMS V8*i* lets you enter I-D-F data into a table and saves the data so you may use it again for other projects. Entering the design intensities is a very simple process of looking up data from a graph and entering it into the I-D-F Table.



I-D-F e, b, d Equation

I-D-F curves can be fit to equations. The most common form of these equations is:

$$i = \frac{b}{(T+d)^e} \tag{14.28}$$

Where	I	=	rainfall intensity (in/hr.)
	T	=	rainfall duration (min.)
	e, b, d	=	rainfall equation coefficients

This equation represents the mathematical relationship between the rainfall intensity and the rainfall duration for a storm of a given frequency and a given geographical location. The rainfall equation coefficients vary with storm event frequency and storm event location.

To use rainfall equations properly requires that they yield results that are consistent with the historical rainfall data for the design locale. If the preceding equation does not provide such consistency, then it is not appropriate for your design.

Rainfall Curves

Rainfall curves fall into two categories:

- "Gauged (Time versus Depth)" on page 14-773
- "Synthetic Rainfall Distributions" on page 14-777

Gauged (Time versus Depth)

A rainfall curve is the measure of total rainfall depth as it varies throughout a gauged storm. A good way to understand a rainfall curve is to visualize the Y-axis as a rainfall gauge (see "Gauged Rainfall Event" on page 14-774). As the storm progresses, the gauge begins to fill. The curve describes the gauged rainfall depth at each point during the storm.

A steeper slope on the curve indicates the gauge is filling faster than it would for a less-steep curve; hence, the rate of rainfall is more intense. The most intense portion of the storm occurs between 0.1 and 0.2 hours and again between 0.5 and 0.6 hours (about 0.6 inches over 0.1 hour = 6 inches-per-hour intensity).

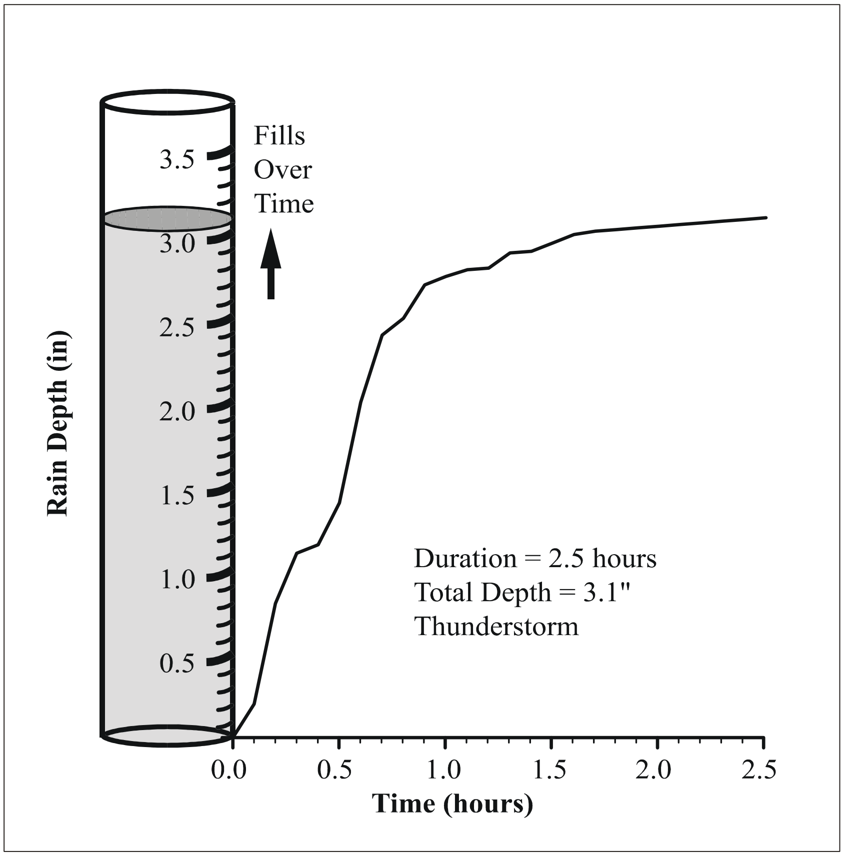


Figure 14-28: Gauged Rainfall Event

Rainfall curves are a mathematical means for simulating different storms. The next figure, “Conditions for Two Storms” on page 14-775, shows conditions for two types of storms. The other two (“Comparison of Two Storms” on page 14-775 and “Hydrographs for Two Storms” on page 14-776)display dramatic differences between these two rainfall events, even though the total depth and volume are the same for each storm.

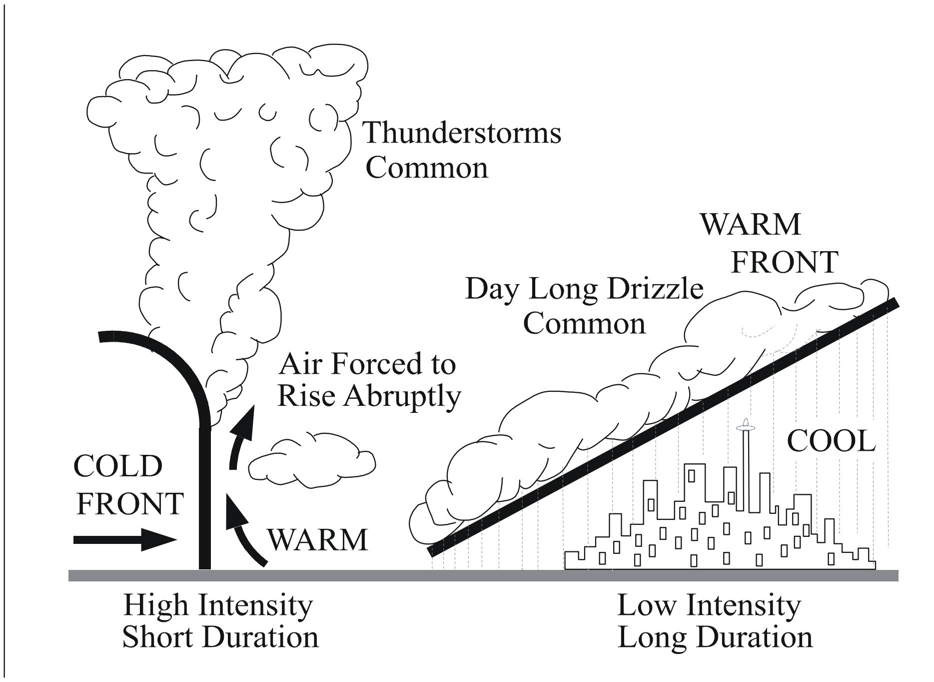


Figure 14-29: Conditions for Two Storms

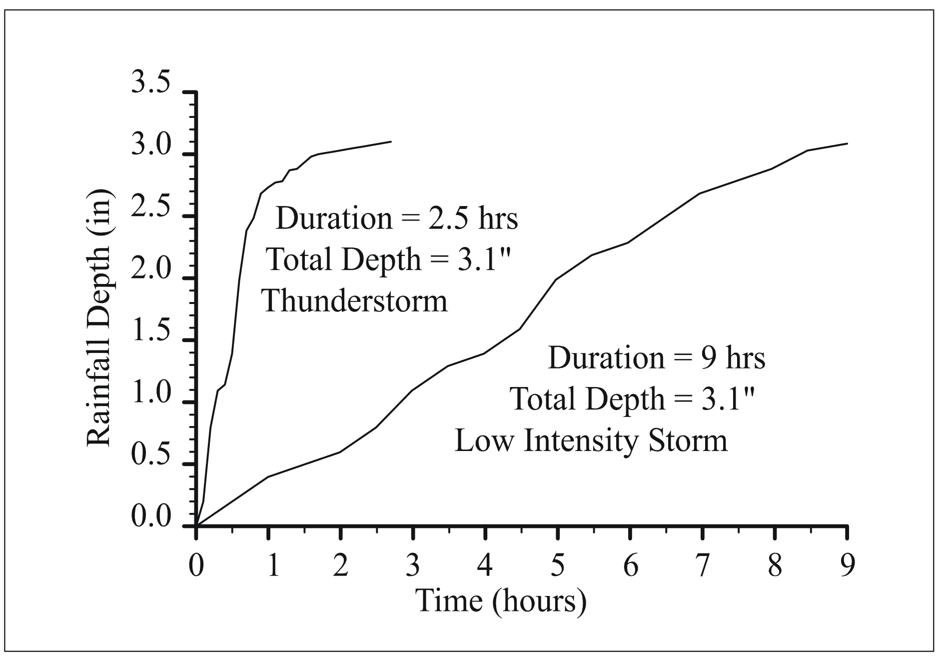


Figure 14-30: Comparison of Two Storms

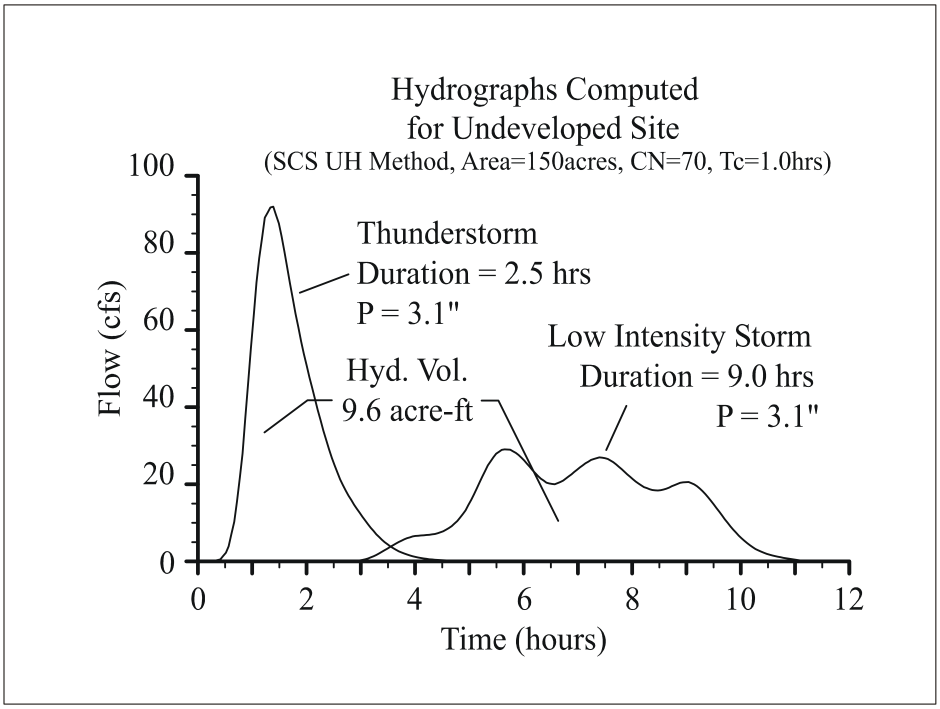


Figure 14-31: Hydrographs for Two Storms

Rainfall Tables

Rainfall hydrographs can be represented by tables. The table relates the cumulative rainfall depth to the time from the beginning of a storm. The following table is an example of a time versus depth rainfall table developed from data taken from a recording rain gauge.

Table 14-1: Time versus Depth

Time (hr.)	Accumulated Rainfall (in)
0.0	0.00
0.3	0.37
0.6	0.87
0.9	1.40
1.2	1.89

Table 14-1: Time versus Depth (Cont'd)

Time (hr.)	Accumulated Rainfall (in)
1.5	2.24
1.8	2.48
2.1	2.63
2.4	2.70
2.7	2.70
3.0	2.70
3.3	2.71
3.6	2.77
3.9	2.91
4.2	3.20
4.5	3.62
4.8	4.08
5.1x	4.43
5.4	4.70
5.7	4.90
6.0	5.00

Synthetic Rainfall Distributions

In most cases, drainage engineers design facilities for future rainfall events (not actual gauged storms). Rainfall distributions provide a way to model statistically predicted events of various magnitudes. These distributions are sometimes referred to as synthetic storms, since they are not actual gauged events.

Rainfall distributions fall into two categories:

- **Dimensionless Depth**—The Y-axis for these distributions range from 0.0 to 1.0 (0% to 100%) of total rainfall depth. The total storm duration is defined on the X-axis, in units of time.
- **Dimensionless Depth and Time**—These are similar to dimensionless depth curves, except that the X-axis is also dimensionless.

Dimensionless Depth: SCS Distributions

The SCS 24-hr. rainfall distributions are classic examples of dimensionless depth rainfall distributions. The Y-axis is dimensionless so that different rainfall depths can be applied to the distributions to create rainfall curves for various storm magnitudes and geographic locations.

The following figure displays four SCS distributions used in the United States (Types I, IA, II, and III).

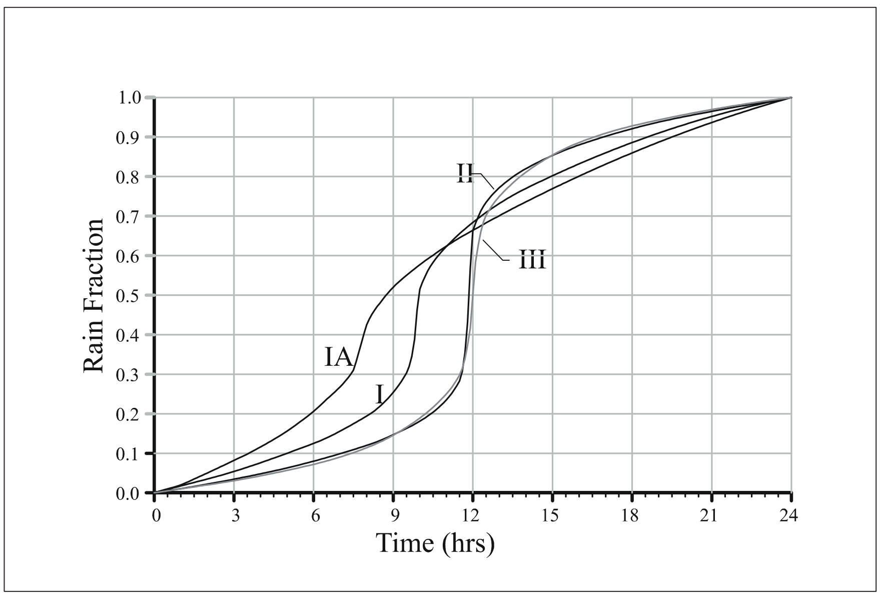


Figure 14-32: 24-Hour Rainfall Distributions

The approximate geographic boundaries for these rainfall distributions are shown below.

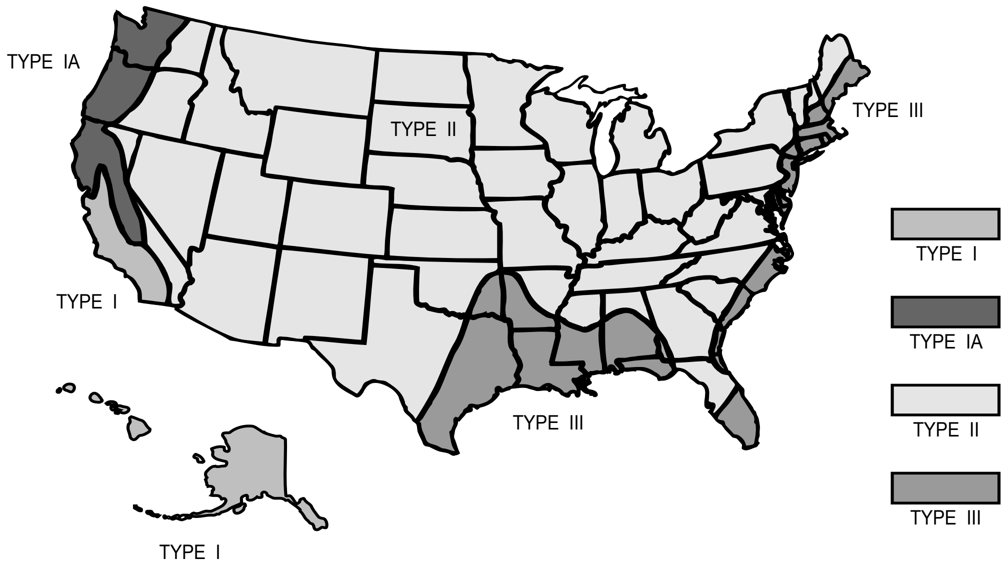


Figure 14-33: Approximate Boundaries

Modeling Storms with SCS Distributions

To create a design rainfall curve, multiply the Y-axis by the 24-hour total rainfall depth. The following figure shows what each distribution looks like when applied to a 24-hour total depth of 3.1 inches. Differences in storm magnitude and geographic variations can be modeled by changing the total rainfall depth on the Y-axis.

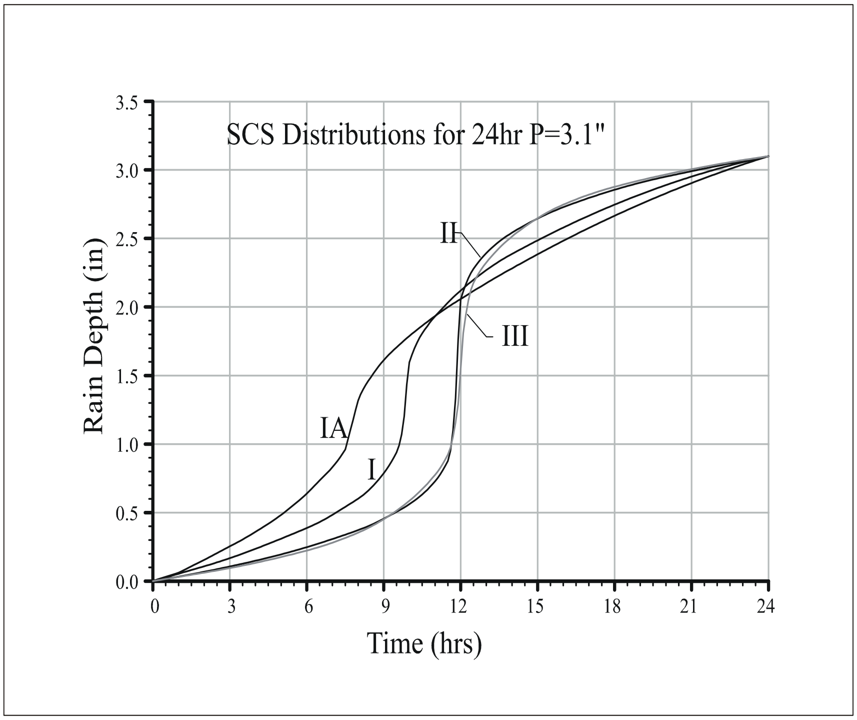


Figure 14-34: SCS Distribution, 24-Hour P = 3.1 in.

Dimensionless Depth and Time

These rainfall curve distributions are typically developed based on statistical analyses of storm events for different durations. When developed properly for a specific location, these types of rainfall distributions provide the flexibility of modeling a variety of storms other than the standard 24-hour event.

The basic philosophy of this approach is that longer-duration storms are expected to behave differently than shorter-duration storms. For example, the most intense portion of a 24-hour storm is expected to differ from the most intense portion of a 1-hour storm.

Typically, these types of curves are dimensionless on both the X and Y axes, so they can be applied to a wide range of durations and rainfall depths. The following graph displays dimensionless rainfall curves established for different ranges of durations. To create a rainfall depth curve, select the curve for the desired duration. Then, multiply the X-axis by total storm duration and multiply the Y-axis by the total rainfall depth for that given duration.

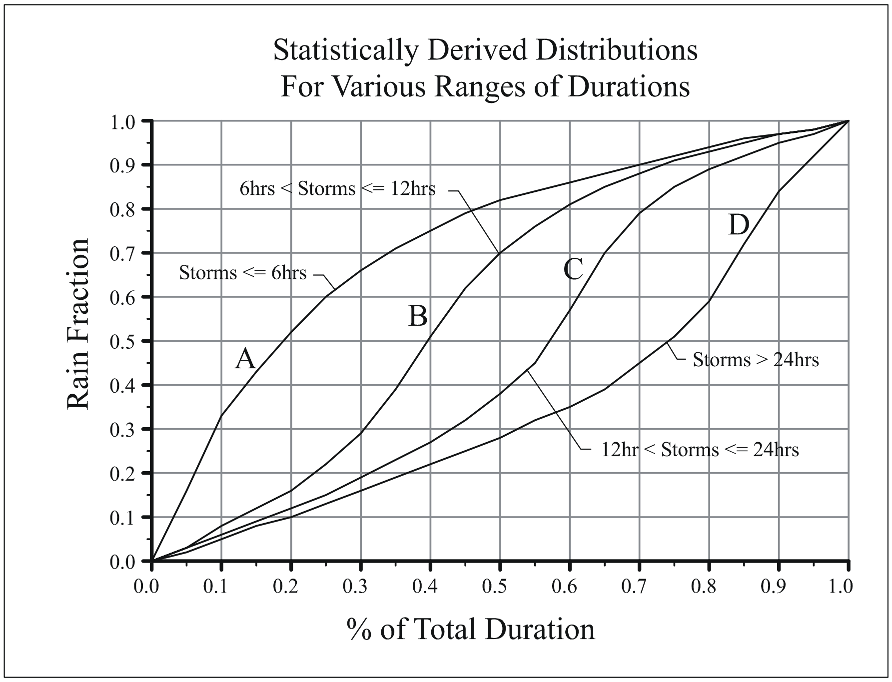


Figure 14-35: Dimensionless Time and Depth Curve

Example: Dimensionless Time and Depth Curves

Statistical analyses were performed using updated rainfall information for a certain geographic location in the United States. This study yielded the statistical distributions shown in the figure below.

Given: A 10 year return event for a certain area has the following total rainfall depths corresponding to various durations:

- 1 hr. P = 2.11 in.
- 6 hr. P = 3.37 in.
- 12 hr. P = 3.91 in.
- 18 hr. P = 4.13 in.

- 24 hr. P = 4.49 in.

Find:	Rainfall curves (time versus depth) for the 1-, 6-, 12-, 18-, and 24-hour durations, using the statistically derived distributions in the following figure.
Solution:	First, select the distribution that corresponds to each desired duration. Then multiply the Y-axis by the total rainfall depth for that duration and the X-axis by that duration.

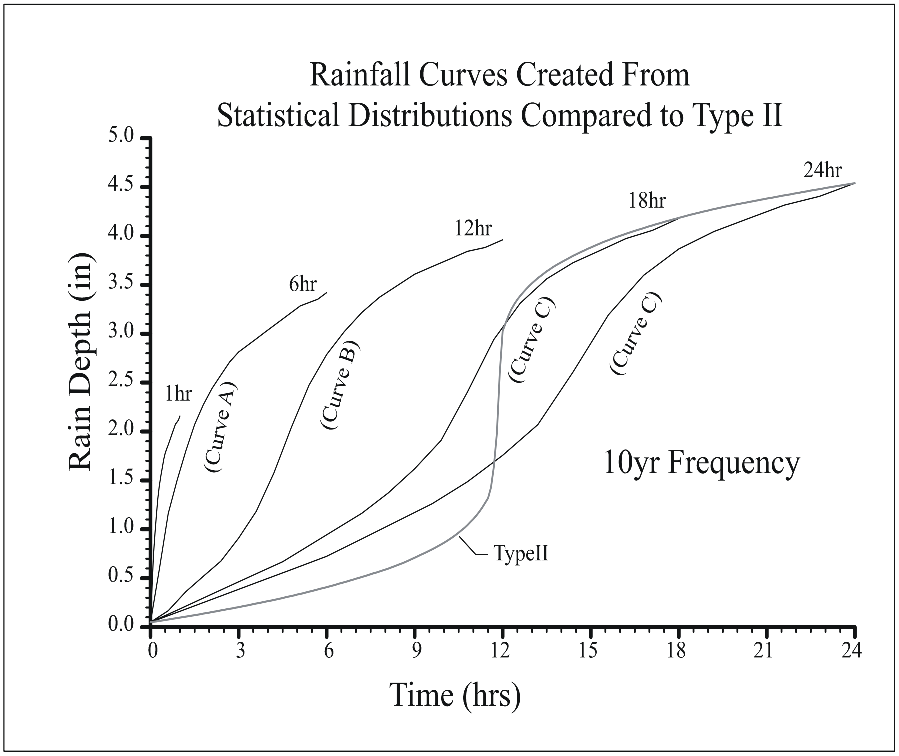


Figure 14-36: Solution to Example

The figure displays the results of this example. Different curve types (A, B and C from the previous graph) were used to model different duration storms. Note how the total depth increases, but overall intensity (slope of the curve) decreases as the duration is lengthened.

Synthetic Rainfall Tables

A synthetic rainfall curve is a plot of rainfall depth versus time that can be used in lieu of actual rainfall event data. A synthetic rainfall distribution is useful because it incorporates maximum rainfall intensities for a given event frequency arranged in a sequence that produces peak runoff. Therefore, a single rainfall distribution can be used to determine peak runoff rates for watersheds of various sizes and times of concentration.

Bulletins 70/71

The following sections describe the use of the data used in rainfall tables:

- "Rainfall Time-Distribution Information" on page 14-783
- "Watershed Area" on page 14-783
- "Rainfall Duration" on page 14-784
- "Data Sources" on page 14-784
- "Data Format" on page 14-785
- "Bulletin 70/71 Data" on page 14-786
- "Circular 173 Data" on page 14-786

Rainfall Time-Distribution Information

Illinois State Water Survey Bulletin 70 and *Bulletin 71* and *Circular 173* data contains synthetic rainfall time-distribution curves for heavy rainstorms in the Midwestern United States. This information comes from *Circular 173 (*Huff 1990*)*. Rainfall time-distribution curves are used for runoff computations related to the design and operation of runoff control structures.

Time-distribution curves are divided into four categories, corresponding to first-, second-, third- and fourth-quartile storms. Time distributions are represented as cumulative fractions of the storm rainfall depth and the storm duration.

