www.haestad.com METHODS

USER'S GUIDE

WaterCAD v4

for Windows

WaterCAD v4 User's Guide (First Printing)

37 Brookside Rd.

Waterbury, CT 06708-1499

Voice: (203) 755-1666

FAX: (203) 597-1488

e-mail: info@haestad.com

Internet: http://www.haestad.com

Table of Contents

Preface

Thank you for purchasing WaterCAD. At Haestad Methods 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.

WaterCAD is a feature-rich program with extensive on-line documentation that is able to provide a level of instruction appropriate to your needs. Do not be fooled by the existence of this user's guide. You do not need to read anything to get started!

When you first use the program, WaterCAD's intuitive interface and interactive dialogs will guide you. If you need more information, go to our comprehensive, context-sensitive, on-line help by either pressing the **Help** button present in each dialog box, pressing the **F1** key, or right-clicking anywhere in a dialog. Help text regarding the area of the program in which you are working will be displayed.

We are betting that you will be able to use our product right out of the package. If you know how to run Setup within Windows, then go ahead and get right to work. Install WaterCAD, and enjoy!

Notes

Chapter 1

Orientation

1.1 What Is WaterCAD?

WaterCAD is a powerful yet easy to use program that helps engineers design and analyze complex pressurized piping systems. WaterCAD's powerful graphical interface (both in Stand-Alone and AutoCAD mode) makes it easy to quickly lay out a complex network of pipes, tanks, pumps, and more. You can use WaterCAD to:

- Perform steady-state analyses of water distribution systems with pumps, tanks, and control valves.
- Perform extended period simulations to analyze the piping system's response to varying supply and demand schedules.
- Perform water quality simulations to determine the water source and age, or track the growth or decay of a chemical constituent throughout the network.
- Perform Fire Flow Analyses on your system to determine how your system will behave under extreme conditions.
- Use the powerful Scenario Management features to mix and match a variety of "*What If?*" alternatives on your system. Create multiple sets of hydraulic, physical property, operational, initial setting, fire flow, cost, and water quality alternatives. Create and run any number of scenarios by mixing and matching alternatives, then view and compare the results quickly and easily with WaterCAD's flexible scenario management feature.

The best introduction to WaterCAD is the One-Minute Tutorial. Explore WaterCAD freely and remember that there is a lot of valuable information in on-line help.

1.2 What is New in WaterCAD v4?

WaterCAD v4 includes a variety of new and enhanced features, including:

- **AutoCAD 2000 Support –** WaterCAD in AutoCAD mode (which is the upgrade for Cybernet v3) can now run inside both AutoCAD 2000 and AutoCAD R14.
- **Microsoft Office 2000 Support** – Now supports Microsoft Excel 2000, and Access 2000.

- **Windows 2000 Support –** Now works with Windows 95, 98, NT 4.0, and Windows 2000.
- **Network Licensing Support –** Now gives you the option to purchase a license for multiple concurrent users, managed by a network based administrator software.
- **User Data Extension** – User-defined attributes can now be added to supplement WaterCAD with your own custom features. This data can then be used like any other WaterCAD data in FlexTables, annotating, color coding, Database Connections, etc.
- **Polyline to Pipe Conversion** – Convert existing AutoCAD drawings directly to WaterCAD models.
- **Element Relabeling** – Edit selected element labels to automatically renumber, replace or append a prefix/suffix.
- **Aerial View –** Optional separate window used to facilitate zooming, panning, and locating a small viewing area in the main window.
- **Aligning Pipe Labels –** Automatically aligns pipe labels and annotations with the associated pipe.
- **Cost Estimating –** Calculate a planning level estimate of the capital cost associated with an entire system or any portion of a system. This makes it easy to compare the costs associated with the various scenarios, thus helping to ensure that the most cost-effective design is chosen.
- **Contouring by Selection Set –** Generate contours for any subset of elements.
- **Drawing Review –** Quickly review your network data for potential problems resulting from data entry errors or data discrepancies in the source (database, Shapefile, or CAD drawing) from which a model was imported.

1.3 Installation, Upgrades, and Updates

1.3.1 Minimum System Requirements

Below are the minimum and recommended system requirements for running WaterCAD without significant delays. Note that some of the requirements for AutoCAD Mode, such as RAM, are fairly high, due to the demands of AutoCAD and operating system demands, not WaterCAD itself.

Stand-Alone Mode:

Processor:	Pentium-166
RAM:	32 Megabytes
Hard Disk:	25 Megabytes of free storage space, with additional room for data files
Operating System:	Windows 95 or later, or Windows NT 4.0
Display:	800 x 600 resolution, 256 colors

AutoCAD Mode:

Processor:	Pentium-166
RAM:	64 Megabytes
Hard Disk:	25 Megabytes of free storage space, with additional room for data files
Display:	800 x 600 resolution, 256 colors

Recommended:

While Haestad Methods' software will perform adequately given the minimum system requirements, performance will only improve with a faster system. Our products are designed to perform at optimal levels with a fast CPU and ample amounts of RAM and free disk space. We highly recommend running our software on the best system possible to maximize its potential, especially for larger models containing thousands of pipes. We understand that an engineer's time is a valuable commodity, and we have designed our software to help make the most of that time.