The *Bulletin 70/71* data contains median (exceedance probability of 50%) curves. The *Circular 173* data gives the curves for exceedance probabilities of 10% and 90%.

Watershed Area

Time-distribution curves vary with the watershed area. Three time distribution types have been presented here depending on the watershed size:

- point distributions (area from 0 to 10 square miles)

- intermediate distributions (area from 10 to 50 square miles)
- area average distributions (area from 50 to 400 square miles)

The curves presented here are applicable only for relatively small watersheds (area less than or equal to 400 square miles).

Rainfall Duration

Storms with durations of 6 hours or less tend to be associated more often with first-quartile distributions, and those lasting more than 6 hours and less than or equal to 12 hours are most commonly the second-quartile type. Storms having durations longer than 12 and less than or equal to 24 hours most commonly follow the third-quartile distribution. Storms with a duration longer than 24 hours are most frequently associated with the fourth-quartile distributions. However, a particular storm from any duration may be associated with any of the four quartile types.

We recommend the use of the most common quartile for the design storms. A design storm with a duration less than or equal to 6 hours should be a first-quartile type storm. The second quartile type design storms should be used for durations longer than 6 and up to 12 hours. For storms longer than 12 hours in duration and less than or equal to 24 hours, we recommend the use of the third-quartile time distribution. Finally, design storms longer in duration than 24 hours should be modeled using the fourth quartile type.

Data Sources

The rainfall time-distribution data given here are obtained from *Circular 173* (Huff 1990). Wherever the tabular data was available in *Circular 173* it was used to develop rainfall tables. However, tabular data in *Circular 173* is given only for every 5% of the time distribution. The tables available in the engineering catalogs give data for every 1% of the rainfall time duration. The data in between tabular values have been obtained from the figures in *Circular 173*. Due to the interpolation procedure used to develop graphs, a slight discordance between tables and figures occurs in the tails of the distributions. Where this was the case, higher precedence was given to the tabular data.

Additional differences between the data presented here and the *Circular 173* tables comes from the precision used in *Circular 173*. While *Circular 173* rounds the data to the first 1%, the data presented in the Bentley SewerGEMS V8*i* engineering libraries has a precision of 0.01%. However, due to the statistical nature of the data presented, these differences are negligible.

Data Format

Data presented here is reported in dimensionless (fractional) distributions both in time and rainfall depth space. The temporal axes starts at 0.0 and ends at 1.0 with a time step of 0.01. Duration Multipliers should be used in SewerGEMS V8*i* to convert the dimensionless time to the desired rainstorm duration.

Quartile distributions are identified using the following notation:

- 1stQ—the first quartile time distribution
- 2ndQ—the second quartile time distribution
- 3rdQ—the third quartile time distribution
- 4thQ—the fourth quartile time distribution

Watershed area ranges are identified using the following notation:

- 00-10—point distributions (0 to 10 square miles)
- 10-50—intermediate distributions (10 to 50 square miles)
- 50-400—area distributions (50 to 400 square miles)

Exceedance probabilities are identified using the following notation:

- 50%—median distributions (for design storms)
- 10% - 90%—exceedance distributions (for analysis storms)
- 90% - 10%—exceedance distributions (for analysis storms)

Bulletin 70/71 Data

The *Bulletin 70/71* tables contain the median (exceedance probability of 50%) time distribution curves. Median distribution curves represent the most common rainfall types and should be used for design purposes. Median temporal distribution curves are given for point distributions, intermediate distributions, and areal distributions containing all four distribution types.

Circular 173 Data

The *Circular 173* (Huff 1990) tables contain the 10% and 90% exceedance probability time-distribution curves. These curves could be used for the analysis of extreme cases (what-if scenarios). These temporal distribution curves are given for point distributions and areal distributions, containing all four distribution types.

Rainfall Curves: Build from I-D-F Data

Intensity-Duration-Frequency (I-D-F) data can be used to build center peaking rainfall curves for any duration contained within that I-D-F curve.

The total rainfall depth is computed by multiplying the intensity corresponding to the desired storm duration and the duration. For example, the total depth for a 5 hour storm whose intensity (found from the I-D-F curve) is 0.469 in./hr. is 2.345 in. This total depth is then temporally distributed throughout the duration of the storm according to a center peaking pattern. The center peaking storm pattern dictates that the most intense portion of the storm is during the middle of the storm, and that the beginning and end of the storm are less intense.

Time of Concentration

The time of concentration (T_c) is found by summing the time for each individual flow segment within the drainage area. Both single and multiple flow segments are modeled with the T_c calculator.

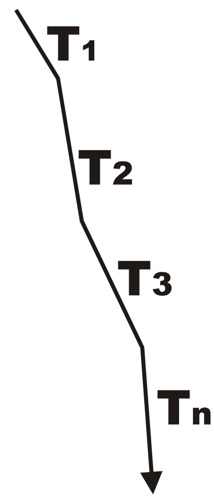


Figure 14-37: The General Equation for T_c

$$T_c = \sum_{n}^{i=1} T_i \tag{14.29}$$

Where	T_c	=	Total time of concentration
	T_i	=	Flow travel time through segment i

$$T_i = \frac{L_i}{V_i} \tag{14.30}$$

Where	L_i	=	Length of flow segment i
	V_i	=	Average velocity through segment i

The T_c equations provided in Bentley SewerGEMS V8*i* can be categorized into two broad categories:

- Equations that solve for velocity, then use velocity to solve for the travel time through a flow segment
- Equations that directly solve for the travel time through a flow segment—in these cases, Bentley SewerGEMS V8*i* back solves for velocity and includes it in the output report

Note: Some types of T_c equations can apply to flow segments within a multiple-segment T_c calculation (see preceding diagram). Other T_c methods are equations intended to model the entire average subarea flow distance and slope in one single flow segment. When combining multiple flow segments to compute T_c, it is up to you to only combine T_c methods that can be modeled in combination with multiple flow segments.

There are 13 different methods for computing the time for an individual flow segment. Each of the 13 methods has different data input requirements:

- "User-Defined" on page 14-789
- "Carter" on page 14-789
- "Eagleson" on page 14-789
- "Espey/Winslow" on page 14-790
- "Federal Aviation Agency" on page 14-790
- "Kerby/Hathaway" on page 14-790
- "Kirpich (PA)" on page 14-791
- "Kirpich (TN)" on page 14-791
- "Length and Velocity" on page 14-791
- "SCS Lag" on page 14-792
- "TR-55 Sheet Flow" on page 14-792
- "TR-55 Shallow Concentrated Flow" on page 14-793
- "TR-55 Channel Flow" on page 14-793

Minimum Time of Concentration

Certain hydrologic methods for computing runoff hydrographs require the time of concentration to be greater than some minimum value. For example the TR-55 methodology dictates that the minimum Tc to be used is 0.1 hr.

The minimum T_c is used in lieu of the calculated T_c whenever the calculated T_c is smaller than the minimum.

User-Defined

The user-defined time of concentration (T_c) is a method that allows the direct input of the T_c rather than using an equation to calculate it. This method would be used when the T_c needs to be calculated using a methodology that is not supported by Bentley SewerGEMS V8*i*, or when a quick estimate of T_c is sufficient for the analysis.

Carter

$$T_c = 1.7 L_m^{0.6} S_m^{\angle 0.3} \tag{14.31}$$

Where	T_c	=	Time of concentration (hr.)
	L_m	=	Flow length (mi)
	S_m	=	Slope (ft/mi)

Eagleson

$$T_c = 0.0001852 L_f n R^{\angle 2/3} S_f^{\angle 1/2} \tag{14.32}$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	n	=	Manning’s n
	R	=	Hydraulic radius (ft)
	S_f	=	Slope (ft/ft)

Espey/Winslow

$$T_c = 0.52\Phi L_f^{0.29} S_f^{-0.145} I_p^{-0.6} \tag{14.33}$$

Where	T_c	=	Time of concentration (hr.)
	ϕ	=	Espey Channelization factor
	L_f	=	Flow length (ft)
	S_f	=	Slope (ft/ft)
	I_p	=	Impervious area (%)

Federal Aviation Agency

$$T_c = 0.03(1.1 - C)L^{0.5}S^{-0.333} \tag{14.34}$$

Where	T_c	=	Time of concentration (hr.)
	C	=	Rational C coefficient
	L	=	Length of overland pipe flow (ft)
	S	=	Slope (%)

Kerby/Hathaway

$$T_c = 0.01377 L_f^{0.47} n^{0.47} S_f^{-0.235} \tag{14.35}$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	n	=	Manning's n
	S_f	=	Slope (ft/ft)

Kirpich (PA)

$$T_c = 0.00002167 L_f^{0.77} S_f^{\angle 0.5} M_t \tag{14.36}$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	S_f	=	Slope (ft/ft)
	M_t	=	T_c Multiplier (T_c adjustment)

Kirpich (TN)

$$T_c = 0.00013 L_f^{0.77} S_f^{\angle 0.385} M_t \tag{14.37}$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	S_f	=	Slope (ft/ft)
	M_t	=	T_c Multiplier (T_c adjustment)

Length and Velocity

$$T_c = \left(\frac{L_f}{V}\right)\left(\frac{1\,\text{hr.}}{3600\,\text{sec.}}\right) \tag{14.38}$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	V	=	Velocity (ft/sec.)

SCS Lag

Note: There is a factor of 0.6 built into this equation (in the constant 0.0000877) to convert this equation from a lag time to a time of concentration.

$$T_c = 0.0000877 L_f^{0.8}\left(\frac{1000}{\text{CN}} \angle 9\right)^{0.7} S_f^{\angle 0.5} \tag{14.39}$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	n	=	Manning's n
	S_f	=	Slope (ft/ft)

TR-55 Sheet Flow

This number represents the sheet flow time computed for each column of sheet flow data. Flow time for sheet flow is computed as:

$$T = \frac{0.007(nL)^{0.8}}{(P_2)^{0.5} S_f^{0.4}} \tag{14.40}$$

Where	T	=	Sheet flow time (hr.)
	n	=	Manning's roughness coefficient from TR-55 table
	L	=	Flow length (ft)
	P_2	=	Two-year, 24-hour rainfall (in)
	S_f	=	Slope (ft/ft)

TR-55 Shallow Concentrated Flow

This number represents the sheet flow time computed for each column of shallow concentrated flow data. Flow velocity for this flow time is computed as:

Unpaved Surfaces

$$V = 16.1345 S_f^{0.5} \quad (14.41)$$

Paved Surfaces

$$V = 20.3282 S_f^{0.5} \quad (14.42)$$

Where	V	=	Average velocity (ft/sec.)
	S_f	=	Slope of hydraulic grade line (ft/ft)

$$T_c = \left(\frac{L_f}{V}\right)\left(\frac{1\,\text{hr.}}{3600\,\text{sec.}}\right) \quad (14.43)$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	V	=	Average velocity (ft/sec.)

TR-55 Channel Flow

This number represents the channel flow time computed for each column of channel flow data. Flow velocity for this flow time is computed as:

$$T_c = \left(\frac{L_f}{V}\right)\left(\frac{1\,\text{hr.}}{3600\,\text{sec.}}\right) \quad (14.44)$$

where

$$V = \frac{1.49 R^{2/3} S_f^{1/2}}{n} \quad (14.45)$$

Where	T_c	=	Time of concentration (hr.)
	L_f	=	Flow length (ft)
	V	=	Average velocity (ft/sec.)
	R	=	Hydraulic radius (ft)
	S_f	=	Average slope (ft/ft)
	n	=	Manning's roughness value

Rational Method

The Rational method solves for peak discharge based on watershed area, Rational coefficient, and rainfall intensity for the watershed. The following equation is used to compute flow using the Rational method:

$$Q = CiA \tag{14.46}$$

Where	Q	=	Flow (cfs) for drainage area A
	C	=	Weighted runoff coefficient for drainage area A
	i	=	Intensity (in/hr.) for the given design frequency and storm duration (this value is taken from the I-D-F curves for your design area)
	A	=	Drainage area (acres)

Note: A conversion factor of 1.008 acre inches/hour per cfs makes the Rational equation unit-consistent, and is used by PondPack.

Some localities have C adjustment factors for different storm frequencies.

C, the Rational coefficient, is the parameter that is most open to engineering judgement. In many cases, an area weighted average of C coefficients is used as the C for the entire drainage area. SewerGEMS V8*i* calculates the weighted C for drainage areas.

Weighting C Values

If the drainage area consists of more than one subarea, a weighted C value for the area must be computed. The weighted C for a drainage area is computed by dividing the sum of all subarea CAs by the total area, where CA is the subarea C value multiplied by the area of the subarea.

Example: An engineer wants to compute the weighted C value for the composite drainage area shown below. In this example the C values are not adjusted for storm frequencies.

Total Area, A = 2.6 + 3.4 + 1.2 = 7.2 acres

$$\frac{(2.6 \times 0.97) + (3.4 \times 0.47) + (1.2 \times 0.98)}{7.2} = 0.736 \approx 0.74 \qquad (14.47)$$

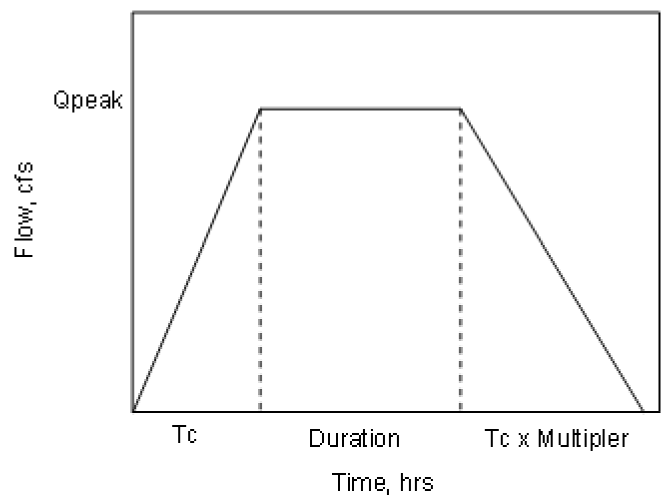
Q (cfs) is computed by:

$$Q = CiA \rightarrow Q = (0.74)i(7.2) \qquad (14.48)$$

Where	i	=	Rainfall intensity (in/hr.) for given design frequency and duration

Modified Rational Method

The Modified Rational Method provides a way to calculate the hydrograph from a catchment based on rational method C values and the peak intensity. There is no "loss method" associated with the modified rational method. The underlying assumption is that the peak intensity is maintained for a long enough duration to reach peak flow at the outlet of the catchment. This results in a trapezoidal hydrograph as shown below.



Qpeak is determined from the rational method (link to rational method topic)

Q = C i A

When using English units i is intensity in in/hr, A = area, acres, Q = flow, cfs and C is runoff coefficient, dimensionless.

The time to reach the peak is based on the time of concentration in the catchment which the user can manually enter or calculate using a variety of methods ("Rational Method" on page 14-794).

The length of the recession leg is based on the time of concentration times a recession multiplier which is set in the calculations options.

The intensity and duration are taken from the IDF curves (tables) based on the duration and frequency (return period) of the storm.

SCS CN Runoff Equation

The SCS Runoff equation is used with the SCS Unit Hydrograph method to turn rainfall into runoff. It is an empirical method that expresses how much runoff volume is generated by a certain volume of rainfall.

The variable input parameters of the equation are the rainfall amount for a given duration and the basin's runoff curve number (CN). For convenience, the runoff amount is typically referred to as a runoff volume even though it is expressed in units of depth (in., mm). In fact, this runoff depth is a normalized volume since it is generally distributed over a sub-basin or catchment area.

In hydrograph analysis the SCS runoff equation is applied against an incremental burst of rain to generate a runoff quantity. This runoff quantity is then distributed according to the unit hydrograph procedure, which ultimately develops the full runoff hydrograph.

The general form of the equation (U.S. customary units) is:

$$Q = \frac{(P \angle I_a)^2}{(P \angle I_a) + S} \tag{14.49}$$

Where	Q	=	Runoff depth (in)
	P	=	Rainfall (in)
	S	=	Maximum retention after runoff begins (in)
	I_a	=	Initial abstraction

The initial abstraction includes water captured by vegetation, depression storage, evaporation, and infiltration. For any P, this abstraction must be satisfied before any runoff is possible. The universal default for the initial abstraction is given by the equation:

$$I_a = 0.2S \tag{14.50}$$

The ratio, 0.2, is rarely, if ever, modified.

The potential maximum retention after runoff begins, S, is related to the soil and land use/vegetative cover characteristics of the watershed by the equation:

$$S = \frac{1000}{CN} \angle 10 \tag{14.51}$$

Where the runoff curve number is developed by coincidental tabulation of soil/land use extents in the weighted runoff curve number parameter, CN.

The Runoff Curve Number

In SewerGEMS V8*i*, the sub-basin runoff is defined solely by the CN input for each watershed. Bentley SewerGEMS V8*i* features built-in spreadsheet forms that aid you by automatically computing weighted CN values as a function of soil hydrologic class and cover characteristics.

The USDA has classified its soil types into four hydrologic soil groups. The CN values for various land uses and cover characteristics for each soil classification are described below. To describe a sub-basin using CN, you must overlay a land cover layer over a hydrologic soil mapping overlay and a delineated drainage basin mapping overlay. You then determine the component CN areas that comprise each sub-basin, and enter these into SewerGEMS V8*i*, which develops the actual weighted CN for use in hydrograph generation.

Definition of SCS Hydrologic Soil Groups

Group A:	Group A soils have low runoff potential and high infiltration rates even when thoroughly wetted. They consist chiefly of deep, well to excessively drained sands or gravels and have a high rate of water transmission (greater than 0.30 in./hr.).
Group B:	Group B soils have moderate infiltration rates when thoroughly wetted and consist chiefly of moderately deep to deep, moderately well to well drained soils with moderately fine to moderately course textures. These soils have a moderate rate of water transmission (0.15 to 0.30 in./hr.).
Group C:	Group C soils have low infiltration rates when thoroughly wetted and consist chiefly of soils with a layer that impedes downward movement of water and soils with moderately coarse textures. These soils have a moderate rate of water transmission (0.05-0.15 in./hr.).
Group D:	Group D soils have high runoff potential. They have very low infiltration rates when thoroughly wetted and consist chiefly of clay soils with a high swelling potential, soils with a permanent high water table, soils with a claypan or clay layer at or near the surface, and shallow soils over nearly impervious material. These soils have a very low rate of water transmission (0.00 to 0.05 in./hr.).

TR-55 provides an extensive table detailing different land uses, soil types and their associated CN values.

Runoff Volume (CN Method)

The amount of actual runoff from a watershed is dependent upon the amount of precipitation that occurs, the initial amount of precipitation that is intercepted, infiltrates, or is stored before runoff begins, the actual retention that occurs after rainfall begins, and the potential maximum retention that can occur after rainfall begins.

The SCS method for estimating the volume of direct runoff from storm rainfall relates the initial abstractions, and retention parameters to watershed properties as described by the curve number (CN).

The potential maximum retention after runoff occurs is related to the CN as follows:

$$S = \frac{1000}{CN} \angle 10 \qquad (14.52)$$

Where	S	=	Potential maximum retention after runoff begins
	CN	=	Curve number

The initial abstraction is related to the potential maximum retention as follows:

$$I_a = 0.2S \qquad (14.53)$$

Where	I_a	=	Initial abstraction (includes interception, surface storage, and infiltration)
	S	=	Potential maximum retention after runoff begins

The runoff volume is related to the precipitation and the potential maximum runoff as follows:

$$Q = \frac{(P \angle 0.2S)^2}{(P + 0.8S)} \qquad (14.54)$$

Where	Q	=	Actual runoff volume
	P	=	Rainfall (P >= Q)
	S	=	Potential maximum retention after runoff begins

For complex watersheds that consist of several subareas each having a distinct CN, the total actual runoff volume can be computed in two different ways.

- The cumulative volume method computes the actual runoff occurring from each subarea individually (using the individual CNs and areas), and then sums these runoff volumes to determine the total for the watershed.
- The composite volume method computes the actual runoff using a composite CN and the total watershed area.

CN Weighting

Note: Figures and tables referred to in this help section are referring to the TR-55 document. The tables are reproduced, see: "Reference Tables" on page B-927**.**

These sections are reproduced from *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b):

Antecedent Runoff Condition

The index of runoff potential before a storm event is the antecedent runoff condition (ARC). ARC is an attempt to account for the variation in CN at a site from storm to storm. The CN for the average ARC at a site is the median value as taken from sample rainfall and runoff data. For more information on the CNs for the average ARC, which is used primarily for design applications, see "Runoff Curve Number Tables" on page B-899. See the *National Engineering Handbook* (U.S. Soil Conservation Service 1969) and *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b) for more detailed discussion of storm-to-storm variation and a demonstration of upper and lower enveloping curves.

Urban Impervious area Modifications

Several factors, such as the percentage of impervious areas and the means of conveying runoff from impervious areas to the drainage system, should be considered in computing CN for urban areas (U.S. Soil Conservation Service 1986b). For example, do the impervious areas connect directly to the drainage system, or do they outlet onto lawns or other pervious areas where infiltration can occur?

Connected Impervious Areas

An impervious area is considered connected if runoff from it flows directly into the drainage system. It is also considered connected if runoff from it occurs as concentrated shallow flow that runs over a pervious area and then into a drainage system.

Urban CNs (for more information, see “Runoff Curve Numbers for Urban Areas” on page B-899) were developed for typical land use relationships based on specific assumed percentages of impervious area. These CN values were developed on the assumptions that:

- Pervious urban areas are equivalent to pasture, in good hydrologic conditions.
- Impervious areas have a CN of 98 and are directly connected to the drainage system. Some assumed percentages of impervious area are shown in “Table B-1: Runoff Curve Numbers for Urban Areas”on page B-899.

If all of the impervious area is directly connected to the drainage system, but the impervious area percentages or the pervious land use assumptions in “Table B-1: Runoff Curve Numbers for Urban Areas”on page B-899 are not applicable, use Figure 2-3 from *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b) to compute a composite CN. For example, “Table B-1: Runoff Curve Numbers for Urban Areas”on page B-899 gives a CN of 70 for a 1/2-acre lot in HSG B, with an assumed impervious area of 25 percent. However, if the lot has 20 percent impervious area and a pervious area CN of 61, the composite CN obtained from Figure 2-3 (U.S. Soil Conservation Service 1986b) is 68. The CN difference between 70 and 68 reflects the difference in percent impervious area.

Unconnected Impervious Areas

Runoff from these areas is spread over a pervious area as sheet flow. To determine CN when all or part of the impervious area is not directly connected to the drainage system, (1) use Figure 2-4 from *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b) if total impervious area is less than 30 percent, or (2) use Figure 2-3 (U.S. Soil Conservation Service 1986b) if the total impervious area is equal to or greater than 30 percent, because the absorptive capacity of the remaining pervious areas do not significantly affect runoff.

When impervious area is less than 30 percent, obtain the composite CN by entering the right half of Figure 2-4 (U.S. Soil Conservation Service 1986b) with the percentage of total impervious area and the ratio of total unconnected impervious area to total impervious area. Then move left to the appropriate pervious CN and read down to find the composite CN. For example, for a one acre lot with 20 percent total impervious area (75 percent of which is unconnected) and pervious CN of 61, the composite CN from Figure 2-4 (U.S. Soil Conservation Service 1986b) is 66. If all of the impervious area is connected, the resulting CN from Figure 2-3 (U.S. Soil Conservation Service 1986b) would be 68.

Equation for composite CN with connected impervious area:

$$CN_c = CN_p + \left(\frac{P_{imp}}{100}\right)(98 \angle CN_p) \tag{14.55}$$

Where CN_c = Composite runoff curve number

CN_p = Pervious runoff curve number

P_{imp} = Percent imperviousness

Equation for composite CN with unconnected impervious areas and total impervious area less than 30%:

$$CN_c = CN_p + \left(\frac{P_{imp}}{100}\right)(98 - CN_p)(1 - 0.5R) \tag{14.56}$$

Where R = Ratio of unconnected impervious area to total impervious area

SCS Peak Discharge

SCS Peak Discharge includes:

- "TR-55 Graphical Peak Discharge (SCS Graphical Peak)" on page 14-803
- "TR-55 Pond Storage Estimate (SCS Storage Estimate)" on page 14-806

TR-55 Graphical Peak Discharge (SCS Graphical Peak)

This option uses the Graphical Peak Discharge method to compute the peak discharge for up to three different storm events. The following information is required:

- The drainage area in acres (Bentley SewerGEMS V8*i* automatically converts it to sq. mi.)
- Amount of pond and swamp areas (percentage of total drainage areas)
- The 24-hr. precipitation (P) for the selected return period
- The appropriate rainfall distribution (Type I, IA, II, or III)
- The time of concentration, T_c
- The runoff curve number, CN

Initial Abstraction, Ia (in)

The initial abstraction is computed from the precipitation and CN number and inserted in this field. For more information, see I_a/P, and Chapter 2 of *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b).

Ia/P Ratio

The initial abstraction (I_a) is divided by the precipitation (P) and printed in this field. For more information on I_a/P, see *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b).

Unit Discharge, qu (csm/in.)

The unit discharge (q_u) for the watershed is computed and printed into this field. Graphs depicting the relationship between time of concentration (T_c), I_a/P, and q_u (csm) are displayed in Exhibit 4-I, 4-IA, 4-II, and 4-III in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b). These graphs are described in the equation below from Appendix F, in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b).

$$\log(q_u) = C_0 + C_1\log(T_c) + C_2[\log(T_c)]^2 \tag{14.57}$$

Where	q_u	=	Unit peak discharge (csm/in)
	T_c	=	Time of concentration (hr.) (minimum T_c = 0.10 hr., maximum T_c = 10.0 hr.)
	C0, C1, C2	=	Coefficients from Table F-1 in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b)

Bentley SewerGEMS V8*i* computes two q_u values by selecting C0, C1, C2 coefficients corresponding to the specified distribution type and the two I_a/P values that are closest to the computed I_a/P for the watershed. Bentley SewerGEMS V8*i* then linearly interpolates between the two computed q_u values to obtain the actual q_u used to compute peak discharge. If the watershed's computed I_a/P ratio exceeds the limits of Table F-1, the limiting value for I_a/P is used to compute q_u (csm).