1.3.2 Installing Haestad Methods' Products

For Windows 95, Windows 98, Windows NT 4.0, and Windows 2000, follow these easy steps, for installing a single user license copy of the program:

1. If you have not done so, turn on your computer.
2. Place the diskette labeled Disk 1 in the floppy disk drive (commonly the a: or b: drive).
3. Place the CD in your CD-ROM drive (commonly the d: or e: drive).

4a. If the Autorun feature of the operating system is enabled, setup will begin automatically. Proceed to step six.

4b. If Autorun is disabled, click the **Start** button on the task bar, select **Run** from the menu, and type d:\setup (use the actual drive letter of the CD-ROM drive if it is not the d: drive), and then click **OK**.

5. Follow the instructions of the Setup Wizard.

If you own a network license version of the software, please refer to the accompanying Network License Document. If you still have questions, consult our online KnowledgeBase on our web site www.haestad.com or contact Haestad Methods technical support.

1.3.3 Uninstalling Haestad Methods' Products

Haestad Methods' products come with an uninstall option. After a single user license copy of a Haestad Methods' product is installed onto a computer, it must be uninstalled before a new installation can occur.

To uninstall the program:

Click **Start\Programs\Haestad Methods*Product_Name*\Uninstall *Product_Name***. The original floppy disk labeled **Disk 1** that came with the product must be in the floppy drive at the time of uninstalling.

If you own a network license version of the software, please refer to the accompanying Network License Document. If you still have questions, consult our online KnowledgeBase on our web site www.haestad.com or contact Haestad Methods technical support.

1.3.4 Troubleshooting Setup or Uninstall

Because of the multi-tasking capabilities of Windows 95, 98, 2000 and NT, you may have applications running in the background that make it difficult for the setup routines to determine the configuration of your current system. If you have difficulties during the install (setup) or uninstall process, please try these steps before contacting our technical support staff:

1. Restart your machine.
2. Verify that there are no other programs running. You can see applications currently in use by pressing **Ctrl-Alt-Del** in Windows 95, Windows 98 or 2000, or **Ctrl-Shift-Esc** in Windows NT. Exit any applications that are running.
3. Run setup or uninstall again without running any other program first.

If these steps fail to successfully install or uninstall the product, contact our support staff immediately.

1.3.5 Software Registration

During the installation of the program, a dialog will appear asking you to register the software. Please note the label with your registration information is on the inside back cover of the manual.

Although this software is not copy protected, registration is required to unlock the software capabilities for the hydraulic network size that you have licensed. All registration information must be entered into the **Registration** dialog exactly as it appears on the label.

- Company
- City
- State/Country
- Registration Number

After you have registered the software, you can check your current registration status by opening the registration dialog in the software itself.

To open the **Registration** dialog:

1. Select **Help\About *Product_Name* …** from the pull-down menu.
2. Click the **Registration** button in the **About *Product_Name*** dialog.

The current registration status (number of licenses, expiration date, feature level, etc) will be displayed.

Use the **Print** button to print a copy of the information shown in the **Registration Form** dialog.

Use the **Copy** button to place the registration information in the Windows Clipboard so that you can paste it into another Windows application.

If you own a network license version of the software, please refer to the accompanying Network License Document. If you still have questions, consult our online KnowledgeBase on our web site www.haestad.com or contact Haestad Methods technical support.

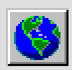
1.3.6 Upgrades

When you click the **Registration** button on the **Help\About *Product Name* …** dialog, the current registration status (number of licenses, expiration date, feature level, etc) is displayed. To upgrade to more pipes or inlets, higher feature levels, or additional licenses, call Haestad Methods (see Contacting Haestad Methods) to obtain a new registration number. We will have you up and running in no time!

1.3.7 Globe Button

Haestad Methods makes it easy to stay up-to-date with the latest advances in our software. Software maintenance releases can be downloaded from the Haestad Methods web site quickly and easily if you are a subscriber to our ClientCare Program. Just click the **Globe** icon on the tool palette to launch your preferred web browser and open the Haestad Methods' Program Update web site. The web site will automatically check to see if your installed version is the latest available, and if not, it will provide you with the opportunity to download the correct upgrade to bring it up-to-date.

The ClientCare Program also gives you access to our extensive KnowledgeBase for answers to all Frequently Asked Questions (FAQ). Contact our sales team for more information on our ClientCare Program.

Use the Globe button  **to keep your investment current.**

1.4 Learning WaterCAD

1.4.1 WaterCAD Documentation

WaterCAD's on-line documentation delivers extensive detail. Simply click the **Help** button, press the **F1** key, or right-click anywhere in the program to access context-sensitive help.

The WaterCAD User's Guide is provided to you as a means to read and learn about WaterCAD while you are away from your computer. The topics you find in the User's Guide will also be found in the on-line help.

The on-line help is typically more complete and up-to-date than the manual, as it is refined with each new software update.

WaterCAD also contains on-line tutorials, lessons, and sample files to help you become familiar with the software's features and capabilities. The tutorials can be accessed by clicking **Help\Tutorials** from the pull-down menu. The lessons can be found in the printed documentation as well as in the on-line help. The sample files are located in your **Haestad\Wtrc\Sample** directory.

1.4.2 How to Use Help

All of our products feature extensive context-sensitive help. There are several ways to obtain help on topics:

- To get help for the window in which you are working, press the **F1** key or click the **Help** button.
- To get help for a specific item, right-click the desired object and select **Help** from the pull-down menu.

To navigate within Help:

- When you click text that is underlined, Help "jumps" to a related topic or definition. If the text is dashed underlined, a pop-up window will appear.
- To return to the previous topic, click the **Back** button at the top of the **Help** window.
- To print a Help topic, click the **Print** button at the top of the **Help** window.
- To make Help stay on top of other windows, open the main **Help** window and select **Options\Keep Help On Top\On Top** from the pull-down menu.

An Example:

Suppose that a primary aspect of your project is to exchange data with ESRI (Environmental Systems Research Institute, Inc.) Shapefiles. To obtain a detailed description on how to import your shapefiles:

1. Click **Help\Contents** to open the main **Help** window.
2. On the left side of the **Help** window, select the **Index** tab. In the box where the cursor is flashing, enter the word **Shapefile**
3. This will automatically scroll through the topics and display a list of topics concerning shapefiles. For instance, you can select **Shapefile\Import**. A sublist will pop up where you can select **Shapefile Import Example**. A detailed Help topic will appear on the right.

1.4.3 How Do I?

"**How Do I**" is an easily referenced topic in WaterCAD's on-line documentation. It is a listing of commonly asked questions about WaterCAD. Follow these steps to find your way to How Do I**:**

1. Click **Help\How Do I** from the pull-down menu.
2. The listing of "**How Do I**" topics will appear. Click on the topic of your choice for a detailed explanation.
3. To return to the listing of "**How Do I**" questions, click the **Back** button.

1.4.4 Help Command

Access the **Help** menu. Using the **Help** command is the same as selecting **Help** from the pull-down menu.

1.4.5 Glossary

The glossary contains many terms used throughout the dialogs and the on-line Help.

To use the Glossary:

1. Click **Help\Contents...** to open the main **Help** window.
2. Click the **Contents** tab, then double-click the **Glossary** book.

3. Click the **Glossary** page and the Glossary topic will appear.

4. Click the first letter of the word for which you are looking for more information.

5. Click the term and a pop-up box will appear with a definition of the selected word.

1.4.6 Tutorials

The tutorials, along with the lessons located in Chapter 3, quickly introduce you to specific features of the program. To access the tutorials click **Help\Tutorials** from the pull-down menus. Run a tutorial by selecting one of the entries in the list and clicking the **OK** button. End a tutorial at any time by either pressing the **Esc** key (in Stand-Alone mode), or by clicking the cross in the upper right-hand corner of any tutorial dialog. If you need further information, access our on-line help by pressing the **F1** key.

1.4.7 Sample Projects

1. Select the **File\Open** command from the pull-down menu to access the **Open Project File** dialog.

2. Choose EXAMPLE?.WCD (or EXAMPLE?.DWG, if using WaterCAD in AutoCAD mode) from the Sample directory, and click **Open**.

These are working network models, so you can explore the systems and see how different elements are modeled. First, calculate the system by using the **GO** button on the main toolbar. Then use Quick View, Graphs, Profiles, Tabular views, Detailed Reports, Color Coding, and Contouring to see how the system behaves. To get the best introduction to a new feature, try running some of the tutorials.

1.4.8 Haestad Methods' Workshops

Haestad Methods offers a variety of workshops dealing with topics ranging from urban stormwater management to water distribution modeling, alternating theory, modeling insights and hands-on practice with software instruction. These workshops are held at various locations, and include discounted pricing when purchasing Haestad Methods software.

For more information on our workshops (such as instructors, schedules, pricing, and locations), please contact our sales department, or visit our web site at www.haestad.com for current workshop schedules and locations. We will be glad to answer any questions you may have regarding the workshops and our other products and services.

Haestad Methods offers a range of other training services including on-site training and on-line training. For detailed information on the availability of these options, visit www.haestad.com/education.

1.5 Contacting Haestad Methods

1.5.1 Sales

Haestad Methods' professional staff is ready to answer your questions Monday through Friday during Haestad Methods' normal operating hours.

Please contact your sales representative for any questions regarding Haestad Methods' latest products and prices.

Fax: +1-203-597-1488

Phone: +1-203-755-1666

Email: Sales@haestad.com

1.5.2 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, and the Internet. For information on the various levels of support that we offer, contact our sales team today and request information on our ClientCare Program.

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:

- Operating system your computer is running (Windows 95, Windows 98, Windows NT, etc.)
- Name and version number of the Haestad Methods software you are calling about
- Version of AutoCAD you are running (if applicable)
- Any error messages or other information that was generated
- A note of exactly what you were doing when you encountered the problem

When e-mailing or faxing for support, please provide the following additional details to enable us to provide a timely and accurate response:

- Company name, address, and phone number
- A detailed explanation of your concerns

Hours:

Monday - Friday: 9:00 AM to 8:00 PM EST

Saturday and Sunday: 9:00 AM to 5:00 PM EST

You can contact our technical support team at:

Phone: +1-203-755-1666

Fax: +1-203-597-1488

e-mail: Support@haestad.com

1.5.3 Your Suggestions Count

At Haestad Methods, we strive to continually provide you with sophisticated software and documentation. We are very interested in hearing your suggestions for improving our products, our on-line Help system, and our printed manuals. Your feedback will guide us in developing products that will make you more productive.