Runoff, Q (in.)

Runoff (inches) is computed from the CN and precipitation (P). For more information, see Chapter 2 in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b).

Pond and Swamp Adjustment Factor

The pond and swamp adjustment factor (F_p) is selected from the values given in Table 4-2 in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b). If the value entered for pond and swamp areas does not exist in Table 4-2, the nearest adjustment factor (F_p) is used.

Peak Discharge, qp (cfs)

The peak discharge for a given storm is computed by the equation below, from Chapter 4 in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b). This value is computed from the watershed's area (sq. mi.), runoff (in.), and unit discharge (csm/in.).

$$q_p = q_u A_m Q F_p \tag{14.58}$$

Where	q_p	=	Peak discharge (cfs)
	q_u	=	Unit peak discharge (csm/in)
	A_m	=	Drainage area (mi^2)
	Q	=	Runoff (in)
	F_p	=	Pond and swamp adjustment factor

TR-55 Pond Storage Estimate (SCS Storage Estimate)

This option estimates storage requirements using the method discussed in Chapter 6 in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b). This is an extremely approximate method.

Information required:

1. Compute peak flow (q_i, cfs), using either the Graphical Peak Discharge method or the Tabular Hydrograph method. Do not use peak discharges computed with any other method. When using the Tabular Hydrograph method to compute q_i, only use peak discharge associated with $T_t = 0.0$.
2. The inflow runoff (Q, in); this parameter is computed for you whenever you use the Graphical Peak Discharge method with this estimation option. See Chapter 2 of *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b) for discussion on runoff, Q.
3. The peak outflow rate (q_o, cfs).

Theory for Computed Spreadsheet Values

q_o/q_i ratio:	This value is the peak outflow rate (q_o) divided by the peak inflow rate (q_i). This parameter is used to compute V_s/V_r.
V_s/V_r ratio:	This number represents the ratio of detention storage volume to inflow volume. Bentley SewerGEMS V8*i* uses the equation below to compute V_s/V_r (see also Appendix F, SCS TR-55 document). The coefficients C_0, C_1, C_2, and C_3 are selected for the specified distribution type.

$$V_s/V_r = C_0 + C_1(q_0/q_i)^2 + C_2(q_0/q_i)^2 + C_3(q_0/q_i)^3 \quad (14.59)$$

Where	V_s/V_r	=	Ratio of storage volume (V_s) to runoff volume (V_r)
	q_0/q_i	=	Ratio of peak outflow (q_0) to peak inflow (q_i)
	C_0, C_1, C_2, C_3	=	Coefficients from Table F-2, in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service 1986b)

Inflow Volume, V_r (ac-ft): The inflow volume represents the total runoff volume for the given inflow storm. It is computed by multiplying the watershed area by the runoff. With area in acres, runoff (Q) in inches, the runoff volume in acre-ft. would be computed by the equation below.

$$V_r = Area(\text{acres}) \times \frac{Q(\text{in})}{12(\text{in/ft})} \tag{14.60}$$

Storage Volume, V_s (acre-ft): This number is the computed value for the estimated detention storage that is required. It is computed by multiplying the inflow volume V_r by V_s/V_r as shown in the equation below.

$$V_s = V_r\frac{V_s}{V_r} \tag{14.61}$$

Hydrograph Methods

Hydrograph methods include:

- "Soil Conservation Service (SCS)" on page 14-809

Unit Hydrograph Methodology

The Unit Hydrograph theory assumes that the watershed is a linear system. This means that the outflow is proportional to the inflow regardless of the magnitude of the inflow. This is generally not the case however. When the flow in stream channels and on overland flow surfaces increases, the velocity also increases, causing a reduction in the time of travel to the outlet. Yet, for most natural streams, the velocity increases as the depth increases only until overbank flow begins. At this point, the velocity tends to remain constant, which satisfies the requirement of linearity. Therefore, unit hydrographs should be derived only from the larger floods for a particular watershed.

The Unit Hydrograph theory also assumes that the input rainfall excess is uniform over the watershed, and that the response to this input is invariable. Typically, the spatial variation of rainfall, and the difference in watershed characteristics can cause the rate of runoff to vary widely from place to place at any time. However, many watersheds do experience similar patterns of rainfall from event to event, and therefore the response to that rainfall excess can be effectively characterized by the unit hydrograph.

The unit hydrograph theory depends on the principle of superposition. This principle states that a flood hydrograph for a particular storm event can be built up from the unit hydrograph applied to the incremental rainfall excess during each period. In other words, the unit hydrograph can be applied to a series of inputs, and the resulting hydrographs can be added together to form the total hydrograph.

Generic Unit Hydrographs

You can directly associate a user-defined unit hydrograph to a catchment for runoff calculations. The unit hydrograph is a time versus flow curve which represents a one-inch volume of rainfall for a given excess duration of rainfall, DD, for a set watershed area.

DD is equivalent to the convolution time steps, which tell Bentley SewerGEMS V8*i* how to subdivide the entered unit hydrograph into equal steps and serves as the internal calculation increment.

For each plug of runoff generated over a single time step, an individual runoff hydrograph is generated. All the successive unit hydrographs are superimposed to form the ultimate runoff hydrograph for the catchment. The underlying theory is described in *Stormwater Conveyance Modeling and Design*, by Bentley Institue Press (pp. 158-162) or *Wastewater Collection System Modeling and Design*, by Bentley Institue Press (pp. 252-254).

The fundamental equation for this method is show below.

$$Q_k = \sum_{i=1}^{k} P_i U_{k-i+1} \tag{14.62}$$

Where	Q_k	=	flow at time step k, cfs
	P_i	=	precipitation during time step i, in./hr
	U_{k-i+1}	=	flow at time step k from rain during time step I, cfs/in
	k	=	duration of rain in time steps + duration of hydrograph

The theory behind unit hydrographs is that the volume of water, calculated as the area under the hydrograph curve, should correspond to 1 inch of excess precipitation over the area. The user needs to check if this is true.

For example, over a 2 acre area, the volume of water calculated under the unit hydrograph should be 2 acre-in (7,260 cubic feet). If there is 1.5 in of excess precipitation (precipitation – losses) over this catchment, the volume of water calculated using the unit hydrograph method should be 3 acre-in (10,890 cubic feet). This runoff volume is displayed under the Catchment tab of the Detailed Calculation Summary.

Soil Conservation Service (SCS)

This section includes:

- "SCS Unit Hydrograph" on page 14-809
- "Governing Equations" on page 14-809

SCS Unit Hydrograph

A unit hydrograph can be a natural hydrograph (i.e., a hydrograph obtained directly from observed flows in a gauged stream), or a synthetic hydrograph (i.e., a hydrograph that simulates a natural hydrograph by using watershed parameters and storm characteristics). Specifically, a unit hydrograph is the hydrograph that results from one inch of direct runoff occurring uniformly over a watershed in a specified amount of time. L.K. Sherman first advanced the theory of the unit hydrograph in 1932, and then Victor Mockus derived the unit hydrograph used by the SCS (from a large number of natural unit hydrographs from both large and small watersheds).

Governing Equations

The SCS Unit Hydrograph is an extremely flexible tool. It is a dimensionless curvilinear graph with ordinate values expressed in a dimensionless ratio Q/Qp (discharge at time t to total discharge) or Q_a/Q (accumulated volume at time t to total volume) and its abscissa values as T/Tp (a selected time from the beginning of the rise to the peak). A watershed specific unit hydrograph can be developed from the dimensionless graph utilizing certain watershed parameters. This watershed specific unit graph can then be used to build a flood hydrograph resulting from a given storm.

The unit hydrograph for any regularly shaped watershed can be constructed once the values of Q_p and T_p are defined. An irregularly shaped watershed should be divided into hydrological units of uniformly shaped areas, such that the drainage area of any unit should be less than 20 square miles and have a homogeneous drainage pattern. Unit hydrographs should be developed for each of the divisions, and then combined to form the watershed unit graph.

Excerpted from NEH-4: The dimensionless curvilinear unit hydrograph has 37.5% of the total volume in the rising side, which is represented by one unit of time and one unit of discharge. This dimensionless unit hydrograph also can be represented by an equivalent triangular hydrograph having the same units of time and discharge, thus having the same percent of volume in the rising side of the triangle.

Figure 14-38: Dimensionless Unit Hydrograph

The peak discharge for the hydrograph can be found from the following equation:

$$q_p = \frac{484AQ}{T_p} \tag{14.63}$$

Where	q_p	=	Peak discharge (cfs)
	A	=	Drainage area (mi^2)
	Q	=	Depth of runoff (in)
	T_p	=	Time to peak (hr.)
	484	=	645.33 x 0.75m, where 645.33 is a conversion factor from in x mi^2/hr. to ft^3/sec.

The time to peak is defined as:

$$T_p = \frac{\Delta D}{2} + L \tag{14.64}$$

Where	ΔD	=	Duration of unit excess rainfall (hr.)
	L	=	Watershed lag (hr.)

The average relationship of watershed lag to the time of concentration is:

$$L = 0.6T_c \tag{14.65}$$

Where	T_c	=	Time of concentration (hr.)

The duration of unit excess rainfall is related to the time of concentration as:

$$\Delta D = 0.133 T_c \tag{14.66}$$

Unit Hydrograph Runoff Methods

Bentley SewerGEMS V8*i* provides various methods for computing the incremental runoff for pervious and directly connected impervious areas to compute the weighted runoff depth for each hydrograph increment. The SCS Unit Hydrograph option can compute runoff using either of the following methods:

> **Note: In either case, you can use the storage depression method for computing runoff for the directly connected impervious area (i.e., the runoff for impervious areas equals the rainfall greater than a specified storage depression depth).**

1. You can enter a value in the **fLoss** field that represents a constant infiltration loss rate (in./hr.) that occurs throughout the entire duration of the storm.
2. You can use the **SCS Runoff Curve Number** method by entering CN values for the pervious and impervious areas.
3. You can use the **Green and Ampt** method and calculate a variable rate of absorption for your subarea.
4. You can use the **Horton** (generic) method to calculate non-constant infiltration for your subarea.

The following approach is used whenever CN values are entered for the pervious and impervious areas.

Pervious Area

If P(t) is less than $0.2S_p$ then $R_p(t) = 0.0$. Otherwise:

$$R_p(t) = \frac{[P(t) + 0.2S_p]^2}{[P(t) + 0.8S_p]} \tag{14.67}$$

Where	$R_p(t)$	=	Pervious area runoff (in) during time step t
	P(t)	=	Rainfall (in) during time step t
	S_p	=	$1000/CN_p - 10$
	CN_p	=	Pervious area runoff curve number

Directly Connected Impervious Area

If P(t) is less than 0.2Si then Ri(t) = 0.0. Otherwise:

$$R_i(t) = \frac{[P(t) \angle 0.2S_i]^2}{[P(t) + 0.8S_i]} \tag{14.68}$$

Where	$R_i(t)$	=	Impervious area runoff (in) during time step t
	P(t)	=	Rainfall (in) during time step t
	S_i	=	$1000/CN_i$ - 10
	CN_i	=	Impervious area runoff curve number

Depression Storage

If a value greater than 0 for depression storage depth is entered, the following method is used for computing Ri(t) (the SCS CN method is not used).

$R_i(t)$ = 0.0 when $P_a(t)$ is less than D_s. Otherwise:

$$_i(t) = [P_a(t) \angle D_s] \angle [P_a(t \angle 1) \angle D_s] = P_a(t) \angle P_a(t \angle 1 \tag{14.69}$$

Where	$R_i(t)$	=	Impervious area runoff (in) during time step t
	$P_a(t)$	=	Cumulative rainfall (in) through time step t
	D_s	=	Storage depression depth (in)

Weighted Incremental Runoff

$$R(t) = ([R_p(t) \times A_p] + [R_i(t) \times A_i])/(A_p + A_i) \tag{14.70}$$

Where $R_p(t)$ = Pervious area runoff (in) during time step t

A_p = Pervious area (acres)

$R_i(t)$ = Impervious area runoff (in) during time step t

A_i = Impervious area (acres)

fLoss Rate

Whenever you enter a value for **fLoss**, Bentley SewerGEMS V8*i* computes the runoff for each hydrograph time increment using an average uniform infiltration rate that does not vary with time or total depth.

Green and Ampt

While the Horton method was empirically derived to describe the exponential decay of infiltration rate over time, the **Green and Ampt** method is based on a theoretical derivation of Darcy's law, which relates flow velocity to the permeability of the soil and the Law of Conservation of Mass. The resulting equation inversely relates the infiltration rate *f* to the total accumulated infiltration *F* as:

$$f = K_s\left(\frac{\psi(\theta_s \angle \theta_i)}{F} + 1\right) \tag{14.71}$$

Where f = Infiltration rate (in/hr, cm/hr)

K_s = Saturated hydraulic conductivity (permeability) (in/hr, cm/hr)

Ψ = Capillary suction (in, cm)

θ_s = Volumetric moisture content (water volume ÷ unit soil volume) under saturated conditions

θ_i = Volumetric moisture content under initial conditions

F = Total accumulated infiltration (in, cm)

The benefit of the Green and Ampt method is that the infiltration rate can be calculated based on physical, measurable soil parameters, as opposed to the more amorphous decay coefficients of the Horton method.

In order to calculate the infiltration rate at a given time, the total infiltration up to that time must be calculated. This value can be determined by integrating the previous equation with respect to time (starting at time = 0) and solving for F.

$$F = K_s \cdot t + \psi(\theta_s - \theta_i)\ln\left(1 + \frac{F}{\psi(\theta_s - \theta_i)}\right) \tag{14.72}$$

Where t = time (hr.)

The equation cannot be explicitly solved, and thus requires the application of a numerical method such as Newton-Rhapson or the bisection method to solve for *F*.

Horton

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

$$f(t) = f_c + (f_0 - f_c)e^{-k(t - t_0)} \tag{14.73}$$

Where	f(t)	=	Infiltration rate (in/hr. or mm/hr.) at time t (min.)
	f_c	=	Steady-state infiltration rate that occurs for sufficiently large t
	f_0	=	Initial infiltration rate at the time that infiltration begins (that is, at time t = t0)
	k (min.-1)	=	Decay coefficient

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

Estimation of the parameters f_c, f_0, and k in the previous equation can be difficult because of the natural variabilities in antecedent moisture conditions and soil properties. The following table provides some values recommended by Rawls et al. (1976), though such tabulations should be used with caution. Singh (1992) recommends that f_0 be taken as roughly 5 times the value of f_c.

Table 14-2: Typical Values of Horton Infiltration Parameters

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

- (Source: Rawls et al. 1976.)

Often, the rainfall intensity during the early part of a storm is less than the potential infiltration capacity of the soil; thus, the supply of rainwater is a limiting factor on the infiltration rate. During the time period when the water supply is limiting, the actual infiltration rate is equal to the rate at which rainwater is supplied to the ground surface. Later in the storm when the rainfall rate is greater than the infiltration rate, the actual infiltration rate will be greater than that predicted by the previous equation, because infiltration was limited in the early in the storm.

An integrated version of the Horton method can account for the underestimation of infiltration rate due to limiting rainfall intensity early in a storm (Viessman et al. 1977; Bedient and Huber 1992; Chin 2000), as can more complicated infiltration models such as the Green and Ampt (1911) model. Nevertheless, the simple Horton model is often used in practice as it yields a larger amount of effective precipitation than does the integrated version of the Horton model, and is thus conservative. Depending on selected parameter values, Horton may or may not yield more effective rainfall than does, for example, the Green and Ampt model (for more information, see "Green and Ampt" on page 14-813).

RTK Methods

The RTK method is based on representing unit hydrographs by a set of triangular hydrographs, which can described by three parameters R, T and K. R is the fraction of runoff that show up as precipitation; T is the time to peak of the hydrograph and K is the ratio of the recession time to time to peak.

A typical hydrograph is shown in the figure below.

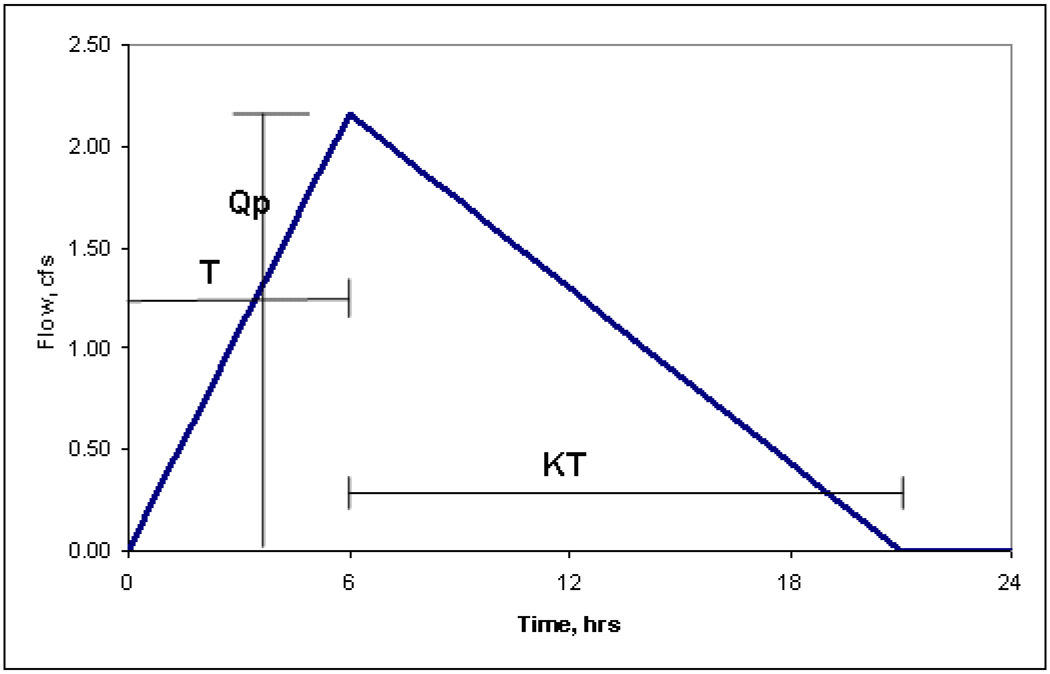


Figure 14-39: A Typical Hydrograph

Most RDII hydrographs do not look like the simple triangular plot above but are really influenced by several different phenomena such as direct inflow and infiltration through groundwater. Investigators have suggested that there should actually be three unit hydrographs representing these processes:

- Rapid inflow
- Moderate infiltration
- Slow infiltration

In Bentley SewerGEMS V8*i*, you determine the nine numerical values: three parameters for each of the three processes. Each hydrograph is generated and they are summed as shown below.

The R values depend on the problems with the sewers (e.g. leaks, illegal connections, sump pumps, roof leaders, etc.). In theory, R should be zero in a sanitary sewer system. The sum of all the R's should be significantly less that one because some water flows to storm systems, some seeps into the ground and some is lost to depression storage and evapotranspiration. The R's are usually around 0.1 but depend heavily on the condition of the system.

The T values depend on the size of the catchment with the rapid inflow T much smaller than the long term infiltration T. K depends on the relative length of the recession curve but is usually on the order of 1.5 to 3, with 1.67 being a typical value (value used in SCS triangular curves).

Procedure

The procedure for developing and applying RTK hydrographs comprises two overall steps:

1. Use precipitation and flow meter data to develop R, T and K values.
2. Given a hyetograph, create a hydrograph for a given storm and place it on loading nodes.The simulation model performs the second step.

Developing Hydrographs

During a model run, you enter a precipitation hyetograph and the R, T and K values convert those values into a hydrograph. The first step is to use the precipitation, area, and R values to determine the three peak flow values:

$$Qp_j(i) = 2.017P(i)\ R_j A/((1+K)T) \tag{14.74}$$

Where	$Qp_j(i)$	=	Hydrograph peak flow for i-th rainfall value for j-th triangular hydrograph, cfs
	$P(i)$	=	Precipitation in i-th hour of event, inches
	A	=	Drainage area, ac
	R_j	=	R value for j-th hydrograph component

P is used here as precipitation although it should be excess precipitation after losses are subtracted out. However, for the RTK method in sanitary sewers, you would most likely set the losses to zero and base everything on precipitation.

As with any unit hydrograph method, we are interested in the time from the i-th hour of precipitation. To get the flow in any hour, the model sums up the hydrographs from all the previous rainfall hours. Each of those values will be based on a sum of the three hydrographs. With that in mind, the individual triangular hydrographs are determined by the following equations:

$$0 \text{for} \quad t < t_i \tag{14.75}$$

Where t_i = The start of i-th precipitation hour

Qp_j(i) (t/T)for 0 < t < T (14.76)

Q_j(t) =

$$Qp_j(i)\left(1-\frac{t-T}{KT}\right)$$ for T < t < (!+K)T (14.77)

0for t > (1+K)T (14.78)

Now that the individual component hydrographs have been created, they are summed to get:

$$Q(t_i)=\sum_{j=1}^{3}Q_j(t_i) \tag{14.79}$$

Now that the hydrographs have been determined for each interval of 0.2T, the hydrographs for the individual units of rain are summed to obtain the overall hydrograph for the catchment.

Thiessen Polygon Generation Theory

"Naïve Method" on page 14-818

"Plane Sweep Method" on page 14-819

Naïve Method

A Thiessen polygon of a site, also called a Voronoi region, is the set of points that are closer to the site than to any of the other sites.

Let $P = \{p_1, p_2,\ldots p_n\}$ be the set of sites and $V = \{v(p_1), v(p_2),\ldots v(p_n)\}$ represent the Voronoi regions or Thiessen polygons for P_i, which is the intersection of all of the half planes defined by the perpendicular bisectors of p_i and the other sites. Thus, a naïve method for constructing Thiessen Polygons can be formulated as follows:

Step 1 For each i such that $i = 1, 2,\ldots, n$, generate n - 1 half planes $H(p_i,p_j)$, $1 </= j </= n$, $i <> j$, and construct their common intersection $v(p_i)$.

Step 2 Report $V = \{v(p_1), v(p_2),\ldots v(p_n)\}$ as the output and stop.

This naïve procedure is, however, very inefficient for generating Thiessen polygons. The computation time increases exponentially as the number of sites increases. There are many other more competent methods for constructing a Thiessen polygon.

Plane Sweep Method

The plane sweep technique is a fundamental method for solving two-dimensional geometric problems. It works with a special line called a sweepline, a vertical line sweeping the plane from left to right. It hits objects one by one as the sweepline moves. Whenever it crosses an object, a portion of the problem is solved. Therefore, it enables a two-dimensional problem to be solved in a sequence of one-dimension processing. Sweep plane technique provides a conceptually simple and efficient algorithm. Steven Fortune (1986; 1987) has developed a sweepline algorithm for constructing Thiessen polygons. This algorithm has been implemented in the SewerGEMS V8*i* Thiessen Polygon Generator. The detailed working algorithm is given as follows:

1. Q <------- P.
2. Choose and delete the left-most point, say p_i from Q.
3. L <------- the list consisting of a single region $\varphi(V(p_i)$.
4. While Q is not empty, repeat Steps 1-3.
5. If w is a site, say $w = p_i$, do:
    a. Find region $\varphi(V(p_i)$ on L containing p_i.
    b. Replace $\varphi(V(p_i)$ on L by the sequence ($\varphi(V(p_j)$, $h^-(pi, pj)$, ($\varphi(V(p_i))$, $h^+(pi, pj)$, $\varphi(V(p_j)$.
    c. Add to Q the intersection of $h^-(pi, pj)$ with the intermediate lower half hyperbola on L and the intersection of $h^+(pi, pj)$ with the immediate upper half hyperbola on L.
6. If w is an intersection, say $w = \varphi(q_t)$, do:
    a. Replace sub-sequence ($h^{\pm}(p_i, p_j)$, $\varphi(V(p_i))$, $h^{\pm}(p_i, p_k)$) on L by $h = h^-(p_i, p_k)$ or $h = h^+(p_i, p_k)$ appropriately.
    b. Delete from Q any intersection of $h^{\pm}(p_i, p_j)$ or $h^{\pm}(p_i, p_k)$ with others.

c. Add to Q any intersection of h with its immediate upper half hyperbola and its immediate lower half parabola on L.

d. Mark $\varphi(q_t)$ as a Voronai vertex incident to $h^{\pm}(p_i, p_j)$, $h^{\pm}(p_i, p_k)$, and h.

7. Repeat all half hyperbolas ever listed on L, all the Voronai vertices marked in the preceding step, and the incidence relations among them.

The sweepline algorithm is an efficient technique for constructing a Thiessen polygon. The computation time required for the worst case is $O(n\log n)$. It produces a far more competent method than the naïve method and provides satisfactory performance for generating Thiessen polygons for a large number of points.

Chapter 15

Editing Attributes in the Property Editor

The Property Editor is a manager (titled, "Properties") that lets you define and view the data that defines your model. If it is not open, press **F4** or click **View > Properties** to open the Property Editor.

Note: Some data described in this chapter may not appear in the Property Editor but are available for use when you set up a FlexTable. For more information on FlexTables, see "Viewing and Editing Data in FlexTables" on page 10-557.

Pressure Pipe Attributes

The pressure pipe attributes comprise the following categories:

Pressure Pipe—General

Table 15-1: Pressure Pipe—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.
Node Reversal	Lets you reverse the direction of the currently highlighted element. Click in the field to display an **Ellipsis** (…) button. Clicking the ellipsis button in this field causes the start node and stop node to be exchanged with one another, which reverses the direction of the currently highlighted element.
Start Node ID	Displays the start, or upstream, node of the currently highlighted element. This field is not editable.
Stop Node ID	Displays the stop, or downstream, node of the currently highlighted element. This field is not editable.