Please let us hear from you!

1.5.4 How to Contact Us

Internet: http://www.haestad.com

Email: Support@haestad.com

Info@haestad.com

Fax: +1-203-597-1488

Phone: +1-203-755-1666

Mail: Haestad Methods

37 Brookside Road

Waterbury, CT 06708-1499

USA

Notes

Chapter 2

WaterCAD Main Window

If you are already familiar with standard Windows interfaces, you will find WaterCAD to be intuitive and comfortable. Even if you are not accustomed to Windows standards, just a few minutes of exploring WaterCAD should be enough to acquaint you with the flexibility and power that this program offers.

In this chapter, we will examine the program's main window, menus, and toolbars. After reading this chapter, you should be able to interact with this software in a quick and efficient manner. Additional tools for layout, annotating, and editing are described in the chapter "Layout and Editing Tools".

2.1 Main Window Components

2.1.1 Stand-Alone Mode, AutoCAD Mode

Both the Stand-Alone graphical editor and the AutoCAD interface perform actions through the WaterCAD model server.

This use of a common central model enables both modes to perform the same functions with the same behaviors. For example, graphical layout and model management are virtually identical between the two modes.

Because of the common WaterCAD model server, model data is easily shared between AutoCAD and Stand-Alone modes.

One advantage of Stand-Alone mode is that your interaction is more streamlined and dynamic by virtue of the fact that the editing environment is a dedicated network editor. Also, since AutoCAD is not needed to run in Stand-Alone mode, less system resources and memory are required.

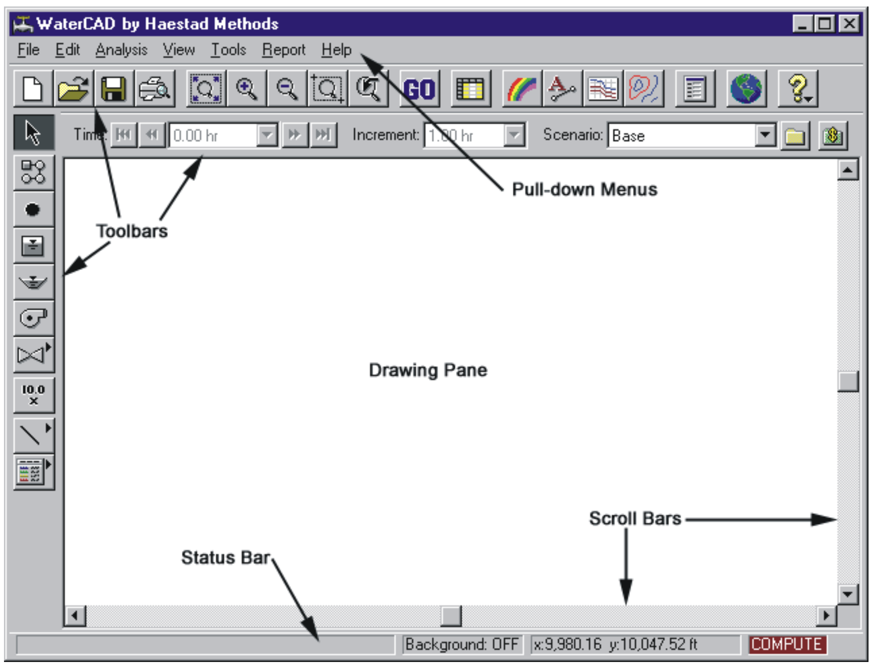
A significant advantage of the AutoCAD mode is that you can create and model your network directly within your primary drafting environment. This gives you access to all of AutoCAD's powerful drafting and presentation tools, while still enabling you to perform WaterCAD modeling tasks like editing, solving, and data management. This relationship between WaterCAD 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.

AutoCAD mode is an available feature level. Contact us to upgrade your WaterCAD Stand-Alone version to include the AutoCAD integration feature level.

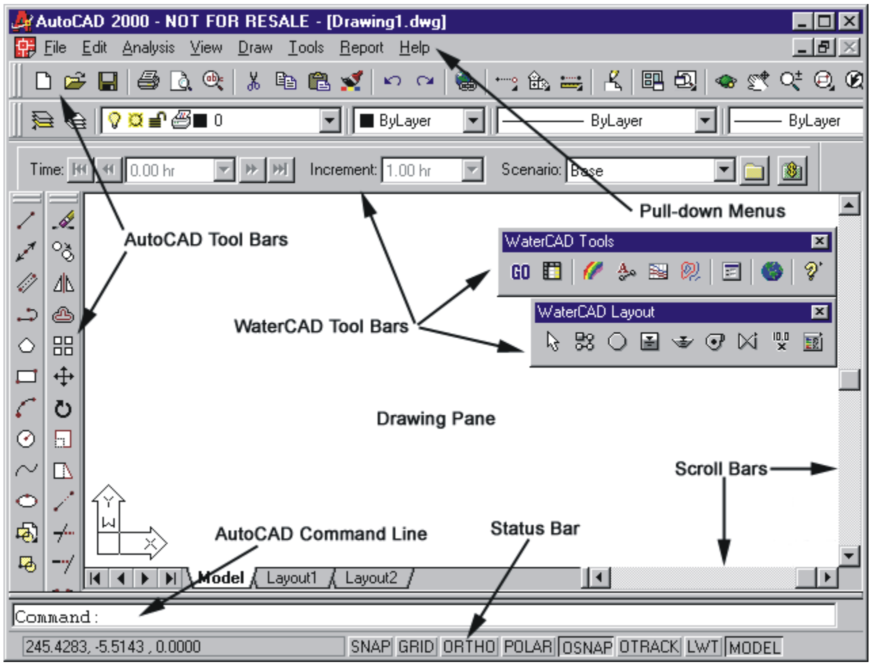
2.1.2 WaterCAD Main Windows

Both the WaterCAD Stand-Alone interface and the AutoCAD interface have many components common to Windows-based programs. The following figures illustrate some of the important areas that make up the WaterCAD Stand-Alone and AutoCAD 2000 interfaces (the WaterCAD main window looks fairly similar in AutoCAD R14 and AutoCAD 2000):

WaterCAD Stand-Alone Interface



WaterCAD AutoCAD 2000 Interface



Notice that many of the window components (such as the menus and toolbars) are very similar for the Stand-Alone editor and AutoCAD. Other features (such as the command line) are only available in AutoCAD.

For more information regarding the various functions and behaviors of AutoCAD, please refer to AutoDesk's documentation.

2.1.3 Drawing Pane

The drawing pane, the center of WaterCAD's graphical activity is where the water network elements are displayed. It is the main interactive area for creating elements, editing data, and even displaying results.

In Stand-Alone mode, the drawing pane can also display a background DXF image. This background can be helpful for aligning and positioning elements, as well as adding additional drafting elements for printing plan views.

In AutoCAD, the drawing pane is where all graphical elements, not just WaterCAD entities, are displayed and manipulated. Lines, arcs, text, and many other drafting elements can be created and modified within the drawing pane.

2.1.4 Status Bar

The status bar, located at the bottom of the window, provides relevant messages about the current state of the model and information about the drawing pane. This information includes things such as cursor location, calculation status, and drawing mode.

2.1.5 Menus, Toolbars, and Shortcut Keys

Anyone who has ever watched someone else use a computer should realize that not all people use computers in the same way. Some prefer to primarily use the mouse, some the keyboard, and others use a mixture of both.

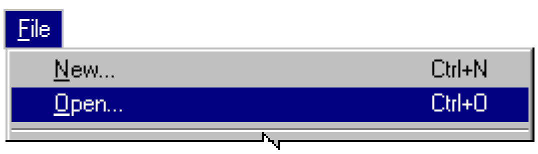
For this reason, Haestad Methods' programs provide access to the most common features through several means, including:

- Pull-Down Menus
- Toolbars
- Shortcut Keys
- Command Line (AutoCAD Only)

Pull-down Menus

As with any Windows-based program, the menu system provides easy access to many features. Items can be accessed by clicking the desired menu text, or by pressing the **Alt** key to activate the menus and then pressing the key for the underlined letter of the menu item you wish to access.

For example, to open an existing file you can use the mouse to select **File\Open** or you can press the **Alt** and **F** keys (**Alt** + **F**), then **O** on the keyboard.



Toolbars

The toolbar buttons offer one-click access to some of the most commonly used features, giving you even quicker passage to the most frequent actions.

For example, to open an existing file (the equivalent of selecting **File\Open** from the pull-down menus), simply click the **File Open Button** .

Shortcut Keys

Shortcut keys are the keyboard equivalent of toolbars. Key combinations – usually a simultaneous application of the **Ctrl** (Control) key and a letter key – can provide instant access to common features. If a shortcut is available for a menu item, it will be indicated in the menu itself.

For example, to open an existing file (the equivalent of selecting **File\Open** from the menus) you can simply press the **Ctrl** and **O** keys (**Ctrl + O**) at the same time.

2.1.6 Command Line

The command line is a special area that is not available in Stand-Alone mode. This area enables you to type commands directly, rather than using the menus, toolbars, or shortcut keys.

For example, to open an existing AutoCAD file (the equivalent of selecting **File\Open** from the pull-down menus) you can simply type the command **OPEN** at the AutoCAD command line.

Many of AutoCAD's commands are easy to enter at the command line, including accessing drafting tools (like **LINE** and **CIRCLE**) and editing tools (like **MOVE** and **ERASE**). Note that modeling elements can also be manipulated through the AutoCAD command line, just as they can be manipulated via the menus or toolbars.

For more information on the AutoCAD command line, please see the AutoCAD documentation.

2.2 WaterCAD Menus

Although the toolbars and shortcut keys provide quick and easy access to commonly used features, the pull-down menu system provides much more comprehensive access to WaterCAD's properties and behaviors. Since toolbar buttons and shortcut keys do not exist for all of these features, the menus are a logical choice for exploring all areas of WaterCAD. This section will introduce you to many of the things you can do with the menus in WaterCAD and show you how you can access these features, including the toolbar buttons and shortcut keys that are available.

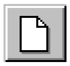
Commands are grouped under several menus, which are nearly identical between Haestad Methods products. This makes any Haestad Methods product look very familiar once you already know one. The menu system for WaterCAD consists of the following selections:

- File
- Edit
- Analysis
- View
- Tools
- Report
- Help

2.2.1 File Menu

The **File** menu contains many of the items dealing with project management. It provides features to create, read, write, and print project files, as well as features for sharing data with databases and GIS systems.