Pressure Pipe—Geometry

Table 15-2: Pressure Pipe—Geometry Attributes

Attribute	Description
Geometry	Lets you view and edit the coordinates of points along a selected element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Polyline Vertices feature. For more information, see “Polyline Vertices Dialog Box” on page 6-198.
Has User Defined Length	Lets you choose whether the highlighted element uses scaled or user-defined length. If this field is set to True, the Length (User Defined) field is activated.
Length (User Defined)/ Scaled Length	Lets you enter a value for the length of the currently highlighted element. To use this field, you must set Has User Defined Length field to **True**. If you set this field to **False**, it displays the scaled length for the currently highlighted element.

Pressure Pipe—Physical

Table 15-3: Pressure Pipe—Physical Attributes

Attribute	Description
Diameter	Lets you enter a value for the diameter of the cross section of the currently highlighted element.
Material	Lets you enter the name of the material used. Alternatively, clicking the ellipsis button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Manning’s n	Lets you enter a value for the Manning’s roughness of the currently highlighted element. This field is available only if you selected Mannings as the default Pressure Friction Method in the Calculation Options Manager.
Hazen-Williams C	Lets you enter a value for the Hazen-Williams roughness of the currently highlighted element. This field is available only if you selected Hazen-Williams as the default Pressure Friction Method in the Calculation Options Manager.

Table 15-3: Pressure Pipe—Physical Attributes

Attribute	Description
Darcy-Weisbach e	Lets you enter a value for the Darcy-Weisbach roughness of the currently highlighted element. This field is available only if you selected Darcy-Weisbach as the default Pressure Friction Method in the Calculation Options Manager.
Kutter's n	Lets you enter a value for the Kutter's roughness of the currently highlighted element. This field is available only if you selected Kutters as the default Pressure Friction Method in the Calculation Options Manager.
Invert (Start)	Lets you enter a value for the start, or upstream, invert of the currently highlighted element.
Set Invert to Start Node?	Sets the start invert of the current element to the elevation of the start node (upstream). Set this to **False** to enter a value for the elevation of the invert, or set this field to **True** to use the start-node elevation.
Invert (Stop)	Lets you enter a value for the stop, or downstream, invert of the currently highlighted element.
Slope	Lets you enter a value for the slope.

Pressure Pipe—Physical: Minor Losses

Table 15-4: Pressure Pipe—Minor Losses Attributes

Attribute	Description
Minor Loss Coefficient	Lets you enter a value for the minor loss coefficient of the currently highlighted element. Alternatively, clicking the ellipse button opens the Minor Loss Collection dialog box, which lets you generate composite minor loss coefficients to be applied to the pressure pipe. For more information, see "Adding a Minor Loss Collection to a Pressure Pipe" on page 6-196.

Pressure Pipe—Active Topology

Table 15-5: Pressure Pipe—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Pressure Pipe—Results

Table 15-6: Pressure Pipe—Result Attributes

Attribute	Description
Flow	Representative calculated flow in the pressure pipe. The flow calculated in the middle section of the pressure pipe. This is a calculated results field and is not editable.
Velocity	Representative calculated velocity of the flow in the pressure pipe. The velocity calculated in the middle section of the pressure pipe. This is a calculated results field and is not editable.

Conduit Attributes

The conduit attributes comprise the following categories:

- “Conduit—General” on page 15-826
- “Conduit—Geometry” on page 15-827
- “Conduit—Infiltration” on page 15-827
- “Conduit—Output Filter” on page 15-828
- “Conduit—Physical” on page 15-829
- “Conduit—Physical: Additional Losses” on page 15-833
- “Conduit—Physical: Control Structure” on page 15-833
- “Conduit—Physical: Section Type: Culvert” on page 15-834
- “Conduit—Active Topology” on page 15-836
- “Conduit—Results” on page 15-837
- “Conduit—Results: Capacities” on page 15-838

- "Conduit—Results: Engine Parsing" on page 15-839

Conduit—General

Table 15-7: Conduit—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see "Adding Hyperlinks to Elements" on page 6-324.
Node Reversal	Lets you reverse the direction of the currently highlighted element. Click in the field to display an **Ellipsis** (…) button. Clicking the ellipsis button in this field causes the start node and stop node to be exchanged with one another, which reverses the direction of the currently highlighted element.
Start Node ID	Displays the start, or upstream, node of the currently highlighted element. This field is not editable.
Stop Node ID	Displays the stop, or downstream, node of the currently highlighted element. This field is not editable.

Conduit—Geometry

Table 15-8: Conduit—Geometry Attributes

Attribute	Description
Has User Defined Length	Lets you choose whether the highlighted element uses scaled or user-defined length. If this field is set to True, the Length (User Defined) field is activated.
Length (User Defined)/ Scaled Length	Lets you enter a value for the length of the currently highlighted element. To use this field, you must set Has User Defined Length field to **True**. If you set this field to **False**, it displays the scaled length for the currently highlighted element.
Geometry	Lets you view and edit the coordinates of points along a selected element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Polyline Vertices feature. For more information, see "Polyline Vertices Dialog Box" on page 6-198.

Conduit—Infiltration

Table 15-9: Conduit—Infiltration Attributes

Attribute	Description
Infiltration Load Type	Lets you select the type of infiltration load associated with the selected conduit. You can select **Pipe Length**, **Pipe Rise-Length**, **Pipe Surface Area**, **Count Based**, **Hydrograph, Pattern Load**, **or None**.
Flow (Infiltration)	Lets you enter any additional infiltration flow at the selected conduit.
Infiltration Loading Unit	Lets you select the unit of measure for the infiltration load. To use this field, you must set the Infiltration Loading Type to **Pipe Length**, **Pipe Rise-Length**, or **Pipe Surface Area**.
Infiltration Rate per Loading Unit	Lets you enter the infiltration rate per load for the selected conduit. To use this field, you must set the Infiltration Loading Type to **Pipe Length**, **Pipe Rise-Length**, **Pipe Surface Area**, or **Count Based**.

Table 15-9: Conduit—Infiltration Attributes

Attribute	Description
Infiltration Unit Count	Lets you enter the infiltration unit count for the selected conduit. To use this field, you must set the Infiltration Loading Type to **Count Based**.
Hydrograph Curve	Lets you define the infiltration load for the selected conduit as a hydrograph. Click the **Ellipsis** (…) button in this field to display the Hydrograph Curve dialog box, where you can define the Time vs. Flow data points that make up the hydrograph curve. To use this field, you must set the Infiltration Loading Type to **Hydrograph**.
Infiltration Pattern	Lets you define the infiltration load as a pattern. You can select **Fixed** (the default value), an existing pattern, or **Edit Pattern**. When you select Edit Pattern, the Patterns dialog box appears. To use this field, you must set the Infiltration Loading Type to **Pattern Load**.
Flow (Infiltration Base)	Lets you enter the base infiltration flow for the selected conduit. To use this field, you must set the Infiltration Loading Type to **Pattern Load**.

Conduit—Output Filter

Table 15-10: Conduit—Output Filter Attributes

Attribute	Description
Output Options	Lets you switch between summary and detailed versions of the calculation results. Select **Detailed Results** to include all the section results for the link in the project file. Select **Summary Results** to include results only for the start, middle, and stop sections of a link. Selecting Summary Results, which stores less data than Detailed Reports, might make color coding, annotation, and other processes quicker than Detailed Results for larger projects. You might use Detailed Results only for a small section of a large model.

Conduit—Physical

Table 15-11: Conduit—Physical Attributes

Attribute	Description
Section Type	Lets you choose the cross-sectional shape of the currently highlighted element: **Irregular Open Section**, **Trapezoidal**, **Circle**, **Box**, **Basket-Handle**, **Ellipse**, **Horseshoe Conduit**, **Egg**, **Semi-Ellipse**, **Catalog Pipe**, **Pipe-Arch**, **Virtual**, **Semi-Circle**, **Catenary**, **Gothic**, **Modified Basket-Handle**, **Rectangular-Round**, **Rectangular-Triangle**, **Power**, **Parabola**, **Triangle**, **Rectangular**, **or Irregular Closed Section**. The value chosen here affects the availability of the following fields.
Station-Depth Curve	Lets you define station-depth points that describe the shape of the irregular channel. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Station-Depth Curve dialog box. To use this field, you must set the Section type to **Irregular Channel**.
Elevations Modifier	The Elevations modifier is a constant value that will be added to each elevation value. This attribute is only used during SWMM calculations.
Left Bank Station	The station value (horizontal distance) in the Station-Depth table that marks the end of the left overbank (set to zero or to the minimum station value in the Station-Depth curve to denote the absence of an overbank). To use this field, you must set the Section type to Irregular Channel.
Stations Modifier	The Stations modifier is a factor by which the distance between each station will be multiplied when the transect data is processed by SWMM. Use a value of 0 if no such factor is needed. This attribute is only used during SWMM calculations.
Right Bank Station	The station value (horizontal distance) in the Station-Depth table that marks the start of the right overbank (set to the maximum station value in the Station-Depth curve to denote the absence of an overbank). To use this field, you must set the Section type to Irregular Channel.
Base Width	Lets you enter a value for the width at the base of the cross section of the currently highlighted element. To use this field, you must set The Section Type to **Trapezoidal Channel**.

Table 15-11: Conduit—Physical Attributes

Attribute	Description
Height	Lets you enter a value for the height of the cross section of the currently highlighted element. To use this field, you must set Section Type to **Trapezoidal Channel**.
Slope (Right Side)	Lets you enter a value for the right slope of the cross section of the currently highlighted element. To use this field, you must set Section Type to **Trapezoidal Channel**.
Slope (Left Side)	Lets you enter a value for the left slope of the cross section of the currently highlighted element. To use this field, you must set Section Type to **Trapezoidal Channel**.
Diameter	Lets you enter a value for the diameter of the cross section of the currently highlighted element. To use this field, you must set The Section Type to **Circle**.
Fill Depth	Is the amount of sedimentation that a section has. To use this field, you must set the Section Type to **Circle**. Only the SWMM engine uses this in calculating your model.
Rise	Lets you enter a value for the rise (height or vertical dimension) of the currently highlighted element. To use this field, you must set Section Type to any of the types except Circle, Irregular Open Section, Irregular Closed Section, Trapezoidal, and Catalog Pipe.
Span	Lets you enter a value for the span (width or horizontal dimension) of the currently highlighted element. To use this field, you must set Section Type to any of the types except Circle, Irregular Channel and Trapezoidal Channel.
Number of Barrels	Lets you set the number of barrels that comprise the currently highlighted element. Note that the diameter, rise, and/or span values are applied to each barrel. To use this field, you must set Section Type to any of the types except Irregular Open Section, Irregular Closed Section, and Trapezoidal.
Catalog Pipe	Lets you define or select a catalog pipe for the selected conduit. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Catalog Pipe dialog box. To use this field, you must set the Section type to **Catalog Pipe**.
Rect Bottom Radius	Lets you enter the radius of the circular portion of the Rectangular-Round section. To use this field, you must set the Section Type to **Rectangular-Round**.

Table 15-11: Conduit—Physical Attributes

Attribute	Description
Rect Triangle Triangle Height	Lets you enter the height of the triangular portion of the Rectangular-Triangle section. To use this field, you must set the Section Type to **Rectangular-Triangle**.
Power Exponent	Lets you enter the power exponent for the Power section. To use this field, you must set the Section Type to **Power**.
Depth-Width Curve	Lets you define depth-width points that describe the shape of the irregular closed section. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Depth-Width Curve dialog box. To use this field, you must set the Section type to **Irregular Closed Section**.
Material	Lets you enter the name of the material used. Alternatively, clicking the ellipsis button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Manning's n	Lets you enter a value for the Manning's roughness of the currently highlighted element. This attribute is active only when the Roughness Type attribute is set to **Single Manning's n**.
Manning's n-Depth Curve	Lets you define points that describe a roughness-depth curve for the currently highlighted element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Manning's n-Depth Curve dialog box. To use this field, you must set Roughness Type attribute is set to **Manning's n-Depth Curve**.
Manning's n-Flow	Lets you define points that describe a roughness-flow curve for the currently highlighted element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Manning's n-Flow Curve dialog box. To use this field, you must set Roughness Type attribute is set to **Manning's n-Flow**.
Roughness Type	Lets you set the roughness method for the currently highlighted: **Single Manning's n**, **Manning's - Depth Curve**, or **Manning's n - Flow**. The value chosen here affects the availability of some fields in the Physical section of the Property Editor.

Table 15-11: Conduit—Physical Attributes

Attribute	Description
Set Invert to Start Node?	Sets the start invert of the current element to the elevation of the start node (upstream). Set this to **False** to enter a value for the elevation of the invert, or set this field to **True** to use the start-node elevation.
Invert (Start)	Lets you enter a value for the start, or upstream, invert of the currently highlighted element. To use this field, you must set Set Invert to Start Node? to **False**.
Set Invert to Stop Node	Sets the stop invert of the current element to the elevation of the stop node (downstream). Set this to **False** to enter a value for the elevation of the invert, or set this field to **True** to use the stop-node elevation.
Invert (Stop)	Lets you enter a value for the stop, or downstream, invert of the currently highlighted element. To use this field, you must set Set Invert to Stop Node to **False**.
Slope	The difference between the start invert and stop invert divided by the length of the conduit. This is a calculated results field and is not editable.
Roughness Type	Lets you select the roughness type for the conduit. For more information see "Roughness Models" on page 14-742.
Overbank Channel	Lets you specify different roughness values for the left overbank flow, the channel flow, and the right overbank flow. To use this field, you must set the Section type to Irregular Channel.
Horizontal Segment	Lets you specify a different roughness for each horizontal segment of the channel. This information is entered in the Station-Depth curve. To use this field, you must set the Section Type to Irregular Channel.
Left Overbank Manning's n	The Manning's roughness coefficient for overbank flow on the left side of the channel (between the minimum station value on the Station-Depth curve and the Left Bank Station). To use this field, you must set the Section Type to Irregular Channel and the Roughness Type to Overbank Channel.
Right Overbank Manning's n	The Manning's roughness coefficient for overbank flow on the right side of the channel (between the Right Bank Station and the maximum station value on the Station-Depth curve). To use this field, you must set the Section Type to Irregular Channel and the Roughness Type to Overbank Channel.

Conduit—Physical: Additional Losses

Table 15-12: Conduit—Additional Loss Attributes

Attribute	Description
Entrance Loss Coefficient	Lets you define entrance loss coefficients, if any, for the currently highlighted element.
Exit Loss Coefficient	Lets you define exit loss coefficients, if any, for the currently highlighted element.

Conduit—Physical: Control Structure

Table 15-13: Conduit—Control Structure Attributes

Attribute	Description
Start Control Structure Type	Lets you choose whether to use an Inline or Side Start Control Structure for the selected conduit. Inline start control structures are used for inline flow regulation while side start control structures are used for flow diversion.
Flap Gate?	Lets you choose whether or not the highlighted element has a flap gate. If this is set to **True**, and icon displays at the stop-end of the conduit to display the presence of the structure. If this is set to True and you design control structures without flap gates selected (see "Defining a Control Structure in a Conduit" on page 6-192), the flap gate check box will be turned on for your control structures and a message displayed.
Has Start/ Stop Control Structure?	Lets you define whether or not the currently highlighted element has a control structure, and if so, which type. The value chosen here affects the availability of the other fields. If this is set to **True**, an icon displays at the start/stop-end of the conduit to display the presence of the structure.
Start/Stop Control Structure	Lets you design a start and/or stop control structure, or choose a preexisting one. Click the Ellipsis (…) button to open the Conduit Control Structure dialog box to set up the control structure you want to use.

Conduit—Physical: Section Type: Culvert

To use these attributes, you must set Section Type to **Circle** or **Box**.

Table 15-14: Conduit—Culvert Attributes

Attribute	Description
Is Culvert?	Lets you choose whether or not the section type for the selected conduit is a culvert. If you select **True**, the other Culvert attributes are enabled in the Property Editor. If you select **False**, none of the other Culvert attributes are available in the Property Editor.
Inlet Description	Lets you type or select a description for the inlet. Click the **Ellipse (...)** button to display the Culvert Inlet Coefficient Engineering Library, where you can select an existing culvert.
Culvert Equation Form	Lets you select Form 1 or Form 2. To use this field, you must set the Section Type to **Circle** or **Box** and the Is Culvert? attribute to **True**.
C	Lets you define the C equation coefficient that is used in the submerged inlet control equation. To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.
M	Lets you define the M equation coefficient that is used in both forms of the unsubmerged inlet control equation. To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.
K	Lets you define the K equation coefficient that is used in both forms of the unsubmerged inlet control equation. To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.
Y	Lets you define the Y equation coefficient that is used in the submerged inlet control equation. To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.
Ke	Lets you define the entrance loss value for the associated conduit. To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.
Kr	Lets you define the reverse flow loss value for the associated conduit. To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.

Table 15-14: Conduit—Culvert Attributes

Attribute	Description
Slope Correction Factor	Lets you define the Slope Correction Factor to be used in inlet control calculations. Normally this factor is –0.5, but for mitered inlets, HDS No. 5 suggests +0.7.To use this field, you must set the Section Type to **Circle** or **Box** and Is Culvert? to **True**.
Has Overtopping Weir?	If True, this allows the user to specify a section of roadway that acts as an overtopping weir in the event that the culvert headwater elevation exceeds the roadway elevation. When set to true, SewerGEMS V8*i* will include the overtopping weir in the elevation-discharge-tailwater (EQT) calculations for the culvert crossing. Weir flow is computed assuming the road acts as a broad-crested weir. Flow that passes over the overtopping weir will discharge to the same downstream node as the culvert.
Elevation (Roadway Crest)	The elevation of the roadway crest. If the culvert headwater elevation exceeds this elevation, flow will overtop the roadway and the roadway will act like a weir.
Roadway Cross Section Length	The length of the roadway section acting as a weir when flow is overtopping the road. Often this is set equal the top width of flow in the channel upstream of the culvert.
Use Weir C-Depth Table?	If True, this lets you define a table of weir coefficient versus flow depth values to simulate a weir coefficient that changes as the flow depth changes.
C-Depth Table	A table of weir coefficient versus flow depth values. To use this field you must set the 'Use Weir C-Depth Table?' field to True.
Roadway Weir Coefficient	The weir coefficient for the roadway (typically between 2.5 and 3.1 for US units, or between 1.4 and 1.7 for SI units). This coefficient is considered dimensionless and is used in the broad crested weir equation (see “Broad-Crested Weir” on page 14-760) to compute the flow that overtops the roadway. To use this field you must set the 'Use Weir C-Depth Table?' field to False.

Table 15-14: Conduit—Culvert Attributes

Attribute	Description
Depth (Maximum Overtopping)	This value is used to determine the maximum headwater elevation to use when computing the culvert EQT table (maximum headwater equals 'Elevation (Roadway Crest)' plus 'Depth (Maximum Overtopping). To ensure accurate calculations this value should be set higher than the maximum expected depth of flow over the weir; however users should note that using very large values for this field may slow down model computations. If the headwater exceeds the highest elevation in the EQT table SewerGEMS V8*i* will linearly extrapolate the values.
Increment	The depth increment used to generate EQT curves. A small value provides more accuracy but reduces the performance.
User Defined Tailwater?	If True, this lets the user specify the maximum tailwater elevation used when computing the culvert EQT table. If False, SewerGEMS V8*i* will chose a maximum tailwater elevation based on the geometry of the downstream element (for example if the culvert discharges to a channel downstream, SewerGEMS V8*i* will use the channel's top water level)
Elevation (Maximum Tailwater)	Lets you define the maximum tailwater elevation used when computing the culvert EQT table. To ensure accurate calculations, this elevation should be greater that the highest expected water surface elevation at the element immediately downstream from the culvert; however users should note that using very large values for this field may slow down model computations. To use this field you must set the 'User Defined Tailwater?' field to True.

Conduit—Active Topology

Table 15-15: Conduit—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Conduit—Results

The hydraulic grade, flow, and depth/rise presented in the Property Editor are the values associated with the middle section of the link. Obviously these values don't give you the full sense of what is going on, so the middle, start, and stop values are also available through the FlexTables, and you can also look at the profile. (For more information, see “Viewing and Editing Data in FlexTables” on page 10-557 and “Using Profiles” on page 10-548.)

Table 15-16: Conduit—Result Attributes

Attribute	Description
Flow	Representative calculated flow in the conduit. The flow calculated in the middle section of the conduit. This is a calculated results field and is not editable.
Velocity	Representative calculated velocity of the flow in the conduit. The velocity calculated in the middle section of the conduit. This is a calculated results field and is not editable.
Sections Results	Lets you view the calculated flow variables at the start, middle and end of the conduit section. Clicking the **Ellipses (...)** button displays the Sections Results dialog box.
Pollutants	Lets you view calculated results for pollutants. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Conduit—Results: Capacities

Table 15-17: Conduit—Result: Capacity Attributes

Attribute	Description
Capacity (Full)	The normal full flow depth of the conduit. This is a calculated results field and is not editable.
Capacity (Excess Full)	The difference between the flow in the conduit and the Capacity (Full). This is a calculated results field and is not editable.
Flow/Full Flow Capacity	The ratio of the flow in the conduit to the Capacity (Full). This is a calculated results field and is not editable.
Design Percent Full	Lets you enter the percentage full that you would like the link to maintain. If you want the pipe to be 75% full, enter in the 75 in the field. These values do not affect network calculations; they are informational only. In the Results attributes is a field called Capacity (Calculated Design). This displays the result of calculating with the Manning's equation the amount of flow going through the link if it were full to the percentage you set in Design Percent Full (in this example, the amount of flow if the link were 75% full).
Capacity (Calculated Design)	The normal flow calculated based on the percent of full flow established by the design percent full. This is a calculated results field and is not editable.
Capacity (Excess Design)	The difference between the flow in the conduit and the Capacity (Calculated Design). This is a calculated results field and is not editable.
Flow/Design Capacity	The ratio of the flow in the conduit to the Capacity (Calculated Design). This is a calculated results field and is not editable.

Conduit—Results: Engine Parsing

Table 15-18: Conduit—Result: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see “Branches” on page 14-722.
Section Count	The number of sections that the implicit engine divides the conduits into during an analysis. Not available during a SWMM analysis. This is a calculated results field and is not editable. For more information, see “Section Count” on page 14-723.

Channel Attributes

The channel attributes comprise the following categories:

- “Channel—General” on page 15-840
- “Channel—Geometry” on page 15-841
- “Channel—Active Topology” on page 15-843
- “Channel—Output Filter” on page 15-842
- “Channel—Physical” on page 15-842
- “Channel—Physical: Control Structure” on page 15-843
- “Channel—Results” on page 15-844
- “Channel—Results: Engine Parsing” on page 15-844

Channel—General

Table 15-19: Channel—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.

Table 15-19: Channel—General Attributes

Attribute	Description
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.
Node Reversal	Lets you reverse the direction of the currently highlighted element. Click in the field to display an **Ellipsis** (…) button. Clicking the ellipsis button in this field causes the start node and stop node to be exchanged with one another, which reverses the direction of the currently highlighted element.
Start Node ID	Displays the start, or upstream, node of the currently highlighted element. This field is not editable.
Stop Node ID	Displays the stop, or downstream, node of the currently highlighted element. This field is not editable.

Channel—Geometry

Table 15-20: Channel—Geometry Attributes

Attribute	Description
Geometry	Lets you view and edit the coordinates of points along a selected element. Click the **Ellipsis** (…) button in this field to open the Polyline Vertices dialog box. For more information, see “Polyline Vertices Dialog Box” on page 6-198.
Has User Defined Length	Lets you choose whether the highlighted element uses scaled or user-defined length. If this field is set to True, the Length (User Defined) field is activated.
Length (User Defined)/ Scaled Length	Lets you enter a value for the length of the currently highlighted element. This attribute is active only when the Has User Defined Length attribute is set to **True**. If this field is set to **False**, it displays the scaled length for the currently highlighted element.

Channel—Output Filter

Table 15-21: Channel—Output Filter Attributes

Attribute	Description
Output Options	Lets you switch between summary and detailed versions of the calculation results. Select **Detailed Results** to include all the section results for the link in the project file. Select **Summary Results** to include results only for the start, middle, and stop sections of a link. Selecting Summary Results, which stores less data than Detailed Reports, might make color coding, annotation, and other processes quicker than Detailed Results for larger projects. You might use Detailed Results only for a small section of a large model.

Channel—Physical

Manning's n is not a property of a channel but a property of the cross-section nodes along the channel. The cross-section defines the shape and other physical properties of the channel (except for length). The properties between two cross sections are interpolated. For more information, see "Cross Section—Physical" on page 15-875.

Table 15-22: Channel—Physical Attributes

Attribute	Description
Invert (Start)	Lets you enter a value for the start, or upstream, invert of the currently highlighted element.
Invert (Stop)	Lets you enter a value for the stop, or downstream, invert of the currently highlighted element.
Slope	The difference between the start invert and stop invert divided by the length of the channel. This is a calculated results field and is not editable.

Channel—Physical: Control Structure

Table 15-23: Channel—Control Structure Attributes

Attribute	Description
Start Control Structure Type	Lets you choose whether to use an Inline or Side Start Control Structure for the selected conduit. Inline start control structures are used for inline flow regulation while side start control structures are used for flow diversion. A side control is applied to control structures which generally divert flow at higher levels and an inline structure is applied to the primary flow direction.
Flap Gate?	Lets you choose whether or not the highlighted element has a flap gate. If this is set to **True**, and icon displays at the stop-end of the conduit to display the presence of the structure. If this is set to True and you design control structures without flap gates selected, the flap gate check box will be turned on for your control structures and a message displayed.
Has Start/ Stop Control Structure?	Lets you define whether or not the currently highlighted element has a control structure, and if so, which type. The value chosen here affects the availability of the other fields. If this is set to **True**, an icon displays at the start/stop-end of the conduit to display the presence of the structure.
Start/Stop Control Structure	Lets you design a start and/or stop control structure, or choose a preexisting one. Click the Ellipsis (…) button to open the Conduit Control Structure dialog box to set up the control structure you want to use.