- **New** – Create a new project. When you choose this item, a dialog will appear so that you can enter a drive, directory, and filename for your new project file. The Project Setup Wizard will then help you set up your new project.

  Toolbar Button: 

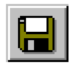
  Shortcut Key: **Ctrl + N**

- **Open** – Load an existing project file from disk. When you select this item, a dialog will appear so that you can choose the name and location of the project you want to open.

  Toolbar Button: 

  Shortcut Key: **Ctrl + O**

- **Save** – Save the current project file to disk. While saving the project file, the status pane will briefly display a message to show you the progress of the save command.

  Toolbar Button: 

  Shortcut Key: **Ctrl + S**

- **Save As** – Save the current project to disk under a different filename. When you use this command, a dialog will appear prompting you to enter the drive, directory, and a new file name for your project.
- **Project Summary** – Access the Project Summary information, such as the project title and the project engineer.
- **Import\Database** – Import data from a Microsoft Access database (*.mdb) using one of the standard database import connections.
- **Import\Shapefile –** Build network elements from ESRI shapefiles. This command will start the Shapefile Wizard, which will help you bring the GIS elements and their associated data into your project.
- **Import\Polyline to Pipe** – (Stand-Alone mode only) Build network elements from a DXF file. This command will start the Polyline to Pipe Wizard, which will help you convert polyline data representing geographical data into your project as pipes and nodes.

A similar command called 'Change Entities to Pipes' is available in the AutoCAD version under the Edit menu.

- **Import\Network –** Import data from KYpipe v1, v2 or v3, EPANet v1 and v2, or Cybernet v1 and v2 file.

WaterCAD v3 projects should simply be opened in WaterCAD v4 (as any WaterCAD v4 project), without using the Import command. However, once you save the project in WaterCAD v4, the project files cannot be opened in WaterCAD v3 any longer.

- **Import\Spot Elevations** – Bring spot elevation data from a space or comma delimited ASCII file in a variety of formats.
- **Import\DXF Background –** (Stand-Alone mode only) Bring a DXF drawing file into your project as a background map. This command will open a dialog that prompts you to select the name and location of the desired DXF file.
- **Import\WaterCAD** – (AutoCAD mode only) Import a Stand-Alone WaterCAD file (*.wcd) into WaterCAD in AutoCAD mode.
- **Export\Database –** Export data to a new Microsoft Access database (*.mdb) using one of the standard database connections.
- **Export\Shapefile** – Export your project to shapefile format in GIS applications. This command will start the Shapefile Wizard, which will help you create shapefiles with the desired project elements and associated data.
- **Export\Spot Elevations** – Export spot elevation data to a space or comma delimited ASCII file in a variety of formats.

- **Export\DXF File –** Export the entire network drawing to a DXF format, which can be read by all popular CAD programs. This command will open a dialog prompting you to enter the name and location for the DXF file you would like to create.

- **Synchronize\Database Connections** – Access the Database Connection Manager, which allows you to share WaterCAD data with external databases, spreadsheets, and other ODBC compliant files. Details of this comprehensive feature are explained in the section entitled "GIS and Database Connections".

- **Synchronize\Shapefile Connections** – Access the Shapefile Connection Manager, which allows you to share WaterCAD data with external GIS systems. Details of this feature are explained in the section entitled "GIS and Database Connections".

- **Print –** Print the current view of the project drawing to a printer. Profiles and tabular reports are printed from their respective windows. The print command invokes the standard Print dialog, which allows you to select the printer and set up properties to be used.

  Shortcut Key: **Ctrl + P**

- **Print Preview –** Open the **Print Preview** dialog for the current view of the project drawing. This feature allows you to see the drawing as it will be printed before sending it to the printer.

  Toolbar Button: 

- **Printer Setup** – Select the default printer for WaterCAD to use. You can also use this command to change options related to the printer driver, such as resolution, portrait or landscape orientation, and other printer details.

- **1, 2, etc. –** The most recently opened project files appear at the bottom of the **File** menu. Using this file list, you can quickly select and open a recently used file without locating its drive and directory.

- **Exit –** Close the current project and exit WaterCAD. If you made any changes to the current project, you will be asked if you want to save the project before you exit WaterCAD.

  Shortcut Key: **Alt + F4**

2.2.2 Edit Menu

The **Edit** menu provides access to basic commands for controlling WaterCAD elements, including element navigation, selection, deletion, and undo and redo.

In Stand-Alone mode:

- **Undo [Last Action Performed]** – Reverse the last reversible action performed. Reversible actions include things such as element creation, deletion, editing, and moves. The Undo command cannot reverse the effects of some model actions, such as calculation, database synchronization, scenario creation, and tabular edits. Additionally, to ensure that the model is maintained in a consistent state, the undo/redo history will be flushed whenever an irreversible menu or button command is issued.

  Shortcut Key: **Ctrl + Z**

- **Redo [Last Action Undone]** – Reverse the effects of the last undone action. Any action that can be undone can be redone.

  Shortcut Key: **Ctrl + Y**

- **Delete –** Erase selected elements. Deleting an element removes it from all aspects of the project, including all scenarios.

  Shortcut Key: **Delete**

- **Select\All –** Select all of the elements in the current project.

  Shortcut Key: **Ctrl + A**

- **Select\By Element\[Element Type]** – Select all elements of a certain type such as all pipes or all junctions.
- **Select\By Selection Set** – Select the elements contained in a predefined selection set.
- **Select\Clear Selection –** Reset (empty) the current selection set.
- **Find Element** – open the **Find Element** dialog, which allows you to locate an element and bring it to the center of the drawing pane. This element search is based on the element label (note that this is not case sensitive).

  Shortcut Key: **Ctrl + F**

- **Drawing Review –** Open the **Drawing Review** window to isolate elements that may need to be scrutinized for potential problems (orphaned nodes, elements with messages, superposed nodes, etc).

In AutoCAD mode:

- **(WaterCAD) Undo [Last Action Performed]** – Reverse the last reversible action performed. Reversible actions include such things as element creation, deletion, editing, and moves. The effects of some model actions cannot be reversed, such as calculation, database synchronization, scenario creation, and tabular edits. Additionally, to ensure that the model is maintained in a consistent state the undo/redo history will be flushed whenever an irreversible menu or button command is issued.
- **(WaterCAD) Redo [Last Action Undone] –** Reverse the effects of the last undone action. Any action that can be undone can be redone.
- **Cut –** Delete the selected entities and place them on the Windows clipboard. These items can be pasted into other Windows programs, or back into AutoCAD.

  Shortcut Key: **Ctrl+X**

- **Copy –** Place the selected entities from the current AutoCAD drawing on the Windows clipboard.

  Shortcut Key: **Ctrl+C**

- **Paste –** Place the items on the Windows clipboard into the current AutoCAD drawing.

  Shortcut Key: **Ctrl+V**

- **Paste Special –** Place special items located on the Windows clipboard, such as Excel Spreadsheets and Word documents, into the current AutoCAD drawing.
- **Select\All** – Select all of the elements in the current project.
- **Select\By Element\[Element Type] –** Select all elements of a certain type, such as all pipes or all junctions.
- **Select\By Selection Set** – Select the elements contained in a predefined selection set.
- **Select\Clear Selection –** Reset (empty) the current selection set.

- **Edit Element –** Open an element's dialog. Select this item from the pull-down menu and click the element you wish to edit.
- **Edit Elements –** Edit a group of elements. Select this item from the pull-down menu, then select a group of elements using the crosshairs or by windowing an area. After the elements have been selected, right-click and the **Table Manager** dialog will appear. The selected elements will be reported in the tables.
- **Modify Elements\Scale Elements –** Scale the symbols representing the elements in the current selection set.
- **Modify Elements\Rotate Labels –** Rotate the labels of the elements in the current selection set.
- **Modify Pipes\Insert Bend –** Insert a bend along a selected Pressure Pipe Element.
- **Modify Pipes\Remove Bend –** Delete a bend along a selected Pressure Pipe Element.
- **Modify Pipes\Remove All Bends –** Delete all the bends along a selected Pressure Pipe Element.
- **Modify Pipes\Change Widths** – Change the Width of the lines representing pipes.
- **Change Entities to Pipes –** Build network elements from AutoCAD entities. This command will start the Polyline to Pipe Wizard, which will help you convert the desired polylines representing geographical data into pipes.

A similar command called 'Import Polyline to Pipe' is available in the Stand-Alone version (under the File pull-down menu).

- **Find Element** – Open the **Find Element** dialog, which allows you to locate an element and bring it to the center of the drawing pane. This element search is based on the element label and is not case sensitive.
- **Review Drawing** – Open the **Drawing Review** window, which is used to isolate elements that need to be scrutinized for potential problems (orphaned nodes, elements with messages, superimposed nodes, etc).

2.2.3 Analysis Menu

The **Analysis** menu contains items useful for managing calculations. These include the scenario and alternative managers and the calculation commands.

- **Scenarios** – Access the Scenario Manager, allowing you to analyze and recall an unlimited number of "*What If?*" alternative calculations for your model.
- **Alternatives** – Access the Alternative Manager, allowing you to organize your data into building blocks to be combined to form scenarios.
- **Patterns –** Access the Hydraulic and Constituent load managers, allowing you to define automatic time-variable changes within the system.
- **Zones**- Define zones in which to place network elements.
- **Cost Computing** – Open the **Cost Manager** in order to view, edit, or perform Cost Estimating calculations.

  Toolbar Button: 

- **Calculation**– Open the **Calculation** dialog, which gives you access to items such as calculation options and referenced alternatives.

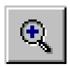
  Toolbar Button: 

2.2.4 View Menu

In both AutoCAD and Stand-Alone mode, the **View** menu provides access to tools dealing with the drawing pane, toolbar visibility, and so forth. Note that the Stand-Alone menu items do not include panning, since it can be accomplished simply by using the scroll bars adjacent to the drawing pane.

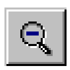
In Stand-Alone mode, you are provided with the following tools:

- **Zoom In –** Enlarge the current view of the drawing.

  Toolbar Button: 

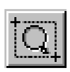
  Shortcut Key: **+** (Keypad)

- **Zoom Out –** Reduce the current view of the drawing.