Channel—Active Topology

Table 15-24: Channel—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Channel—Results

The hydraulic grade, flow, and depth/rise presented in the Property Editor are the values associated with the middle section of the link. Obviously these values don't give you the full sense of what is going on, so the middle, start, and stop values are also available through the FlexTables, and you can also look at the profile. (For more information, see "Viewing and Editing Data in FlexTables" on page 10-557 and "Using Profiles" on page 10-548.)

Table 15-25: Channel—Result Attributes

Attribute	Description
Flow	Representative calculated flow in the channel. The flow is calculated in the middle section of the channel. This is a calculated results field and is not editable.
Velocity	Representative calculated velocity of the flow in the channel. The velocity is calculated in the middle section of the channel. This is a calculated results field and is not editable.
Sections Results	Lets you view the calculated flow variables at the start, middle and end of the channel section. Clicking the **Ellipses (...)** button displays the Sections Results dialog box.
Pollutants	Lets you view calculated results for pollutants. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Channel—Results: Engine Parsing

Table 15-26: Channel—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see "Branches" on page 14-722.
Section Count	The number of sections that the implicit engine divides the channels into during an analysis. Not available during a SWMM analysis. This is a calculated results field and is not editable. For more information, see "Section Count" on page 14-723.

Gutter Attributes

The gutter attributes comprise the following categories:

- "Gutter—General" on page 15-845
- "Gutter—Geometry" on page 15-846
- "Gutter—Physical" on page 15-846
- "Gutter—Active Topology" on page 15-847
- "Gutter—Results" on page 15-848

Gutter—General

Table 15-27: Gutter—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see "Adding Hyperlinks to Elements" on page 6-324.
Node Reversal	Lets you reverse the direction of the currently highlighted element. Click in the field to display an **Ellipsis** (…) button. Clicking the ellipsis button in this field causes the start node and stop node to be exchanged with one another, which reverses the direction of the currently highlighted element.
Start Node ID	Displays the start, or upstream, node of the currently highlighted element. This field is not editable.
Stop Node ID	Displays the stop, or downstream, node of the currently highlighted element. This field is not editable.

Gutter—Geometry

Table 15-28: Gutter—Geometry Attributes

Attribute	Description
Geometry	Lets you view and edit the coordinates of points along a selected element. Click the **Ellipsis** (…) button in this field to open the Polyline Vertices dialog box. For more information, see “Polyline Vertices Dialog Box” on page 6-198.
Has User Defined Length	Lets you choose whether the highlighted element uses scaled or user-defined length. If this field is set to True, the Length (User Defined) field is activated.
Length (User Defined)/ Scaled Length	Lets you enter a value for the length of the currently highlighted element. This attribute is active only when the Has User Defined Length attribute is set to **True**. If this field is set to **False**, it displays the scaled length for the currently highlighted element.

Gutter—Physical

Table 15-29: Gutter—Physical Attributes

Attribute	Description
Open Cross Section	Lets you choose the cross-sectional shape of the currently highlighted element. The value chosen here (Trapezoidal or Irregular) affects the availability of the following fields.
Irregular Channel	Lets you define station-elevation points that describe the shape of the irregular channel. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Irregular Channel dialog box. To use this field, you must set Open Cross-Section attribute to **Irregular Channel**.
Bottom Width	Lets you enter a value for the width at the base of the cross section of the currently highlighted element. To use this field, you must set Open Cross-Section attribute to **Trapezoidal Channel**.
Slope (Left Side)	Lets you enter a value for the left slope of the cross section of the currently highlighted element. To use this field, you must set Open Cross-Section attribute to **Trapezoidal Channel**.

Table 15-29: Gutter—Physical Attributes

Attribute	Description
Slope (Right Side)	Lets you enter a value for the right slope of the cross section of the currently highlighted element. To use this field, you must set the Open Cross-Section attribute to **Trapezoidal Channel**.
Trapezoidal Channel Depth	Lets you enter a value for the depth.To use this field, you must set the Open Cross-Section attribute to **Trapezoidal Channel**.
Gutter Material	Lets you enter the name of the material used. Alternatively, clicking the ellipsis button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Manning's n (Gutter)	Lets you enter a value for the Manning's roughness of the currently highlighted gutter. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Engineering Library to select a Manning's n value for the gutter material you are using.
Slope	Lets you enter a value for the slope of the gutter.

Gutter—Active Topology

Table 15-30: Gutter—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Gutter—Results

Table 15-31: Gutter—Results Attributes

Attribute	Description
Flow	Representative calculated flow in the gutter. The flow is calculated in the middle section of the gutter. This is a calculated results field and is not editable.
Velocity	Representative calculated velocity of the flow in the gutter. The velocity is calculated in the middle section of the gutter. This is a calculated results field and is not editable.
Depth/Rise	The ratio of the depth of flow in the channel divided by the height of the gutter. This is a calculated results field and is not editable.
Depth	The depth of flow measured from the invert of the gutter. A gutter is assumed to have the same depth throughout the link (i.e., there is no middle, start, and stop depths, just a constant).

Manhole Attributes

The manhole attributes comprise the following categories:

Manhole—General

Table 15-32: Manhole—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Manhole—Geometry

Table 15-33: Manhole—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Manhole—Physical

Table 15-34: Manhole—Physical Attributes

Attribute	Description
Elevation (Ground)	Displays the ground elevation for the currently highlighted node.
Set Rim to Ground Elevation?	Enables or disables a data entry shortcut. If the value is **True**, the node's rim elevation is set equal to the ground elevation automatically.
Elevation (Rim)	Lets you define the top elevation of the currently highlighted node. This elevation is typically flush with the ground surface. In some cases, the rim elevation may be slightly below the ground surface elevation (sunk) or slightly above the ground surface elevation (raised).
Elevation (Invert)	Lets you define the elevation at the bottom of the currently highlighted node.
Structure Shape Type	Lets you choose the shape of the currently highlighted element. You can select **Rectangular Structure** or **Circular Structure**. The value chosen here affects the availability of other fields.
Diameter	Lets you enter a value for the diameter of the currently highlighted element. This field is available only when the Structure Shape Type attribute is set to **Circular Structure**.
Length	Lets you enter a value for the length of the currently highlighted element. This field is available only when the Structure Shape Type attribute is set to **Rectangular Structure**.
Width	Lets you enter a value for the width of the currently highlighted element. This field is available only when the Structure Shape Type attribute is set to **Rectangular Structure**.
Bolted Cover	Lets you set whether a manhole has a bolted cover or not. A value of **True** in this field indicates that the associated manhole has a bolted cover. If the manhole cover is bolted, then the hydraulic grade line is not reset to the rim elevation at the downstream end of the upstream pipes in the case of a flooding situation (the calculated HGL being higher than the rim elevation).

Manhole—Physical: Structure Losses

Table 15-35: Manhole—Structure Loss Attributes

Attribute	Description
Headloss Method	Lets you select the headloss method to use: **Absolute**, **HEC-22**, **Generic**, or **Standard**.
Absolute Headloss	Lets you enter a value for the headloss. This field is only available if you selected the **Absolute Headloss Method**.
Headloss Coefficient Stop	Lets you enter a value for the headloss coefficient at the stop section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient Start	Lets you enter a value for the headloss coefficient at the start section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient	Lets you enter a value for the headloss coefficient for the manhole. This field is only available if you selected the **Standard Headloss Method**.

Manhole—Physical: Surface Storage

Table 15-36: Manhole—Surface Storage Attributes

Attribute	Description
Surface Storage Type	Lets you choose how surface storage is handled. The value chosen here affects the availability of other fields: select **Default Storage Equation**, **Surface Depth-Area Curve**, **No Storage**, or **Ponded Area**.The Default Storage Equation uses the following formula: Area = $0.262\ y - 0.068\ y^2 + 0.006\ y3$ (y<4.0 ft) Area=0.344 (y>4.0 ft) where Y is the depth above the ground.
Surface Depth-Area Curve	Lets you describe the volume of the surface storage by defining depth vs. area points. Click the **Ellipsis** (…) button in this field to open the Surface Depth-Area dialog box (see “Surface Depth-Area Curve Editor” on page 6-210). This field is available only when the Surface Storage Type attribute is set to **Surface Depth-Area Curve**.
Area (Constant Surface)	Lets you define the area in which ponding occurs at the currently selected element. It is available only when the Surface Storage Type attribute is set to **Ponded Area**.

Manhole—Sanitary Loading

Table 15-37: Manhole—Sanitary Loading Attributes

Attribute	Description
Sanitary Loads	Lets you define a sanitary (dry weather) flow collection for the selected manhole. Click the **Ellipses (...)** button to display the Sanitary (Dry Weather) Flow Collection Editor, which lets you define collections of sanitary (dry weather) loads.

Manhole—SWMM Extended Data

Table 15-38: Manhole—SWMM Extended Data Attributes

Attribute	Description
Apply Treatment?	Lets you specify whether or not treatment is applied at the currently highlighted element. Select **True** to apply treatment. This field is only used during SWMM calculations.
Treatment	Lets you add a collection of pollutants and their associated treatment functions that will be applied at the currently highlighted element. This field is used only during SWMM calculations and is available only if Apply Treatment? is set to **True**. For more information, see “Adding Treatment to a Node” on page 6-321.
Pollutographs	Lets you define a pollutograph collection for the selected manhole. Click the **Ellipses (...)** button to display the Pollutograph Collection dialog box, which lets you add multiple pollutographs to the collection. For more information, see “Adding Pollutographs to a Node” on page 6-307.

Manhole—Active Topology

Table 15-39: Manhole—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Manhole—Inflow

Table 15-40: Manhole—Inflow Attributes

Attribute	Description
Inflow Collection	Lets you define an inflow collection for the selected manhole. Click the **Ellipsis** (…) button to display the Inflow Collection dialog box for the associated element (see “Defining Inflow Collections” on page 7-378).

Manhole—Results

Table 15-41: Manhole—Results Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the manhole. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded?	If this field displays True, flooding occurs at the currently highlighted element during the current time step. For manhole and catch basin elements, this field will display True if the HGL goes above the rim elevation. This is a calculated results field and is not editable.
Total Outflow	Sum of all flows leaving manhole. This is a calculated results field and is not editable.
Overflow	Flow that exits system during analysis. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Manhole—Results: Engine Parsing Attributes

Table 15-42: Manhole—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see “Branches” on page 14-722.

Manhole—Results: Extended Node Attributes

Table 15-43: Manhole—Results: Extended Node Attributes

Attribute	Description
Freeboard Height	Distance between the top of the manhole and the water surface. This is a calculated results field and is not editable.
Depth (Flooding)	Depth between top of manhole and the water surface when the manhole is flooded. This is a calculated results field and is not editable.
Volume	Calculated volume at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded Ever?	If this field displays True, flooding occurs at least once during the simulation at the currently highlighted element. This is a calculated results field and is not editable.

Manhole—Results: Flows Attributes

Table 15-44: Manhole—Result: Flow Attributes

Attribute	Description
Local Inflow?	This field displays True if there is a user-defined flow at the element, and False if there is not. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.
Total Surface Inflow	Total flow coming into manhole from runoff. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into manhole. This is a calculated results field and is not editable.

Catch Basin Attributes

The catch basin attributes comprise the following categories:

- “Catch Basin—General” on page 15-856
- “Catch Basin—Geometry” on page 15-857

Catch Basin—General

Table 15-45: Catch Basin—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Catch Basin—Geometry

Table 15-46: Catch Basin—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Catch Basin—Physical

Table 15-47: Catch Basin—Physical Attributes

Attribute	Description
Elevation (Ground)	Displays the ground elevation for the currently highlighted node.
Set Rim to Ground Elevation?	Enables or disables a data entry shortcut. If the value is **True**, the node's rim elevation is set equal to the ground elevation automatically. If this value is set to **False**, you must manually enter the rim elevation.
Elevation (Rim)	Lets you define the top elevation of the currently highlighted node. This elevation is typically flush with the ground surface. In some cases, the rim elevation may be slightly below the ground surface elevation (sunk) or slightly above the ground surface elevation (raised). You can only enter a value in this field if Set Rim to Ground Elevation? is set to **False**.
Elevation (Invert)	Lets you define the elevation at the bottom of the currently highlighted node.

Table 15-47: Catch Basin—Physical Attributes

Attribute	Description
Structure Shape Type	Lets you choose the shape of the currently highlighted element. You can select **Rectangular Structure** or **Circular Structure**. The value chosen here affects the availability of other fields.
Diameter	Lets you enter a value for the diameter of the currently highlighted element. This field is available only when the Structure Shape Type attribute is set to **Circular Structure**.
Length	Lets you enter a value for the length of the currently highlighted element. This field is available only when the Structure Shape Type attribute is set to **Rectangular Structure**.
Width	Lets you enter a value for the width of the currently highlighted element. This field is available only when the Structure Shape Type attribute is set to **Rectangular Structure**.

Catch Basin—Physical: Structure Losses

Table 15-48: Catch Basin—Structure Loss Attributes

Attribute	Description
Headloss Method	Lets you select the headloss method to use: **Absolute**, **HEC-22**, **Generic**, or **Standard**.
Absolute Headloss	Lets you enter a value for the headloss. This field is only available if you selected the **Absolute Headloss Method**.
Headloss Coefficient Stop	Lets you enter a value for the headloss coefficient at the stop section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient Start	Lets you enter a value for the headloss coefficient at the start section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient	Lets you enter a value for the headloss coefficient for the manhole. This field is only available if you selected the **Standard Headloss Method**.

Catch Basin—Physical: Surface Storage

Table 15-49: Manhole—Surface Storage Attributes

Attribute	Description
Surface Storage Type	Lets you choose how surface storage is handled. The value chosen here affects the availability of other fields: select **Default Storage Equation**, **Surface Depth-Area Curve**, **No Storage**, or **Ponded Area**.The Default Storage Equation uses the following formula: $\text{Area} = 0.262\,y - 0.068\,y^2 + 0.006\,y3$ $(y<4.0 \text{ ft})$ $\text{Area}=0.344$ $(y>4.0 \text{ ft})$ where Y is the depth above the ground.
Surface Depth-Area Curve	Lets you describe the volume of the surface storage by defining depth vs. area points. Click the **Ellipsis** (…) button in this field to open the Surface Depth-Area dialog box (see “Surface Depth-Area Curve Editor” on page 6-210). This field is available only when the Surface Storage Type attribute is set to **Surface Depth-Area Curve**.
Area (Constant Surface)	Lets you define the area in which ponding occurs at the currently selected element. It is available only when the Surface Storage Type attribute is set to **Ponded Area**.

Catch Basin—Sanitary Loading

Table 15-50: Catch Basin—Sanitary Loading Attributes

Attribute	Description
Sanitary Loads	Lets you define a sanitary (dry weather) flow collection for the selected catch basin. Click the **Ellipses (...)** button to display the Sanitary (Dry Weather) Flow Collection Editor, which lets you define collections of sanitary (dry weather) loads.

Catch Basin—SWMM Extended Data

Table 15-51: Catch Basin—SWMM Extended Data Attributes

Attribute	Description
Apply Treatment?	Lets you specify whether or not treatment is applied at the currently highlighted element. Select **True** to apply treatment. This field is only used during SWMM calculations.
Treatment	Lets you add a collection of pollutants and their associated treatment functions that will be applied at the currently highlighted element. This field is used only during SWMM calculations and is available only if Apply Treatment? is set to **True**. For more information, see “Adding Treatment to a Node” on page 6-321.
Pollutographs	Lets you define a pollutograph collection for the selected manhole. Click the **Ellipses (...)** button to display the Pollutograph Collection dialog box, which lets you add multiple pollutographs to the collection. For more information, see “Adding Pollutographs to a Node” on page 6-307.

Catch Basin—Active Topology

Table 15-52: Catch Basin—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Catch Basin—Inflow

Table 15-53: Catch Basin—Inflow Attributes

Attribute	Description
Inflow Collection	Contains an **Ellipsis** (…) button that lets you access the Inflow Collection dialog box for the associated element (see “Defining Inflow Collections” on page 7-378).

Catch Basin—Inlet

Table 15-54: Catch Basin—Inlet Attributes

Attribute	Description
Inlet Type	Lets you choose the type of element associated with the currently highlighted element: **Maximum Capacity**, **Inflow-Capture Curve**, or **Full Capture**. The value chosen here affects the availability of the other fields described below.
Flow (Maximum in)	Lets you define the maximum inflow accepted by the inlet associated with the currently highlighted element. This field is available only when the Inlet Type attribute is set to **Maximum Capacity**.
Inflow-Capture Curve	Lets you define inflow vs. capture percentage points for the currently highlighted element. Clicking the **Ellipsis** (…) button in the field opens the Inflow-Capture Curve dialog box (see “Inflow-Capture Curve Dialog Box” on page 6-208). This field is available only when the Inlet Type attribute is set to **Inflow-Capture Curve**.

Catch Basin—Results

Table 15-55: Catch Basin—Results Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the catch basin. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.

Table 15-55: Catch Basin—Results Attributes

Attribute	Description
Is Flooded?	If this field displays True, flooding occurs at the currently highlighted element during the current time step. For manhole and catch basin elements, this field will display True if the HGL goes above the rim elevation. This is a calculated results field and is not editable.
Total Outflow	Sum of all flows leaving catch basin. This is a calculated results field and is not editable.
Overflow	Flow that exits system during analysis. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Catch Basin—Results: Engine Parsing Attributes

Table 15-56: Catch Basin—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see “Branches” on page 14-722.

Catch Basin—Results: Extended Node Attributes

Table 15-57: Catch Basin—Results: Extended Node Attributes

Attribute	Description
Freeboard Height	Distance between the top of the catch basin and the water surface. This is a calculated results field and is not editable.
Depth (Flooding)	Depth between top of catch basin and the water surface when the catch basin is flooded. This is a calculated results field and is not editable.
Volume	Calculated volume at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded Ever?	If this field displays True, flooding occurs at least once during the simulation at the currently highlighted element. This is a calculated results field and is not editable.

Catch Basin—Results: Flows Attributes

Table 15-58: Catch Basin—Result: Flow Attributes

Attribute	Description
Local Inflow?	This field displays **True** if there is a user-defined flow at the element, and **False** if there is not. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.
Total Surface Inflow	Total flow coming into catch basin from runoff. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into catch basin. This is a calculated results field and is not editable.

Catch Basin—Results: Inlet Capture

Table 15-59: Catch Basin—Result: Inlet Capture Attributes

Attribute	Description
Flow (Captured)	Total amount of flow captured at the currently highlighted element. This is a calculated results field and is not editable.
Capture Efficiency	Percentage of flow that is captured at the currently highlighted element. This is a calculated results field and is not editable.

Outfall Attributes

The outfall attributes comprise the following categories:

- "Outfall—General" on page 15-865
- "Outfall—Geometry" on page 15-865
- "Outfall—Boundary Condition" on page 15-866
- "Outfall—Physical" on page 15-868
- "Outfall—Active Topology" on page 15-869
- "Outfall—SWMM Extended Data" on page 15-868
- "Outfall—Inflow" on page 15-869
- "Outfall—Results" on page 15-869
- "Outfall—Results: Flows" on page 15-870

Outfall—General

Table 15-60: Outfall—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see "Adding Hyperlinks to Elements" on page 6-324.

Outfall—Geometry

Table 15-61: Outfall—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Outfall—Boundary Condition

Table 15-62: Outfall—Boundary Condition Attributes

Attribute	Description
Boundary Condition Type	Lets you define what type of boundary condition the currently highlighted element is operating under. The value chosen here affects the availability of other fields. You can choose from: **Free Outfall** - No tailwater condition. No further input is necessary. **Time-Elevation Curve** - Enables Time-Elevation Curve field. **Elevation (User Defined Tailwater)** - Enables the User Defined Tailwater field. **Elevation-Flow Curve** - Enables Elevation Flow Curve field. **Boundary Element** - Enables Boundary Element field, allowing you to specify the element to which flow received by the outfall is discharged. **Normal** - No further input is necessary. Normal in this case means that the depth at the outlet is the normal depth for the last conduit or channel before the outlet. **Tidal** - Enables the Cyclic Time-Elevation Curve field.
Time-Elevation Curve	Lets you describe the elevation changes at the boundary condition over time. Click the **Ellipsis** (…) button in this field to open the Time-Elevation Curve dialog box (see “Time-Elevation Curve Dialog Box” on page 6-223). It is available only when the Boundary Condition Type attribute is set to **Time-Elevation Curve**.

Table 15-62: Outfall—Boundary Condition Attributes

Attribute	Description
Elevation (User Defined Tailwater)	Lets you enter a value for tailwater at the boundary. It is available only when the Boundary Condition Type is set to **User Defined Tailwater**.
Elevation-Flow Curve	Lets you define the elevation changes at the boundary condition over a range of flows. Click the **Ellipsis** (…) button in this field to open the Elevation-Flow Curve dialog box (see “Elevation-Flow Curve Dialog Box” on page 6-225). This property is available only when the Boundary Condition Type attribute is set to **Elevation-Flow Curve**.
Boundary Element	Lets you choose the boundary element for the currently highlighted element from all of the valid elements in the network. To use this feature, click **Select** in the Boundary Element field. Move the cursor over the drawing pane and click the element you want to select for the boundary. This property is available only when the Boundary Condition Type attribute is set to **Boundary Element**. **Tip: Press the Esc key to exit out of “Select” mode.**
Cyclic Time-Elevation Curve	Lets you describe the elevation changes at the boundary condition that repeat over time. Click the **Ellipsis** (…) button in this field to open the Cyclic Time-Elevation Curve dialog box (see “Time-Elevation Curve Dialog Box” on page 6-223). It is available only when the Boundary Condition Type attribute is set to **Tidal**.

Outfall—Physical

Table 15-63: Outfall—Physical Attributes

Attribute	Description
Elevation (Ground)	Lets you define the elevation of the currently highlighted element.
Set Invert Equal to Ground Elevation?	Sets the invert of the current element to the value specified in the Ground Elevation field.
Elevation (Invert)	Lets you define the elevation at the bottom of the currently highlighted node.

Outfall—SWMM Extended Data

Table 15-64: Outfall—SWMM Extended Data Attributes

Attribute	Description
Apply Treatment?	Lets you specify whether or not treatment is applied at the currently highlighted element. This field is only used during SWMM calculations.
Treatment	Lets you add a collection of pollutants and their associated treatment functions that will be applied at the currently highlighted element. This field is used only during SWMM calculations and is available only if **Apply Treatment?** is set to True. For more information, see "Adding Treatment to a Node" on page 6-321.
Pollutograph s	Lets you define a pollutograph collection for the selected manhole. Click the **Ellipses (...)** button to display the Pollutograph Collection dialog box, which lets you add multiple pollutographs to the collection. For more information, see "Adding Pollutographs to a Node" on page 6-307.

Outfall—Active Topology

Table 15-65: Outfall—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Outfall—Inflow

Table 15-66: Outfall—Inflow Attributes

Attribute	Description
Inflow Collection	Contains an **Ellipsis** (…) button that lets you access the Inflow Collection dialog box for the associated element (see "Defining Inflow Collections" on page 7-378).

Outfall—Results

Table 15-67: Outfall—Result Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the outfall. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.
Total Outflow	Sum of all flows leaving outfall. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Outfall—Results: Flows

Table 15-68: Outfall—Results: Flow Attributes

Attribute	Description
Local Inflow?	User defined inflow at outfall. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into outfall. This is a calculated results field and is not editable.

Pond Outlet Structure Attributes

The pond outlet structure attributes comprise the following categories:

- "Pond Outlet Structure—General" on page 15-871
- "Pond Outlet Structure—Geometry" on page 15-872
- "Pond Outlet Structure—Pond Outlet" on page 15-872
- "Pond Outlet Structure—Active Topology" on page 15-872
- "Pond Outlet Structure—Results" on page 15-873

Pond Outlet Structure—General

Table 15-69: Pond Outlet Structure—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see "Adding Hyperlinks to Elements" on page 6-324.

Pond Outlet Structure—Geometry

Table 15-70: Pond Outlet Structure—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Pond Outlet Structure—Pond Outlet

Table 15-71: Pond Outlet Structure—Pond Outlet Attributes

Attribute	Description
Upstream Pond	Lets you choose the upstream pond for the currently highlighted element from a list of all of the valid ponds in the network.
Has Control Structure?	Lets you choose whether the highlighted element has a control structure. If this field is set to **True**, the Control Structure field is activated.
Control Structure	Lets you define the components that make up the control structure. Click the **Ellipsis** (…) button in this field to open the Composite Outlet Structure dialog box (see “Composite Outlet Structures Dialog Box” on page 6-214). This field is active only when the Has Control Structure? attribute is set to **Yes**.

Pond Outlet Structure—Active Topology

Table 15-72: Pond Outlet Structure—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Pond Outlet Structure—Results

Table 15-73: Pond Outlet Structure—Result Attributes

Attribute	Description
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Cross Section Attributes

The cross section attributes comprise the following categories:

- “Cross Section—General” on page 15-874
- “Cross Section—Geometry” on page 15-874
- “Cross Section—Physical” on page 15-875
- “Cross Section—Active Topology” on page 15-877
- “Cross Section—Inflow” on page 15-878
- “Cross Section—Results” on page 15-878
- “Cross Section—Results: Engine Parsing Attributes” on page 15-878
- “Cross Section—Results: Flows” on page 15-879

Cross Section—General

Table 15-74: Cross Section—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Cross Section—Geometry

Table 15-75: Cross Section—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Cross Section—Physical

Table 15-76: Cross Section—Physical Attributes

Attribute	Description
Section Type	Lets you choose the cross-sectional shape of the currently highlighted element. You can select **Trapezoidal Cross Section** or **Irregular Channel**. The value chosen here affects the availability of other fields.
Station-Elevation Curve	Lets you define station-elevation points that describe the shape of the irregular channel. Click the **Ellipsis** (…) button in this field to open the Station-Elevation Curve dialog box (see "Station-Elevation Curve/Depth Dialog Box" on page 6-200). This field is available only when the Section Type attribute is set to **Irregular Channel**.
Elevations Modifier	The Elevations modifier is a constant value that will be added to each elevation value. This attribute is only used during SWMM calculations.
Stations Modifier	The Stations modifier is a factor by which the distance between each station will be multiplied when the transect data is processed by SWMM. Use a value of 0 if no such factor is needed. This attribute is only used during SWMM calculations.
Elevation (Invert)	Lets you define the invert elevation at the currently highlighted element. This attribute is active only when the Section Type attribute is set to **Trapezoidal Channel**.
Bottom Width	Lets you enter a value for the width at the base of the cross section of the currently highlighted element. This attribute is active only when the Section Type attribute is set to **Trapezoidal Channel**.
Slope (Left Side)	Lets you enter a value for the left slope of the cross section of the currently highlighted element. This attribute is active only when the Section Type attribute is set to **Trapezoidal Channel**.
Slope (Right Side)	Lets you enter a value for the right slope of the cross section of the currently highlighted element. This attribute is active only when the Section Type attribute is set to **Trapezoidal Channel**.
Height	Lets you enter a value for the height of the cross section of the currently highlighted element. This attribute is active only when the Section Type attribute is set to **Trapezoidal Channel**.

Table 15-76: Cross Section—Physical Attributes

Attribute	Description
Roughness Type	Lets you set the roughness method for the currently highlighted: **Single Manning's n**, **Manning's - Depth Curve**, or **Manning's n - Flow**. The value chosen here affects the availability of some fields in the Physical section of the Property Editor.
Material	Lets you enter the name of the material used. Alternatively, clicking the ellipsis button opens the Material Engineering Library, allowing you to select a pre-defined material. If a pre-defined material is chosen, the roughness value will change accordingly.
Left Bank Station	The distance values appearing in the Station/Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Right Bank Station	The distance values appearing in the Station/Elevation grid that mark the end of the left overbank and the start of the right overbank. Use 0 to denote the absence of an overbank.
Manning's n	Lets you enter a value for the Manning's roughness of the currently highlighted element. This attribute is active only when the Roughness Type attribute is set to **Single Manning's n**.

Table 15-76: Cross Section—Physical Attributes

Attribute	Description
Manning's n-Depth Curve	Lets you define points that describe a roughness-depth curve for the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the Manning's n-Depth Curve dialog box (see "Manning's n–Depth Curve Dialog Box" on page 6-202). This attribute is active only when the Roughness Type attribute is set to **Manning's n-Depth Curve**.
Manning's n-Flow	Lets you define points that describe a roughness-flow curve for the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the Manning's n-Flow Curve dialog box (see "Manning's n–Flow Curve Dialog Box" on page 6-203). This attribute is active only when the Roughness Type attribute is set to **Manning's n-Flow**.
Transition Type	When you connect a channel to a conduit at a cross-section node, a transition part is added between the channel cross-section and the conduit cross-section. This field lets you specify the transition type of the currently highlighted cross-section node. You can select either **Gradual** or **Abrupt**. If you select **Abrupt**, the top width of the channel cross-section node is used as the length of the transition part. If you select **Gradual**, the Transition Length field is made available. If the Transition Length is larger than the top width of the cross-section node, the Transition Length value is used as the length of the transition part. **Note: Transition Type and Transition Length are not used for cross-section nodes that connect two channels.**
Transition Length	Lets you define the length of the transition between a channel cross-section and a conduit cross-section. This field is available only if you select **Gradual** as the Transition Type.

Cross Section—Active Topology

Table 15-77: Cross Section—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Cross Section—Inflow

Table 15-78: Cross Section—Inflow Attributes

Attribute	Description
Inflow Collection	Contains an **Ellipsis** (…) button that lets you access the Inflow Collection dialog box for the associated element (see “Defining Inflow Collections” on page 7-378).

Cross Section—Results

Table 15-79: Cross Section—Result Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the cross section. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.
Overflow	Flow that exits system during analysis. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Cross Section—Results: Engine Parsing Attributes

Table 15-80: Cross Section—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see “Branches” on page 14-722.

Cross Section—Results: Flows

Table 15-81: Cross Section—Results: Flow Attributes

Attribute	Description
Local Inflow?	User defined inflow at cross section. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.

Pump Attributes

The pump attributes comprise the following categories:

- “Pump—General” on page 15-880
- “Pump—Geometry” on page 15-880
- “Pump—Physical” on page 15-881
- “Pump—Active Topology” on page 15-881
- “Pump—Results” on page 15-882
- “Pump—Results: Engine Parsing Attributes” on page 15-882

Pump—General

Table 15-82: Pump—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.
Downstream Link	Displays the ID of the downstream link element to which the pump is connected.

Pump—Geometry

Table 15-83: Pump—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Pump—Physical

Table 15-84: Pump—Physical Attributes

Attribute	Description
Suction Element ID	This field allows you to define the upstream, or suction-side node for the current pump. Select **Select Suction Element** from the drop-down menu; your mouse cursor changes to a Pick Element tool, allowing you to click on the desired element in the Drawing Pane. After you have selected the suction-side node, the pump and the node are connected by a dotted line in your model.
Elevation	Lets you define the elevation of the currently highlighted element.
Pumps	Lets you define pump settings for the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the Pumps dialog box (see “Pumps Dialog Box” on page 6-230).

Pump—Active Topology

Table 15-85: Pump—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Pump—Results

Table 15-86: Pump—Results

Attribute	Description
Head	Calculated pump head for the currently highlighted pump. This is a calculated results field, and as such is not editable.
Flow (Pump)	Calculated flow at the currently highlighted pump. This is a calculated results field, and as such is not editable.
Upstream Head	Calculated head on the upstream side of the currently highlighted pump. This is a calculated results field, and as such is not editable.
Downstream Head	Calculated head on the downstream side of the currently highlighted pump. This is a calculated results field, and as such is not editable.
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Pump—Results: Engine Parsing Attributes

Table 15-87: Pump—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see "Branches" on page 14-722.

Wet Well Attributes

The wet well attributes comprise the following categories:

- "Wet Well—General" on page 15-883
- "Wet Well—Geometry" on page 15-883
- "Wet Well—Physical" on page 15-884
- "Wet Well—Sanitary Loading" on page 15-885

Wet Well—General

Table 15-88: Wet Well—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see "Adding Hyperlinks to Elements" on page 6-324.

Wet Well—Geometry

Table 15-89: Wet Well—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Wet Well—Physical

The following illustration shows a typical wet well.

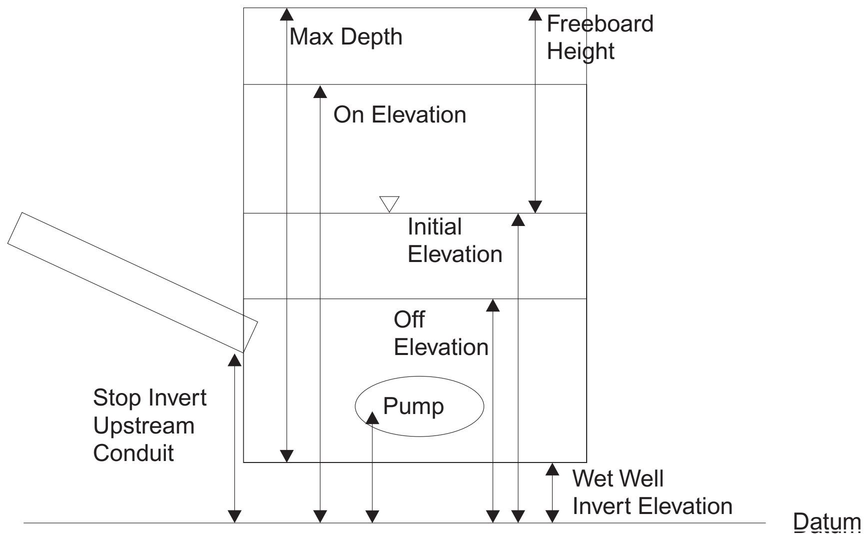


Table 15-90: Wet Well—Physical Attributes

Attribute	Description
Wet Well Volume Type	Lets you choose the method used to define volume for the currently highlighted element: **Wet Well Depth-Area Curve**, **Wet Well Constant Area**, or **Wet Well Area Function**. The value chosen here affects the availability of other attributes.
Wet Well Depth-Area Curve	Lets you define the size of the currently highlighted element by entering points in the depth vs. area table. Click the **Ellipsis** (…) button in this field to open the Wet Well Depth-Area Curve dialog box (see "Wet Well Depth-Area Curve Dialog Box" on page 6-228). This field is available only when **Wet Well Depth-Area Curve** is chosen as the Wet Well Volume Type.
Area	Lets you define the area of the currently highlighted element. It is available only when **Wet Well Constant Area** is chosen as the Wet Well Volume Type.
Max. Level	Lets you set the maximum depth for the wet well. It is available only when **Wet Well Constant Area** or **Wet Well Area Function** is chosen as the Wet Well Volume Type.

Table 15-90: Wet Well—Physical Attributes

Attribute	Description
Coefficient	Lets you set the coefficient of the area function for the currently highlighted element. It is available only when **Wet Well Area Function** is chosen as the Wet Well Volume Type.
Exponent	Lets you set the exponent of the area function for the currently highlighted element. It is available only when **Wet Well Area Function** is chosen as the Wet Well Volume Type.
Constant	Lets you set the constant of the area function for the currently highlighted element. It is available only when **Wet Well Area Function** is chosen as the Wet Well Volume Type.
Elevation (Invert)	Lets you define the invert, or bottom, elevation for the currently highlighted element.

Wet Well—Sanitary Loading

Table 15-91: Wet Well—Sanitary Loading Attributes

Attribute	Description
Sanitary Loads	Lets you define a sanitary (dry weather) flow collection for the selected wet well. Click the **Ellipses (...)** button to display the Sanitary (Dry Weather) Flow Collection Editor, which lets you define collections of sanitary (dry weather) loads.

Wet Well—Simulation Initial Condition

Table 15-92: Wet Well—Simulation Initial Condition Attributes

Attribute	Description
Initial Elevation Type	Lets you choose which type of initial condition type to be applied to the currently highlighted element: **Invert** or **User Defined Initial Elevation**. The value chosen here affects the availability of other attributes.
Elevation (Initial)	Lets you define the initial water surface elevation for the currently highlighted element. It is available only when the **User Defined Initial Elevation** value is selected for the Initial Elevation Type attribute.

Wet Well—SWMM Extended Data

Table 15-93: Wet Well—SWMM Extended Data Attributes

Attribute	Description
Evaporation Factor	Fraction of evaporation rate realized. This attribute is only used during SWMM calculations.
Apply Treatment?	Lets you specify whether or not treatment is applied at the currently highlighted element. Select **True** to apply treatment. This field is only used during SWMM calculations.
Treatment	Lets you add a collection of pollutants and their associated treatment functions that will be applied at the currently highlighted element. This field is used only during SWMM calculations and is available only if **Apply Treatment?** is set to True. For more information, see "Adding Treatment to a Node" on page 6-321.
Pollutograph s	Lets you define a pollutograph collection for the selected manhole. Click the **Ellipses (...)** button to display the Pollutograph Collection dialog box, which lets you add multiple pollutographs to the collection. For more information, see "Adding Pollutographs to a Node" on page 6-307.

Wet Well—Active Topology

Table 15-94: Wet Well—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Wet Well—Inflow

Table 15-95: Wet Well—Inflow Attributes

Attribute	Description
Inflow Collection	Contains an **Ellipsis** (…) button that lets you access the Inflow Collection dialog box for the associated element (see "Defining Inflow Collections" on page 7-378).

Wet Well—Results

Table 15-96: Wet Well—Results Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the wet well. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded?	If this field displays **True**, flooding occurs at the currently highlighted element during the current time step. For Ponds and Wet Wells, this field will display True if the water surface elevation goes above the highest elevation defined for the element. This is a calculated results field and is not editable.
Total Outflow	Sum of all flows leaving wet well. This is a calculated results field and is not editable.
Overflow	Flow that exits system during analysis. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants for the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Wet Well—Results: Extended Node

Table 15-97: Wet Well—Results: Extended Node Attributes

Attribute	Description
Freeboard Height	Distance between the top of the wet well and the water surface. This is a calculated results field and is not editable.
Depth (Flooding)	Depth between top of wet well and the water surface when the wet well is flooded. This is a calculated results field and is not editable.
Volume	Calculated volume at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded Ever?	If this field displays True, flooding occurs at least once during the simulation at the currently highlighted element. This is a calculated results field and is not editable.

Wet Well—Results: Flows

Table 15-98: Wet Well—Results: Flow Attributes

Attribute	Description
Local Inflow?	User defined inflow at wet well. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.
Total Surface Inflow	Total flow coming into wet well from runoff. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into wet well. This is a calculated results field and is not editable.

Catchment Attributes

The catchment attributes comprise the following categories:

Catchment—General

Table 15-99: Catchment—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Catchment—Geometry

Table 15-100: Catchment—Geometry Attributes

Attribute	Description
Geometry	Lets you view and edit the coordinates of points along a selected element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Polyline Vertices feature. For more information, see “Polyline Vertices Dialog Box” on page 6-198.
Scaled Area	Displays the scaled area of the currently highlighted element. This field in not editable.

Catchment—Catchment

Table 15-101: Catchment—Catchment Attributes

Attribute	Description
Area	Lets you define the area of the currently highlighted element. Clicking the ellipsis button in this field opens the Cn Area Collection dialog box (see “Defining CN Area Collections for Catchments” on page 7-387), allowing you to define SCS values and the pervious/impervious ratio for the currently highlighted element.
Outflow Node	Lets you choose the node to which flow flows from the currently highlighted element. To use this feature, click **Select** in the Outfall Node field. Move the cursor over the drawing pane and click the element you want to select for the outflow node.

Catchment—Runoff

Table 15-102: Catchment—Runoff Attributes

Attribute	Description
Runoff Method	Lets you set what type of runoff method the currently highlighted element uses. The value chosen here affects the availability of other fields. You can select **Unit Hydrograph**, **EPA-SWMM Runoff**, **User-Defined Hydrograph**, **None**, **or Modified Rational Method.**
Characteristic Width	Characteristic width of the currently highlighted catchment. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Storage (Impervious Depression)	Depth of depression storage on impervious portion of the catchment. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Storage (Pervious Depression)	Depth of depression storage on the pervious portion of the catchment. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Manning’s n (Impervious)	Manning's N for overland flow over the impervious portion of the currently highlighted catchment. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Manning’s n (Pervious)	Manning's N for overland flow over the pervious portion of the currently highlighted catchment. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Percent Impervious	Percent of land area which is impervious. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Slope	Average slope, in percent, of the currently highlighted catchment. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Percent Impervious Zero Storage	Percent of the impervious area with no depression storage. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Subarea Routing	Lets you define the type of subarea routing at the currently highlighted element. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.

Table 15-102: Catchment—Runoff Attributes

Attribute	Description
Percent Routed	Percent of runoff routed between subareas. This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff**.
Runoff Hydrograph	Lets you define time vs. flow points for the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the User Defined Hydrograph dialog box (see “Specifying a Time of Concentration (Tc) Method for a Catchment” on page 6-239). This attribute is available only when the Runoff Method attribute is set to **User Defined Hydrograph**.
Loss Method	Lets you define what type of loss method the currently highlighted element uses. You can select **fLoss**, **Green and Ampt**, **SCS CN**, or **(Generic) Horton**. The value chosen here affects the availability of other fields.
fLoss	Lets you define the initial infiltration rate at the time that infiltration begins for the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **fLoss**.
Capillary Suction	Lets you define the capillary suction value for the soil type associated with the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **Green and Ampt**.
Ks	Lets you set the saturated hydraulic conductivity (the rate at which water travels through the soil when it is saturated) for the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **Green and Ampt**.
Moisture Deficit	Lets you set the value for moisture deficit, which is the saturated moisture content minus the original moisture content, for the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **Green and Ampt**.
SCS CN	Lets you set a Cn value for the catchment. You can either type a value in the field or click the **Ellipsis** (…) button to open the Cn Area Collection dialog box (see “Defining CN Area Collections for Catchments” on page 7-387). This attribute is active only when the Loss Method attribute is set to **SCS CN**.
SCS CN (Composite)	The weighted CN value. This attribute is a calculated field and is active only when the Loss Method attribute is set to **SCS CN**.

Table 15-102: Catchment—Runoff Attributes

Attribute	Description
fc fo	Lets you define for the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **(Generic) Horton.** Lets you define the initial infiltration rate at the time that infiltration begins for the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **(Generic) Horton.**
Initial Abstraction K Recovery Constant	Lets you define the initial abstraction (Ia) for the currently highlighted element. The initial abstraction is a parameter that accounts for all losses prior to runoff and consists mainly of interception, infiltration, evaporation, and surface depression storage. This attribute is active only when the Loss Method attribute is set to **(Generic) Horton**. This field allows you to define the decay coefficient for the currently highlighted element. This attribute is active only when the Loss Method attribute is set to **(Generic) Horton**. Dry weather regeneration rate constant for the Horton curve. This attribute is active only when the Loss Method attribute is set to **(Generic) Horton**.
Maximum Volume Unit Hydrograph Method RTK Set	This attribute is available only when the Runoff Method attribute is set to **EPA-SWMM Runoff** and the Loss Method attribute is set to **(Generic) Horton**. Lets you define the type of unit hydrograph method the currently highlighted element uses. The value chosen here affects the availability of other fields. You can select **Generic Unit Hydrograph**, **SCS Unit Hydrograph**, or **RTK Unit Hydrograph**. Lets you assign an RTK table to the catchment. If there is no RTK table associated with your project, click the **Ellipsis** (…) button in this field to open the RTK Tables dialog box (“Creating an RTK Table and Assigning it to a Catchment” on page 7-418), where you can create new RTK tables. This field is available only when the Unit Hydrograph Method is set to **RTK Unit Hydrograph**.
Tc	Lets you set the time of concentration for the currently highlighted element. It is available only when the Unit Hydrograph Method attribute is set to **SCS Unit Hydrograph**.

Table 15-102: Catchment—Runoff Attributes

Attribute	Description
Tc (Composite)	The total summed Tc value derived from the individual Tc Methods in the Tc Data Collection.
Generic Unit Hydrograph	Lets you define time vs. flow points for the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the Unit Hydrograph Data dialog box ("Adding Generic Unit Hydrographs" on page 7-411). This field is available only when the Unit Hydrograph Method is set to **Generic Hydrograph**.
Convolution Time Step	Lets you define the time step for the currently highlighted element. It is available only when the Unit Hydrograph Method is set to **Generic Unit Hydrograph**.
Tc Data Collection	Contains individual Tc components for calculating the total composite Tc for the current catchment.
Rational C	The Rational C coefficient of the catchment section. This field is available when the Runoff Method attribute is set to **Modified Rational Method**.

Catchment—SWMM Extended Data

Table 15-103: Catchment—SWMM Extended Data Attributes

Attribute	Description
Curb Length	The length of curb on the catchment. This is used to normalize water quality calculations.
Land Uses	Collection of Land Uses applied to the catchment for use during Water Quality calculations. Clicking the **Ellipses (...)** button opens the Land Uses Collection dialog box, where you can add land use entries to the collection. This field is available when you select **EPA-SWMM Runoff** as the Runoff Method.
Initial Buildup Collection	Lets you specify the initial quantities of pollutant buildup over the catchment in a collection. Clicking the **Ellipses (...)** button opens the Initial Buildup Collection dialog box, where you can add initial buildup entries to the collection. This field is available when you select **EPA-SWMM Runoff** as the Runoff Method.
Apply Groundwater	Lets you apply groundwater to the catchment. If you select **True**, several additional fields become available.

Table 15-103: Catchment—SWMM Extended Data Attributes

Attribute	Description
Aquifer	Select the aquifer to apply to the catchment or select **Edit...** from the drop-down menu and click the **Ellipses (...)** button to define new aquifers in the SWMM Aquifers dialog box.
Surface Elevation	The elevation of ground surface for the subcatchment that sits above the aquifer.
Groundwater Flow Coefficient	The groundwater flow formula is described as: $$Q_{gw} = A1(H_{gw} - E)B1 - A2(H_{sw} - E)B2 + A3H_{gw}H_{sw}$$ where Q_{gw} = groundwater flow (cfs per acre or cms per hectare) H_{gw} = elevation of groundwater table (ft or m) H_{sw} = elevation of surface water at receiving node (ft or m) E = elevation of node invert (ft or m) The groundwater flow coefficient is the value of A1 in the groundwater flow formula.
Groundwater Flow Exponent	The value of B1 in the groundwater flow formula.
Surf. Water Flow Coefficient	Value of A2 in the groundwater flow formula.
Surf. Water Flow Exponent	Value of B2 in the groundwater flow formula.
Surface-GW Interaction Coefficient	Value of A3 in the groundwater flow formula.
Fixed Surf. Water Depth	This is the fixed depth of surface water at the receiving node (in feet or meters). This is set to zero if the surface water depth varies as computed by flow routing.
Receiving Node	Lets you select the receiving node by clicking on the desired node in the drawing pane.

Catchment—Active Topology

Table 15-104: Cross Section—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Catchment—Inflow

Table 15-105: Catchment—Inflow Attributes

Attribute	Description
Inflow Collection	Contains an **Ellipsis** (…) button that lets you access the Inflow Collection dialog box for the associated element (see “Defining Inflow Collections” on page 7-378).

Catchment—Rainfall

Table 15-106: Catchment—Rainfall Attributes

Attribute	Description
Use Local Rainfall?	Lets you define whether or not the currently highlighted element uses local rainfall, and if so, which type. Select **True** to display the Local Storm Event field. The value chosen here affects the availability of other fields.
Local Storm Event	Lets you create or select a storm event from those previously created. Select a storm event from the drop-down list in this field, select the **Ellipsis** (…) item in the drop down list, or select the **New** (…) item to open the Rainfall Curves dialog box (see “Adding Storm Events” on page 7-395) and edit an existing storm event or create a new one. This field is available only when the Use Local Rainfall? attribute is set to **True**.
Return Event	Lets you define the frequency of the storm at the currently highlighted element. This attribute is available only when the Use Local Rainfall? attribute is set to **True**.

Catchment—Results

Table 15-107: Catchment—Result Attributes

Attribute	Description
Total Outflow	Sum of all flows leaving catchment. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants assigned to the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Catchment—Results: Extended Catchment

Table 15-108: Catchment—Results: Extended Catchment Attributes

Attribute	Description
Precipitation (Cumulative)	Total rainfall on catchment up until current time step. This is a calculated results field, and as such is not editable.
Loss (Cumulative)	Total amount of rainfall that has infiltrated into catchment up until current time step. This is a calculated results field, and as such is not editable.
Precipitation (Incremental)	The depth of precipitation over the catchment for the current time step.
Loss (Incremental)	The amount of precipitation that is lost or absorbed over the catchment for the current time step.

Catchment—Results: Flows

Table 15-109: Catchment—Results: Flow Attributes

Attribute	Description
Local Inflow?	User defined inflow at catchment. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into catchment. This is a calculated results field and is not editable.

Pond Attributes

The pond attributes comprise the following categories:

- "Pond—General" on page 15-899
- "Pond—Geometry" on page 15-899
- "Pond—Physical" on page 15-900
- "Pond—Simulation Initial Condition" on page 15-901
- "Pond—SWMM Extended Data" on page 15-902
- "Pond—Active Topology" on page 15-902
- "Pond—Inflow" on page 15-902
- "Pond—Results" on page 15-903
- "Pond—Results: Engine Parsing Attributes" on page 15-903
- "Pond—Results: Extended Node" on page 15-904
- "Pond—Results: Flows" on page 15-904

For an overview of the physical characteristics of ponds, see "Physical Characteristics of Ponds" on page 6-244

Pond—General

Table 15-110: Pond—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Pond—Geometry

Table 15-111: Pond—Geometry Attributes

Attribute	Description
Geometry	Lets you view and edit the coordinates of points along a selected element. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Polyline Vertices feature. For more information, see “Polyline Vertices Dialog Box” on page 6-198.
Scaled Area	Displays the scaled area of the currently highlighted element. This field in not editable.

Pond—Physical

Table 15-112: Pond—Physical Attributes

Attribute	Description
Volume Type	Lets you select the volume type that gets used: **Elevation-Area Curve**, **Elevation-Volume Curve**, **Pipe Volume**, **Functional**. The value chosen here affects the availability of other fields.
Elevation-Area Curve	Lets you define points to describe the shape of the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the Elevation-Area Curve dialog box (“Elevation-Area Curve Dialog Box” on page 6-249). This attribute is active only when the Volume Type attribute is set to **Elevation-Area Curve**.
Elevation-Volume Curve	Lets you define points to describe the shape of the currently highlighted element. Click the **Ellipsis** (…) button in this field to open the Elevation-Volume Curve dialog box (“Elevation-Volume Curve Dialog Box” on page 6-251). This attribute is active only when the Volume Type attribute is set to **Elevation-Volume Curve**.
Number of Barrels	Lets you define the number of barrels that comprise the currently highlighted element. Note that the diameter, length, and invert values are applied to each barrel. This field is available only when the Volume Type attribute is set to **Pipe Volume**.
Length	Lets you define the length of the currently highlighted element. It is available only when the Volume Type attribute is set to **Pipe Volume**.
Invert (Start)	Lets you enter a value for the start, or upstream, invert of the currently highlighted element. It is available only when the Volume Type attribute is set to **Pipe Volume**.
Invert (Stop)	Lets you enter a value for the stop, or downstream, invert of the currently highlighted element. It is available only when the Volume Type attribute is set to **Pipe Volume**.
Pipe Diameter	Lets you define the diameter of the currently highlighted element. It is available only when the Volume Type attribute is set to **Pipe Volume**.

Table 15-112: Pond—Physical Attributes

Attribute	Description
Pond Coefficient	“A” value in expression A*Depth^B+C for Depth in feet. This field is available only when the Functional Volume Type is chosen.
Pond Exponent	“B” value in expression A*Depth^B+C for Depth in feet. This field is available only when the Functional Volume Type is chosen.
Elevation (Invert)	The invert elevation of the currently highlighted pond.
Depth (Maximum)	Lets you set the maximum depth of the pond.
Pond Constant	“C” value in expression A*Depth^B+C for Depth in feet. This field is available only when the Functional Volume Type is chosen.

Pond—Simulation Initial Condition

Table 15-113: Pond—Simulation Initial Condition Attributes

Attribute	Description
Initial Elevation Type	Lets you choose which type of initial condition type to be applied to the currently highlighted element: **Invert** or **User Defined Initial Elevation**. The value chosen here affects the availability of other attributes.
Elevation (Initial)	Lets you define the initial water surface elevation for the currently highlighted element. It is available only when the **User Defined Initial Elevation** value is selected for the Initial Elevation Type attribute.

Pond—SWMM Extended Data

Table 15-114: Pond—SWMM Extended Data Attributes

Attribute	Description
Evaporation Factor	Fraction of evaporation rate realized. This attribute is only used during SWMM calculations.
Apply Treatment?	Lets you specify whether or not treatment is applied at the currently highlighted element. Select **True** to apply treatment. This field is only used during SWMM calculations.
Treatment	Lets you add a collection of pollutants and their associated treatment functions that will be applied at the currently highlighted element. This field is used only during SWMM calculations and is available only if **Apply Treatment?** is set to True. For more information, see “Adding Treatment to a Node” on page 6-321.
Pollutograph s	Lets you define a pollutograph collection for the selected manhole. Click the **Ellipses (...)** button to display the Pollutograph Collection dialog box, which lets you add multiple pollutographs to the collection. For more information, see “Adding Pollutographs to a Node” on page 6-307.

Pond—Active Topology

Table 15-115: Pond—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Pond—Inflow

Table 15-116: Pond—Inflow Attributes

Attribute	Description
Inflow Collection	Contains an **Ellipsis** (…) button that lets you access the Inflow Collection dialog box for the associated element (see “Defining Inflow Collections” on page 7-378).

Pond—Results

Table 15-117: Pond—Results Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the pond. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded?	If this field displays **True**, flooding occurs at the currently highlighted element during the current time step. For Ponds and Wet Wells, this field will display True if the water surface elevation goes above the highest elevation defined for the element. This is a calculated results field and is not editable.
Total Outflow	Sum of all flows leaving pond. This is a calculated results field and is not editable.
Overflow	Flow that exits system during analysis. This is a calculated results field and is not editable.
Pollutants	Lets you view calculated results for pollutants assigned to the node. Clicking the **Ellipses (...)** button displays the Pollutants Results dialog box.

Pond—Results: Engine Parsing Attributes

Table 15-118: Catch Basin—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see “Branches” on page 14-722.

Pond—Results: Extended Node

Table 15-119: Pond—Results: Extended Node Attributes

Attribute	Description
Freeboard Height	Distance between the top of the pond and the water surface. This is a calculated results field and is not editable.
Depth (Flooding)	Depth between top of pond and the water surface when the pond is flooded. This is a calculated results field and is not editable.
Volume	Calculated volume at the currently highlighted element. This is a calculated results field and is not editable.
Is Flooded Ever?	If this field displays True, flooding occurs at least once during the simulation at the currently highlighted element. This is a calculated results field and is not editable.

Pond—Results: Flows

Junction Chamber Attributes

Table 15-120: Wet Well—Results: Flow Attributes

Attribute	Description
Local Inflow?	User defined inflow at pond This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.
Total Surface Inflow	Total flow coming into pond from runoff. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into pond. This is a calculated results field and is not editable.

The junction chamber attributes comprise the following categories:

- "Junction Chamber—General" on page 15-906
- "Junction Chamber—Geometry" on page 15-906
- "Junction Chamber—Physical" on page 15-907
- "Junction Chamber—Physical: Structure Losses" on page 15-907
- "Junction Chamber—Active Topology" on page 15-908
- "Junction Chamber—Results" on page 15-908
- "Junction Chamber—Results: Engine Parsing Attributes" on page 15-908
- "Junction Chamber—Results: Flows" on page 15-909

Junction Chamber—General

Table 15-121: Junction Chamber—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Junction Chamber—Geometry

Table 15-122: Junction Chamber—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Junction Chamber—Physical

Table 15-123: Junction Chamber—Physical Attributes

Attribute	Description
Diameter	Lets you enter a value for the diameter of the currently highlighted element.
Elevation (Ground)	Lets you enter a value for the ground elevation for the currently highlighted node.
Elevation (Top)	Lets you enter a value for the top elevation for the currently highlighted node.
Elevation (Bottom)	Lets you enter a value for the bottom elevation for the currently highlighted node.

Junction Chamber—Physical: Structure Losses

Table 15-124: Junction Chamber—Physical: Structure Losses Attributes

Attribute	Description
Headloss Method	Lets you select the headloss method to use: **Absolute**, **HEC-22**, **Generic**, or **Standard**.
Absolute Headloss	Lets you enter a value for the headloss. This field is only available if you selected the **Absolute Headloss Method**.
Headloss Coefficient Start	Lets you enter a value for the headloss coefficient at the start section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient Stop	Lets you enter a value for the headloss coefficient at the stop section. This field is only available if you selected the **Generic Headloss Method**.
Headloss Coefficient	Lets you enter a value for the headloss coefficient for the manhole. This field is only available if you selected the **Standard Headloss Method**.

Junction Chamber—Active Topology

Table 15-125: Junction Chamber—Active Topology

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Junction Chamber—Results

Table 15-126: Junction Chamber—Results Attributes

Attribute	Description
Hydraulic Grade	Representative calculated hydraulic grade at the junction chamber. This is a calculated results field and is not editable.
Depth (Node)	Depth of the water at the currently highlighted element. This is a calculated results field and is not editable.
Maximum HGL	
TimeToMaximumHGL	

Junction Chamber—Results: Engine Parsing Attributes

Table 15-127: Junction Chamber—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see “Branches” on page 14-722.

Junction Chamber—Results: Flows

Table 15-128: Junction Chamber—Results: Flow Attributes

Attribute	Description
Flow (Total In)	Sum of all flows coming into junction chamber. This is a calculated results field and is not editable.

Pressure Junction Attributes

The pressure junction attributes comprise the following categories:

Pressure Junction—General

Table 15-129: Pressure Junction—General Attributes

Attribute	Description
ID	Displays the unique identifier for the currently highlighted element. The ID is automatically assigned to each discrete block of data in the project data store (the .mdb file) by the program. It is not editable.
Label	Displays the label for the currently highlighted element. The Label can be edited.
Notes	Lets you enter descriptive text that is associated with the currently highlighted element.
Hyperlinks	Lets you add, edit, delete, and view external files that are associated with the element using the Hyperlinks feature. Click in the field to display an Ellipsis (…) button, and click the **Ellipsis** (…) button to use the Hyperlinks feature. For more information about the Hyperlink Manager, see “Adding Hyperlinks to Elements” on page 6-324.

Pressure Junction—Geometry

Table 15-130: Pressure Junction—Geometry Attributes

Attribute	Description
X	Contains the coordinate of the currently highlighted element along the X (horizontal) axis.
Y	Contains the coordinate of the currently highlighted element along the Y (vertical) axis

Pressure Junction—Physical

Table 15-131: Pressure Junction—Physical Attributes

Attribute	Description
Elevation (Ground)	Lets you enter a value for the ground elevation of the currently highlighted element.
Elevation	Lets you enter a value for the elevation of the currently highlighted element.

Pressure Junction—Sanitary Loading

Pressure Junction—Sanitary Loading Attributes

Attribute	Description
Sanitary Loads	Lets you define sanitary loads for the pressure junction. Clicking the **Ellipse** (...) button displays the Sanitary (Dry Weather) Flow Collection Editor, which lets you define Hydrographs, Unit Loads, and Pattern Loads.

Pressure Junction—Active Topology

Table 15-132: Pressure Junction—Active Topology Attributes

Attribute	Description
Is Active	Lets you choose whether or not the corresponding element is active in the current alternative. Select **True** to make the element active in the current alternative.

Pressure Junction—Inflow

Table 15-133: Pressure Junction—Inflow Attributes

Attribute	Description
Inflow Collection	Lets you define an inflow collection for the selected pressure junction. Clicking the Ellipse (...) button displays the Inflow Collection Editor, which lets you create collections containing Fixed Inflows, Hydrograph Inflows, and Pattern Inflows.

Pressure Junction—Results

Table 15-134: Pressure Junction—Results Attributes

Attribute	Description
Total Outflow Maximum HGL TimeToMaximumHGL	Sum of all flows leaving the pressure junction. This is a calculated results field and is not editable.
Pressure HGL	The pressure value at the junction for the current time step.

Pressure Junction—Results: Engine Parsing Attributes

Table 15-135: Pressure Junction—Results: Engine Parsing Attributes

Attribute	Description
Branch	Returns the ID of the branch which the current element is part of. The branch is determined during the calculations using the implicit engine. This is a calculated results field and is not editable. For more information, see "Branches" on page 14-722.

Pressure Junction—Results: Flows

Table 15-136: Pressure Junction—Results: Flow Attributes

Attribute	Description
Local Inflow?	User defined inflow at the pressure junction. This is a calculated results field and is not editable.
Flow (Total In)	Sum of all flows coming into the pressure junction. This is a calculated results field and is not editable.
Total Local Inflow	Sum of all user defined inflows. This is a calculated results field and is not editable.

Chapter

Frequently Asked Questions 16

This chapter contains answers to some of the frequently asked questions about using Bentley SewerGEMS V8*i*.

The following FAQs are included:

What Project Files Does Bentley SewerGEMS V8*i* Maintain?

These are filename extensions of the files associated with a Bentley SewerGEMS V8*i* project:

- **Project files**—.swg, .dwh, .mdb
- **Results files**—.out (model output), .bin, .gut, .eqt, .hyg (hydrology output), .cul, .swm (SWMM data file). There is one set per scenario.

The .mdb file is the most important file because it contains all modeling data, and includes everything needed to perform a calculation. This file is an open access database format file and can be viewed and edited.

The .swg file contains data such as annotation and color-coding definitions. This file is an open .xml format file and can be viewed and edited.

The .dwh file is used by the Stand-Alone mode only, and contains the drawing. This is a binary format file.

Whenever you create a file with a name of Filename, three files are created:

- Filename.mdb
- Filename.swg
- Filename.dwh

When you move a file and speed is important, it is best to place all three in a .zip file, which will greatly reduce file size. Then copy or move the zip file and unzip it.

Every time you save the file, you not only overwrite the previous version but you also convert the old files to .bak files. Each file will have it's original file name plus the extension .bak, for example Filename.swg.bak.

If your file becomes corrupted or you accidentally delete it, you can go back to the previous version by simply removing the .bak suffix from the file name and using that file (after deleting the corrupted file if necessary).

You can usually delete the results files because they you can regenerated them by rerunnng your model.

To send your model to other Bentley SewerGEMS V8*i* users:

To send your SewerGEMS V8*i* model to other SewerGEMS V8*i* users, you only need to send the project files (.swg, .dwh, and .mdb). The results files will be recreated when the scenarios associated with the model are calculated.

What Kind of Graphs Can I Create and How Do I Create Them?

In Bentley SewerGEMS V8*i*, there are two kinds of graphical displays: profiles and graphs (charts). Profiles show cross sections of the network to scale along some path through the network and are described in “Using Profiles” on page 10-548. Graphs are plots of some attribute or group of attributes vs. time at one or more elements. Profiles and graphs can only be drawn for results of model runs; in order to view a graph or profile, you must first successfully compute your model (run the model successfully).

- For details on how create a graph, see “Creating a Graph” on page 10-578. Bentley SewerGEMS V8*i* contains a powerful graph editing tool to help you customize your graphs.
- For more information about this tool, see “Graph Dialog Box” on page 10-579.

How Do I Enter the Scale of a Background Image If it is a File Type without an Inherent Scale?

Image file formats such as BMP or JPG do not have scales inherently associated with them.

When you create a new background layer (by clicking New File in the Background Layer manager, then selecting the image file to use), Bentley SewerGEMS V8*i* displays the Image properties dialog box (see “Image Properties Dialog Box” on page 10-533). In the Image Properties dialog box, you can specify the true distances of the corners of the drawing from the bottom left corner, which in the image is considered by default to be 0,0. If you know the coordinates of each corner of the map, you can enter them in the Image Properties dialog box as well.

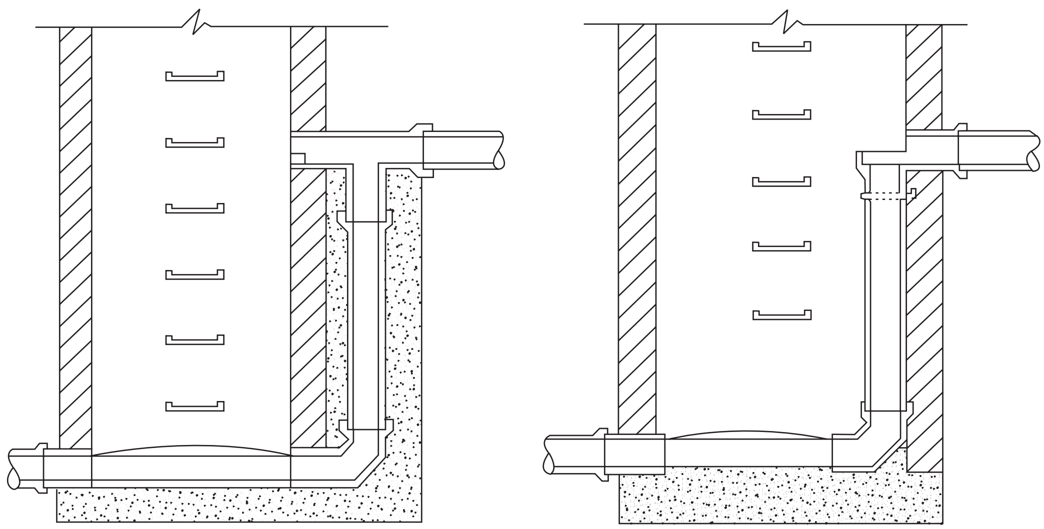
The image scale is used internally by Bentley SewerGEMS V8*i*, while the drawing scale represents the actual length and is used to determine scaled lengths in Bentley SewerGEMS V8*i*.

What is the Difference Between a Drop Manhole and a Regular Manhole?

A drop manhole is used in areas with a steep slope when one or more of the inlet pipes has an invert elevation significantly higher than the invert of the outlet pipe. Typically the invert elevation of the "stop" end of the inlet pipe is set to the invert elevation of the manhole. However, in the case of a drop manhole, the stop invert of the pipe is not set to the manhole invert elevation but is at a significantly higher elevation.

The following illustration shows a drop manhole.

Figure 16-1: Drop Manhole



How Do I Manage the Size of My Database Files?

If your database files are getting too large, you can compress them by compacting the database. This operation eliminates much of the wasted space in your file. To do this, select **Tools > Database Utilities > Compact Database**. To be able to use this feature, you must select the Compact Database Enabled option on the Global tab of the Options dialog box.

For more information, see “Tools Menu” on page 2-22 and “Options Dialog Box - Global Tab” on page 4-147.

Appendix

About Haestad Methods

A

Bentley Systems, Incorporated provides software for the lifecycle of the world's infrastructure. The company's comprehensive portfolio for the building, plant, civil, and geospatial vertical markets spans architecture, engineering, construction (AEC) and operations. Bentley is the leading provider of AEC software to the Engineering News-Record Design 500 and major owner-operators. For more information, visit the Bentley Web site at http://www.bentley.com.

Bentley Systems, Inc. offers software solutions to civil engineers throughout the world for analyzing, modeling, and designing all sorts of hydrologic and hydraulic systems, from municipal water and sewer systems to stormwater ponds, open channels, and more. With point-and-click data entry, flexible units, and report-quality output, Bentley Systems, Inc. is the ultimate source for your modeling needs.

In addition to the ability to run in Stand-Alone mode with a CAD-like interface, four of our products—WaterCAD, StormCAD, SewerCAD, and Bentley SewerGEMS V8*i*—can be totally integrated within AutoCAD. These three programs also share numerous powerful features, such as scenario management, unlimited undo/redo, customizable tables for editing and reporting, customizable GIS, database and spreadsheet connection, and annotation.

Be sure to contact us or visit our Web site at http://www.bentley.com to find out about our latest software, books, training, and open houses.

Software

Bentley Systems, Inc. software includes:

- "CivilStorm"
- "WaterGEMS"
- "WaterCAD"
- "SewerCAD"
- "StormCAD"

- “PondPack”
- “FlowMaster”
- “CulvertMaster”
- “HAMMER”
- “GISConnect”
- “GISConnect”

CivilStorm

CivilStorm revolutionizes municipal stormwater management. Whether your concern is a stormwater master plan, localized flooding, GASB34 requirements, water quality BMPs, NPDES permitting, or just simply being able to do faster and smarter designs every day, CivilStorm fits your needs. It is the only commercially available software package that lets you analyze all your system elements in one package. CivilStorm also gives you the ability to perform analyses using either the SWMM algorithm or CivilStorm’s own implicit solution of full Saint-Venant equations.

CivilStorm provides numerical solutions for the toughest interconnected pipe, pond, and open channel networks, and provides stunning graphics and reporting tools for visualizing your storm systems in action.

Use CivilStorm For:

- Comprehensive Stormwater Master Plans
- Watershed-Based Master Planning
- Analysis of Open-channel, Closed-conduit and Combination systems
- Floodplain Studies
- Complex Flow Regime Analysis
- Water Quality Assessments
- Integrated Stormwater Quantity and Quality Assessments
- NPDES Permitting

CivilStorm can be run in a MicroStation integrated interface, a Stand-Alone graphical user interface, or an AutoCAD integrated interface.

WaterGEMS

WaterGEMS brings the concept of water modeling and GIS integration to the next level. It is the only water-distribution modeling software that provides full, completely seamless integration with GIS applications. Now the combined functionality of WaterCAD and GIS can be utilized simultaneously, synthesizing the distinct advantages of each application to create a modeling tool with an unprecedented level of freedom, power, efficiency, and usability.

You can create, display, edit, run, map, and design water models from within the GIS environment, and view the results of the simulations as native GIS maps or with traditional Haestad Methods modeling tools. These abilities, in conjunction with the cross-product functionality provided by the core Unified Data and Object Model architecture, provide a powerful cutting-edge solution for your modeling projects.

WaterGEMS works within your choice of environments: MicroStation, ArcView, ArcEdit, ArcInfo, AutoCAD, or the standalone WaterGEMS Modeler interface.

WaterCAD

WaterCAD is the definitive model for complex pressurized-pipe networks, such as municipal water-distribution systems. You can use WaterCAD to perform a variety of functions, including steady-state and extended-period simulations of pressure networks with pumps, tanks, control valves, and more.

WaterCAD's abilities also extend into public safety and long-term planning issues, with extensive water quality features, automated fire protection analyses, comprehensive scenario management, and enterprise-wide data-sharing capabilities.

WaterCAD is available with your choice of a MicroStation integrated interface, Stand-Alone graphical user interface, and an AutoCAD integrated interface.

SewerCAD

SewerCAD is a powerful design and analysis tool for modeling sanitary sewage collection and pumping systems. With SewerCAD, you can develop and compute sanitary loads, track and combine loads from dry-weather and wet-weather sources. You can also simulate the hydraulic response of the entire system (gravity collection and pressure force mains), observe the effects of overflows and diversions, and even automatically design selected portions of the system. Output covers everything from customizable tables and detailed reports to plan and profile sheets.

StormCAD

StormCAD is a highly efficient model for the design and analysis of storm sewer collection systems. From graphical layout and intelligent network connectivity to flexible reports and profiles, StormCAD covers all aspects of storm-sewer modeling.

Surface inlet networks are independent of pipe connectivity and inlet hydraulics conform to FHWA HEC-22 methodologies. Gradually varied flow algorithms and a variety of popular junction-loss methods are the foundation of StormCAD's robust gravity piping computations, which handle everything from surcharged pipes and diversions to hydraulic jumps.

PondPack

PondPack is a comprehensive, Windows-based hydrologic modeling program that analyzes a tremendous range of situations, from simple sites to complex networked watersheds. PondPack analyzes pre- and post-developed watershed conditions and estimates required storage ponds. PondPack performs interconnected pond routing, and also computes outlet rating curves with tailwater effects, multiple outfalls, pond infiltration, and pond-detention times.

PondPack builds customized reports organized by categories, automatically creating section and page numbers, tables of contents, and indexes. You can quickly create an executive summary for an entire watershed or build an elaborate drainage report showing any or all report items. Graphical displays, such as watershed diagrams, rainfall curves, and hydrographs, are fully compatible with other Windows software.

FlowMaster

FlowMaster is an efficient program for the design and analysis of a wide variety of hydraulic elements, such as pressure pipes, open channels, weirs, orifices, and inlets. FlowMaster's Hydraulics Toolbox can create rating tables and performance curves for any variables, using popular friction methods. Inlet calculations follow the latest FHWA guidelines, and weighting of irregular section roughness can be based on any popular techniques.

CulvertMaster

CulvertMaster helps engineers design new culverts and analyze existing culvert hydraulics, from single-barrel crossings to complex multibarrel culverts with roadway overtopping. CulvertMaster computations use *HDS No. 5* methodologies, allowing you to solve for whatever hydraulic variables you do not know, such as culvert size, peak discharge, and headwater elevation. Output capabilities include comprehensive detailed reports, rating tables, and performance curves.

HAMMER

HAMMER is the premier software in the world for analyzing hydraulic transients, surge control devices, and water hammer effects. HAMMER models any hydraulic element, transient source or surge protection devices, including:

- Transients for flow, head, or entrained vapor
- Pressurized pipelines and networks
- Ingress of contaminants into pipe networks
- Surcharged sewers or storage tunnels
- Pump start and shut down scenarios
- Flow shifting via pumps or valves
- Power or pump failure
- Rapid valve closure
- Catastrophic pipe or pump breaks

HAMMER can easily import steady-state model results from industry-standard models such as WaterCAD, WaterGEMS, EPANET, or EXTRAN (for line filling). Users can also build their own models using the advanced graphical interface or robust database connections.

GISConnect

Run your GIS inside AutoCAD®. GISConnect is the long anticipated product that brings together the data management power of ArcGIS® and the drawing capabilities of AutoCAD®. Master your company's existing CAD expertise to deliver the GIS solutions that your clients demand.

Bentley Institute Press

Bentley Institute Press provides civil engineering professionals with affordable, quality reference and textbooks dedicated to the practical application of engineering theory to hydraulics and hydrology. Bentley Institute Press publications include:

References and Textbooks:	Authored by industry-recognized experts, Bentley Institute Press offers a complete line of reference books for use in both academic and professional settings.
Technical Journals:	With an eye towards computer technology, journals like "Current Methods" address the latest innovations in water-resources modeling and practical modeling case studies, as well as offering credit towards certification.
Independent Papers:	Bentley Institute Press also provides funding for engineers to write case studies of their projects, with potential publication in a variety of industry journals and magazines.

Books from Bentley Institute Press:

Advanced Water Distribution Modeling and Management, first edition
Haestad, Walski, Chase, Savic, Grayman, Beckwith, and Koelle

Computer Applications in Hydraulic Engineering, fifth edition
Haestad, Walski, Barnard, Durrans, and Meadows

Floodplain Modeling Using HEC-RAS, first edition
Haestad, Dyhouse, Hatchett, and Benn

Proceedings of the First Annual Water Security Summit, first edition
Haestad

Stormwater Conveyance Modeling and Design, first edition
Haestad and Durrans

Wastewater Collection System Modeling and Design, first edition
Haestad, Walski, Barnard, Merritt, Harold, Walker, and Whitman

Water Distribution Modeling, first edition
Haestad, Walski, Chase, and Savic

To order or to receive additional information on these or any other Bentley Institute Press titles, please call 800-727-6555 (U.S. and Canada) or +1-203-755-1666 (worldwide) or visit www.bentley.com/books.

Training

The Bentley Institute manages professional training programs to ensure consistent, high quality, user training for a variety of Bentley products and varying levels of application experience. Bentley Institute training is developed to maximize your productivity by using examples relevant to your day-to-day project efforts. Training is developed concurrently with software applications to provide knowledge of the latest tools and features. Additionally, all Bentley Institute faculty meet rigorous certification requirements.

The Bentley Institute offers complete training for Haestad Methods products.These training programs are famous for efficiently and effectively teaching engineers how to apply hydraulic theory and state-of-the-art software to real-world design situations.

- Modelers can become certified in a variety of water-related fields, through an assortment of teaching methods including:
- JumpStart Seminars
- Comprehensive Workshops
- Publication-Based Programs

To obtain more information about Bentley Systems, Inc. certification programs or to see upcoming events in a city near you, visit http://www.bentley.com.

Accreditations

Bentley Systems has achieved the highest levels of accreditation from both the International Association for Continuing Education and Training (IACET) and the Professional Development Registry for Engineers and Surveyors (PDRES). In addition to our own prestigious certifications, these endorsements enable modelers to earn Continuing Education Units (CEUs) and Professional Development Hours (PDHs) for their satisfactory participation in various training and educational programs.

Appendix B
Reference Tables

Manning's n Coefficients

Table B-1: Manning's n Coefficient Table

Lined Channels		Manning's n
Concrete, with surfaces as indicated	Formed, no finish	0.013 – 0.017
	Trowel finish	0.012 – 0.014
	Float finish	0.013 – 0.015
	Float finish, gravel on bottom	0.015 – 0.017
	Gunite, good section	0.016 – 0.019
	Gunite, wavy section	0.018 – 0.022
Concrete, bottom float-finished, sides as indicated	Dressed stone in mortar	0.015 – 0.017
	Random stone in mortar	0.017 – 0.020
	Cement rubble masonry	0.020 – 0.025
	Cement rubble masonry, plastered	0.016 – 0.020
	Dry rubble (riprap)	0.020 – 0.030
Gravel bottom, sides as indicated	Formed concrete	0.017 – 0.020

Table B-1: Manning's n Coefficient Table (Cont'd)

	Random stone in mortar	0.020 – 0.023
	Dry rubble (riprap)	0.023 – 0.033
	Brick	0.014 – 0.017
Asphalt	Smooth	0.013
	Rough	0.016
	Wood, planed, clean	0.011 – 0.013
Concrete-lined excavated rock	Good section	0.017 – 0.020
	Irregular section	0.022 – 0.027
Unlined Channels		**Manning's n**
Earth, uniform section	Clean, recently completed	0.016 – 0.018
	Clean, after weathering	0.018 – 0.020
	With short grass, few weeds	0.022 – 0.027
	In gravely soil, uniform section, clean	0.022 – 0.025
Earth, fairly uniform section	No vegetation	0.022 – 0.025
	Grass, some weeds	0.025 – 0.030
	Dense weeds or aquatic plants in deep channels	0.030 – 0.035
	Sides clean, gravel bottom	0.025 – 0.030
	Sides clean, cobble bottom	0.030 – 0.040
Dragline excavated or dredged	No vegetation	0.028 – 0.033
	Light brush on banks	0.035 – 0.050
Rock: Based on design section		0.035
Based on actual mean section	Smooth and uniform	0.035 – 0.040
	Jagged and irregular	0.040 – 0.045
Channels not maintained, weeds and brush uncut	Dense weeds, high as flow depth	0.08 – 0.12
	Clean bottom, brush on sides	0.05 – 0.08
	Clean bottom, brush on sides, highest stage of flow	0.07 – 0.11

Table B-1: Manning's n Coefficient Table (Cont'd)

	Dense brush, high stage	0.10 – 0.14
Highway Channels and Swales with Maintained Vegetation		**Manning's n**
(Values shown are for velocities of 2 and 5 fps)		
Depth of flow up to 0.7 ft.		
Bermuda grass, Kentucky bluegrass, buffalo grass	Mowed to 2 in.	0.07 – 0.045
	Length 4 to 6 in.	0.09 – 0.05
Good Stand, any grass	Length about 12 in.	0.18 – 0.09
	Length about 24 in.	0.30 – 0.15
Fair stand, any grass	Length about 12 in.	0.14 – 0.08
	Length about 24 in.	0.25 – 0.13
Depth flow 0.7 ft. to 1.5 ft.		
Bermuda grass, Kentucky bluegrass, buffalo grass	Mowed to 2 in.	0.05 – 0.035
	Length 4 to 6 in.	0.06 – 0.04
Good Stand, any grass	Length about 12 in.	0.12 – 0.07
	Length about 24 in.	0.20 – 0.10
Fair stand, any grass	Length about 12 in.	0.10 – 0.06
	Length about 24 in.	0.17 – 0.09
Gutters		**Manning's n**
Concrete gutter	Troweled finish	0.012
Asphalt pavement	Smooth texture	0.013
	Rough texture	0.016
Concrete gutter with asphalt pavement	Smooth	0.013
	Rough	0.015
Concrete pavement	Float finish	0.014
	Broom finish	0.016
For gutters with small slope, where sediment may accumulate, increase all above values of n by		0.002

(Source: Searcy 1973.)

- (Source: U.S. Soil Conservation Service 1986.)

1. The n values are a composite of information compiled by Engman (1986).
2. Includes species such as weeping lovegrass, bluegrass, buffalo grass, blue grama grass, and native grass mixtures.
3. When selecting n, consider cover to a height of about 0.1 ft. This is the only part of the plant cover that obstructs sheet flow.

Inlet Design Coefficients

Table B-2: Coefficients for Inlet Control Design Equations

					Unsubmerged		Submerged	
Chart No.	**Shape and Material**	**Nomograph Scale**	**Inlet edge description**	**FORM**	**K**	**M**	**C**	**Y**
1	Circular Concrete	1	Square edge with headwall	1	0.0098	2	0.0398	0.67
		2	Groove end with head wall	1	0.0018	2	0.0292	0.74
		3	Groove end projecting	1	0.0045	2	0.0317	0.69
2	Circular CMP	1	Headwall	1	0.0078	2	0.0379	0.69
		2	Mitered to slope	1	0.021	1.33	0.0463	0.75
		3	Projecting	1	0.034	1.5	0.0553	0.54
3	Circular	A	Beveled ring, 45° bevels	1	0.0018	2.5	0.0300	0.74
		B	Beveled ring, 33.7° bevels	1	0.0018	2.5	0.0243	0.83
8	Rectangular Box	1	30° to 75° wingwall flares	1	0.026	1	0.0347	0.81
		2	90° to 15° wingwall flares	1	0.061	0.75	0.0400	0.8
		3	0° wingwall flares	1	0.061	0.75	0.0423	0.82
9	Rectangular Box	1	45° Wingwall flare, d=.0430	2	0.51	0.667	0.0309	0.80

Table B-2: Coefficients for Inlet Control Design Equations (Cont'd)

					Unsubmerged		Submerged	
Chart No.	**Shape and Material**	**Nomograph Scale**	**Inlet edge description**	**FORM**	**K**	**M**	**C**	**Y**
		2	18°-33.7° wingwall flare, d=.0830	2	0.486	0.667	0.0249	0.83
10	Rectangular Box	1	90° headwall with ¾ chamfers	2	0.515	0.667	0.0375	0.79
		2	90° headwall with 45° bevels	2	0.495	0.667	0.0314	0.82
		3	90° headwall with 33.7° bevels	2	0.486	0.667	0.0252	0.865
11	Rectangular Box	1	¾ chamfers; 45° skewed headwall	2	0.545	0.667	0.0505	0.73
		2	¾ chamfers; 30° skewed headwall	2	0.533	0.667	0.0425	0.705
		3	¾ chamfers; 15° skewed headwall	2	0.522	0.667	0.0402	0.68
		4	45° bevels; ¾ skewed headwall	2	0.498	0.667	0.0327	0.75
12	Rectangular Box – ¾ Chamfers	1	45° non-offset wingwall flares	2	0.497	0.667	0.0339	0.803
		2	18.4° non-offset wingwall flares	2	0.493	0.667	0.0361	0.806
		3	18.4° non-offset wingwall flares, 30° skewed barrel	2	0.495	0.667	0.0386	0.71
13	Rectangular Box – Top Bevels	1	45° wingwall flares – offset	2	0.497	0.667	0.0302	0.835
		2	33.7° wingwall flares – offset	2	0.495	0.667	0.0252	0.881
		3	18.4° wingwall flares – offset	2	0.493	0.667	0.0227	0.887
16-19	C M Boxes	2	90° headwall	1	0.0083	2	0.0379	0.69
		3	Thick wall projecting	1	0.0145	1.75	0.0419	0.64

Table B-2: Coefficients for Inlet Control Design Equations (Cont'd)

Chart No.	Shape and Material	Nomograph Scale	Inlet edge description	FORM	Unsubmerged K	Unsubmerged M	Submerged C	Submerged Y
		5	Thin wall projecting	1	0.034	1.5	0.0496	0.57
29	Horizontal Ellipse – Concrete	1	Square edge with headwall	1	0.01	2	0.0398	0.67
		2	Groove end with head wall	1	0.0018	2.5	0.0292	0.74
		3	Groove end projecting	1	0.0045	2	0.0317	0.69
30	Vertical Ellipse – Concrete	1	Square edge with headwall	1	0.01	2	0.0398	0.67
		2	Groove end with head wall	1	0.0018	2.5	0.0292	0.74
		3	Groove end projecting	1	0.0095	2	0.0317	0.69
34	Pipe Arch 18" Corner Radius CM	1	90° headwall	1	0.0083	2	0.0379	0.69
		2	Mitered to slope	1	0.03	1	0.0463	0.75
		3	Projecting	1	0.034	1.5	0.0496	0.57
35	Pipe Arch 18" Corner Radius CM	1	Projecting	1	0.0300	1.5	0.0496	0.57
		2	No Bevels	1	0.0088	2	0.0368	0.68
		3	33.7° bevels	1	0.003	2	0.0269	0.77
36	Pipe Arch 31" Corner Radius CM	1	Projecting	1	0.0300	1.5	0.0496	0.57
		2	No Bevels	1	0.0088	2	0.0368	0.68
		3	33.7° bevels	1	0.003	2	0.0269	0.77
40-42	Arch CM	1	90° headwall	1	0.0083	2	0.0379	0.69
		2	Mitered to slope	1	0.03	1	0.0463	0.75
		3	Thin wall projecting	1	0.034	1.5	0.0496	0.57

Table B-2: Coefficients for Inlet Control Design Equations (Cont'd)

Chart No.	Shape and Material	Nomograph Scale	Inlet edge description	FORM	Unsubmerged K	Unsubmerged M	Submerged C	Submerged Y
55	Circular	1	Smooth tapered inlet throat	2	0.534	0.555	0.0196	0.90
		2	Rough tapered inlet throat	2	0.519	0.64	0.0210	0.90
56	Elliptical Inlet Face	1	Tapered inlet – beveled edges	2	0.536	0.622	0.0368	0.83
		2	Tapered inlet – square edges	2	0.5035	0.719	0.0478	0.8
		3	Tapered inlet – thin edge projecting	2	0.547	0.8	0.0598	0.75
57	Rectangular	1	Tapered inlet throat	2	0.475	0.667	0.0179	0.97
58	Rectangular Concrete	1	Side tapered – less favorable edges	2	0.56	0.667	0.0466	0.85
		2	Side tapered – more favorable edges	2	0.56	0.667	0.0378	0.87
59	Rectangular Concrete	1	Slope tapered – less favorable edges	2	0.5	0.667	0.0466	0.65
		2	Slope tapered – more favorable edges	2	0.5	0.667	0.0378	0.71

Headloss Coefficients for Junctions

These are typical headloss coefficients used in the standard method for estimating headloss through manholes and junctions.

Table B-3: Typical Headloss Coefficients

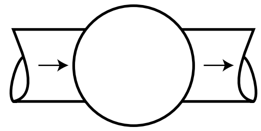
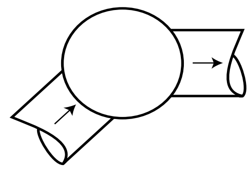
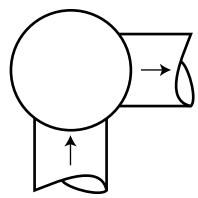
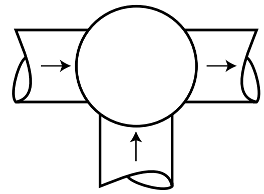
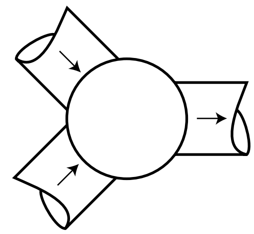
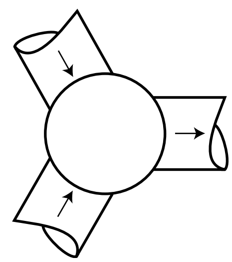
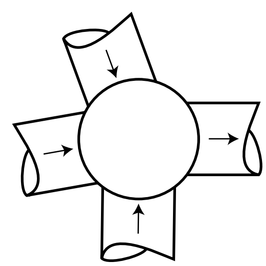
Type of Manhole	Diagram	Headloss Coefficient
Trunkline only with no bend at the junction		0.5
Trunkline only with 45° bend at the junction		0.6
Trunkline only with 90° bend at the junction		0.8

Table B-3: Typical Headloss Coefficients

Type of Manhole	Diagram	Headloss Coefficient
Trunkline with one lateral		Small 0.6 Large 0.7
Two roughly equivalent entrance lines with angle < 90° between lines		0.8
Two roughly equivalent entrance lines with angle > 90° between lines		0.9
Three or more entrance lines		1.0

Roughness Values—Manning's Equation

Commonly used roughness values for different materials are:

Table B-4: Manning's Coefficients n for Closed-Metal Conduits Flowing Partly Full

Channel Type and Description	Min.	Normal	Max.
a. Brass, smooth	0.009	0.010	0.013
b. Steel			
1. Lockbar and welded	0.010	0.012	0.014
2. Riveted and spiral	0.013	0.016	0.017
c. Cast iron			
1. Coated	0.010	0.013	0.014
2. Uncoated	0.011	0.014	0.016
d. Wrought iron			
1. Black	0.012	0.014	0.015
2. Galvanized	0.013	0.016	0.017
e. Corrugated metal			
1. Subdrain	0.017	0.019	0.021
2. Storm drain	0.021	0.024	0.030

Appendix

References C

Ming Jin, Samuel Coran and Jack Cook (2004), “New One-Dimensional Implicit Numerical Dynamic Sewer and Storm Model”, Haestad Methods Inc., Waterbury, CT

Ben C. Yen (2001), "Hydraulics of Sewer System", in Stormwater Collection Systems Design Handbook, ed. Larry W. Mays, McGraw-Hill, New York

Danny L. Fread (1993), "Flow Routing", in Handbook of Hydrology, ed David R. Maidment, McGraw-Hill, New York

Ming Jin and Danny L. Fread (2000) "Discussion on the Application of Relaxation Scheme to Wave-Propagation Simulation in Open-Cannel Networks", Journal of Hydraulic Eng., ASCE, 126(1), 89-91.

Ming Jin and Danny L. Fread (1999) "One-dimensional modeling of mud/debris unsteady flows", Journal of Hydraulic Eng., ASCE, 25(8), 827-834.

Ming Jin and Danny L. Fread (1997) "Dynamic flood routing with explicit and implicit numerical solution schemes", Journal of Hydraulic Eng., ASCE, 123(3), 166-173.

Glossary

Alternative:	A categorized data set that create scenarios when placed together. Alternatives hold the input data in the form of records. A record holds the data for a particular element in your system.
Aquifer:	A sub-surface groundwater area used to model the vertical movement of water infiltrating from the subcatchments which lie above them. Aquifers also permit the infiltration of groundwater into the conveyance system, or exfiltration of surface water from the conveyance system, depending on the hydraulic gradient that exists.
Backflow:	The backing up of water through a conduit or channel in the direction opposite to normal flow.
Basin:	An area having a common outlet for its surface runoff.
Batch Run:	A set of multiple scenarios that are computed together. This is helpful if you want to queue a large number of calculations, or manage a group of smaller calculations as a set.
bmp:	File name extension for bitmap image files, which can be used as background layers in SewerGEMS V8*i*.
Branch:	In Bentley SewerGEMS V8*i*, a series of connected elements. The Bentley SewerGEMS V8*i* calculation engine includes a heuristic routine that decomposes a network into its component branches, and each branch is solved independently using an implicit solver. Each branch comprises a series of connected elements. Elements with the same branch ID are solved together.
Calculation Options Profile:	A set of calculation options associated with a specific scenario.
Flow (Captured):	The portion of the inlet flow which actually drains into the catch basin. The captured flow is dependent on the inlet capacity of the catch basin.

Catalog Pipes:	User-defined re-usable data sets that define common physical characteristics of pipes.
Catch basin:	The geographical area that "catches" the rainfall and directs it towards a common discharge point within the storm collection network.
Catchment:	The area drained by a stream, lake or other body of water in a sewer or stormwater system.
Channel:	A channel refers to a channel that changes geometry from the upstream cross section to the downstream cross section. Channels can be used to model natural streams or swales which are not prismatic in cross section. Channels must have a cross section element at each end and properties are interpolated along the channel.
CN:	SCS Curve Number
Collection:	Bentley SewerGEMS V8*i* uses the term Collection to identify some fields where you enter a set of data instead of a single value. To use this feature, click the field in which the word **Collection** appears, then click the **Ellipsis** (…) button to open a dialog box into which you enter data.
Composite Hydrograph:	A graph of the total flow over time from multiple defined fixed/unit loads, hydrographs, and pattern loads.
Conduit:	An open- or closed-section element through which water moves. A conduit has a constant roughness and cross section shape along its entire length. In Bentley SewerGEMS V8*i*, a conduit can refer to any prismatic channel or pipe that conveys flow. The cross section of a conduit must remain constant from one end to the next.
Cross Section Node:	An element which is perpendicular to flow across a channel.
DD:	Excess duration rainfall.
Diurnal Curve:	A pattern that relates to the changes in loads over the course of the day, reflecting times when people are using more or less water than average.
Drag:	Dragging is an action you perform with the mouse to select items in the drawing pane. Click in the drawing pane, hold down the mouse button, and move the mouse to form a rectangle around the elements you want to select. After you have formed the rectangle around the items you want to select, let go of the mouse button.

dwh:	File name extension for the binary format file used by the SewerGEMS V8*i* Stand-Alone mode only. The .dwh file contains the drawing.
dxf:	File name extension for Data Exchange File format image files, which can be used as background layers in SewerGEMS V8*i*. Dxf files store vector data for drawings typically produced in CAD programs.
Dynamic Manager:	A dialog box that can be displayed floating above the workspace or attached (docked) to any one of the sides of the workspace.
Dynamic selection set:	A selection created by running a query.
English:	E-Q-TWU.S. customary units, such as inch or acre
Element Symbology:	Refers to the way in which elements and their associated labels are displayed in Bentley SewerGEMS V8*i* You use the Element Symbology manager to manage element annotations and color coding.
Element Table:	A read-only, predefined FlexTable that you can use to review data about a specific element type. There is one predefined table for every element available in Bentley SewerGEMS V8*i*.
E-Q-TW:	Elevation-flow-tailwater rating curves
Extended Period Simulation:	A calculation type where the model is analyzed over a specified duration of time.
FlexTable:	A customizable table that lets you view input data and results for all elements of a specific type in a tabular format. You can use the standard set of FlexTables or create your own FlexTables to compare data and create reports.
Freeboard:	The difference between the top of dam elevation and the water surface elevation.
gif:	File name extension for Graphic Image Format image files, which can be used as background layers in SewerGEMS V8*i*.
GIS:	Geographic information system
Global storm event:	In Bentley SewerGEMS V8*i*, project-wide storm events, usually applied to catchments that do not use local rainfall. Project-wide global storm events are typically associated with Hydrology alternatives.

Gutter:	An open-section element that models overflow. Gutters are used in Bentley SewerGEMS V8*i* only to model the water which exceeds the capacity of in catch basin inlet and must flow through a surface gutter to the next catch basin. A Bentley SewerGEMS V8*i* gutter can only receive water from a catch basin.
Hydrograph:	A graph of discharge versus time.
Hyetograph:	A graphical representation of rainfall intensity with respect to time.
Hyperlink:	In Bentley SewerGEMS V8*i*, a link to an external file, such as an image or movie file, for the purpose of associating the file with a specific element.
I_a:	Initial abstraction
ICPM:	Interconnected Pond Modeling
IDF/I-D-F:	Intensity-Duration-Frequency
Implicit Engine:	One of two calculation engines available in Bentley SewerGEMS V8*i*. The implicit engine uses a four-point implicit finite difference solver which tends to be more stable than an explicit solver. The implicit engine in Bentley SewerGEMS V8*i* is based on the solver in the National Weather Service FLDWAV model.
Infiltration:	Water that enters the system from the ground through defective pipes, pipe joints, connections, or manhole walls
Inflow:	Inflow is specified at a manhole or wet well as the total amount of wet weather inflow. Together with the infiltration along the pipes, the inflow forms the wet weather part of the sewer load.
Inflow Collection:	A collection of any combination of fixed, hydrograph, or pattern inflows for an element.
Inlet flow:	Flows added to the manholes, catch basins, ponds, and wet wells in your model, including runoffs from catchments, user input hydrographs, and flows from other sources, such as pumps.
inp:	File name extension for SWMM v5 project files.
jpg:	File name extension for image files produced using the Joint Photographic Experts Group (JPEG) standard. JPEG files, which can also have the file name extension jpeg, can be used as background layers in SewerGEMS V8*i*.

Junction chamber:	A structure in which flow can be mixed or split. In combined sewers systems, flow can be split between that going to treatment and that going to an overflow.
K_e:	Entrance loss coefficient
K_r:	Reverse flow loss coefficient
Land Uses:	Categories of activities or land surface characteristics that are assigned to catchment elements. Examples of land use activities are residential, commercial, industrial, and undeveloped.
Manhole:	Element that provides access to the system for inspection and maintenance. Manholes are usually installed where there is a change in horizontal (plan-view) pipe direction or pipe slope, where several pipes join, or where the pipe size changes.
mdb:	File name extension for SewerGEMS V8*i* database files. The .mdb file contains all modeling data, and includes everything needed to perform a calculation. This file is an open access database format file and can be viewed and edited.
MrSID:	File name extension for Multi-resolution Seamless Image Database format raster image files, which can be used as background layers in SewerGEMS V8*i*.
Orifice Coefficient:	For stormwater openings, an orifice coefficient of 0.60 is often used.
Outfall:	The ultimate termination points in a network.
Overflow:	The flow lost when the water surface elevation at a manhole or other type of node is above the rim elevation or user-specified overtopping elevation.
Pattern:	A series of multipliers which describe how the base load varies over time.
Pattern Load:	A single average base load and a series of dimensionless multipliers used to delineate how the load varies over time.
Pattern Setup:	In SewerGEMS V8*i*, a pattern setup allows you to match unit sanitary (dry weather) loads with appropriate loading patterns. Each scenario can use a different pattern setup, thus allowing you to model different loading alternatives for different extended period simulations.
png:	File name extension for Portable Network Graphic format image files, which can be used as background layers in SewerGEMS V8*i*.

Pollutograph:	A plot of time vs. concentration or mass rate.
Pond outlet structure:	A structure that allows flow to discharge from a pond.
Pressure junction:	A connection between two or more pressure pipes of varying characteristics. Loads may enter a pressure portion of a network through a pressure junction.
Pressure pipe:	In SewerGEMS V8*i*, a type of link element used to connect node elements in pressure portions of a network.
Profile:	A graph that plots a particular attribute across a distance, such as ground elevation along a section of piping. As well as these side or sectional views of the ground elevation, profiles can be used to show other characteristics, such as hydraulic grade, pressure, and constituent concentration.
Pump:	A structure in a wastewater system designed to add the energy (head) necessary to overcome elevation differences and head losses.
Q:	Flow
Query:	A SQL expression you build in Bentley SewerGEMS V8*i* to filter a FlexTable or create a selection set.
Sanitary Flow Collection:	A collection of loads that contains any combination of hydrograph, unit, or pattern loads.
Sanitary Load:	A load that results from human activity and are not weather-dependent.
Scenario:	A set of input data (in the form of alternatives), calculation options, results, and notes associated with a set of calculations. Scenarios let you set up an unlimited number of "What If?" situations for your model, and then modify, compute, and review your system under those conditions.
Selection Sets:	User-defined groups of network elements that let you predefine a group of elements to manipulate together.
shp:	File name extension for shapefiles, which you can import as background layers for your model.
SI:	International System of units or metric units.
Static selection set:	A selection set created by selecting a group of elements in your model.
Storm event:	In Bentley SewerGEMS V8*i*, a single rainfall curve that represents one rainfall event for a given recurrence interval.

swg:	File name extension for SewerGEMS V8*i* project files. The .swg file contains data such as annotation, color-coding, and project-level options. This file is an open .xml format file and can be viewed and edited.
SWMM Engine:	One of two calculation engines available in Bentley SewerGEMS V8*i*. The SWMM engine uses the solver from the EPA Stormwater Management Model version 5. This is an explicit solver which, while more prone to stability problems, exactly matches the results from SWMM 5.
T_c:	The time of concentration (T_c) is the time it takes for runoff to travel from the hydraulically most distant part of the watershed subarea to its outfall point. It is computed by summing the times it takes the water to travel through the different components of the subarea drainage system.
tiff:	File name extension for Tagged Image File Format raster image files, which can be used as background layers in SewerGEMS V8*i*. The extension .tif is also used for these types of files.
Unit Hydrograph:	A time versus flow curve that represents a one-inch volume of rainfall for a given excess duration of rainfall, DD, for a set watershed area.
Unit Sanitary Load:	Loading unit count representing the local count of loading units for a specified unit dry weather load.
User data extensions:	A set of one or more attribute fields that you can define to hold data to be stored in the model. User data extensions allow you to add your own data fields to your project.
User Notifications:	Messages that appear after you compute your model. These messages can help you troubleshoot errors in your model.
Virtual Conduit:	Virtual conduits are a special compatibility element included in Bentley SewerGEMS V8*i* to help modelers achieve fidelity between the Bentley SewerGEMS V8*i* model and other storm modeling solutions that model pumps and control structures such as weirs, orifices, and rating tables as network links. Virtual conduits are displayed in your model as dashed lines.

Watershed:	A watershed is a collection of catchments which outfall to the same point. In other words, for each storm in the selected design storm collection, a hydrograph is generated for each catchment. The sum of all those hydrographs is the watershed hydrograph.
Wet Weather Load:	A load that is not related to rainfall activity, such as groundwater infiltration (water leaking into a pipe through cracks, joints, and other defects) and structure inflow (surface water entering a structure through the cover).
Wet well:	A boundary condition between the pressure and gravity portions of a network. Wet wells serve as a collection point for gravity systems, and as an HGL boundary node for the pressure system. Dry loads can also enter the sewer network at these locations.

Symbols

A

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z