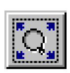
  Toolbar Button: 

  Shortcut Key: - (Keypad)

- **Zoom Window** – Activate the user-defined zoom tool. This tool enables you to select the corners of the area within the drawing pane that you wish to enlarge.

  Toolbar Button: 

- **Zoom Extents** – Reset the drawing zoom factor such that all elements are displayed in the drawing pane.

  Toolbar Button: 

- **Zoom Previous** – Return the drawing pane to the most recent view.

  Toolbar Button: 

- **Zoom Center** – Open the **Zoom Center** dialog, which enables you to specify the central coordinates and zoom factor to change the view in the drawing pane.

- **Aerial View –** Enable or disable the **Aerial View** window. This window allows you to display a second view of the drawing at a larger scale.

- **Quick View –** Enable or disable the **Quick View** window, which allows you to quickly view input and output data for any element.

  Toolbar Button: 

- **Toolbars\Standard** – Toggle the display of the **Standard** toolbar at the top of the window, which provides shortcuts to the most commonly used commands.

- **Toolbars\Analysis Toolbar** – Toggle the display of the **Analysis** toolbar, which includes the scenario selection list and buttons to access the **Scenario Manager** and **Cost Manager**.
- **Status Pane** – Toggle the display of information at the bottom of the window regarding your current project.
- **Background –** Toggle the visibility of the project's DXF background. Note that if there is no DXF background specified for the current project, this menu item will be disabled.

In AutoCAD mode, refer to the AutoCAD on-line help.

2.2.5 Tools Menu

The **Tools** menu provides general tools for placing or modifying graphical elements, annotating, color coding, contouring, changing the projects options, etc.

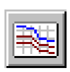
- **Selection Sets** – Access the **Selection Set** dialog, which allows you to create selection sets of elements based on element labels, element types, filters, and other means.
- **Color Coding** – Open the **Color Coding** dialog, which allows you to control the display of elements based on value ranges such as pipe diameter, hydraulic grade, and so forth.

  Toolbar Button: 

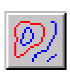
- **Element Annotation** – Access the **Element Annotation** dialog, which allows you to display element attribute labeling, such as pipe diameter and pipe flow.

  Toolbar Button: 

- **Profiling –** Open the **Profile Setup** dialog, which allows you to generate a profile of your piping system along a specified path.

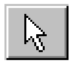
  Toolbar Button: 

- **Contouring** – Access the **Contour Map Manager** to create and view contours.

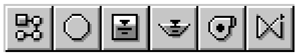
  Toolbar Button: 

- **Relabel Elements –** Open the **Relabel Elements** dialog, which enables you to renumber some or all of your project elements.
- **Element Labeling –** Set the format for the labels applied to elements as they are added to the drawing.
- **Prototypes –** Specify the default values for new network elements.
- **Engineering Libraries –** Declare the paths to and edit the libraries used in this project.
- **User Data Extension –** Open the **User Data Extension** dialog, where you can add and define custom data fields. For instance, you can add new fields such as the pipe installation date.
- **FlexUnits –** Open the **FlexUnits** dialog, where you can control units and display precision for any parameter. Note that you can also change the unit and display precision of variables from several other areas within the program.

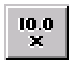
- **Layout\Select –** Activate the selection tool used to highlight elements. Once elements are selected, they may be moved or edited.

  Toolbar Button: 

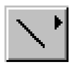
- **Layout\[Element Type] –** Activate the corresponding element tool to place elements in the graphical editor.

  Tool Palette: 

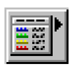
- **Layout\Spot Elevation –** Activate the spot elevation tool, used to add spot elevations.

  Toolbar Button: 

- **Layout\Graphic Annotation –** (Stand-Alone mode only). Activate various annotation tools, which enable you to add lines, borders, and text elements to the project drawing.

  Toolbar Button: 

- **Layout\Legend –** Activate the legend tool used to add a key for the current drawing color coding for links and nodes.

  Toolbar Button: 

- **Options –** Specify settings for the current project, such as the friction method, coordinate system, unit system, and auto-prompting.

- **Element Properties** – (AutoCAD mode only). Control on which layer each type of object and its associated text is placed. Also controls the text style for labels and annotations.

- **Preferences** – (AutoCAD mode only). Activate the AutoCAD Preferences command in order to edit general AutoCAD settings.

2.2.6 Draw Menu (in AutoCAD Mode Only)

This is only available in the AutoCAD mode. The Draw menu provides access to a variety of AutoCAD commands. Refer to the standard AutoCAD help system for more information.

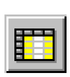
2.2.7 Report Menu

The **Report** menu provides access to a collection of preformatted textual and graphical reports. Furthermore, the report menu provides access to FlexTables, which enable you to create your own custom reports.

- **Report\Element Details** – Open the **Detailed Reports** dialog, which enables you to print detailed reports for any set of elements.

- **Report\Element Results** – Open the **Analysis Results** dialog, which enables you to print reports of the results for any set of elements.

- **Tables** – Access the **Table Manager**, which enables you to open predefined tables or generate your own custom tables.

  Toolbar Button: 

- **Scenario Summary** – Generate a report for the current scenario, including an alternative summary, calculation options, and so forth.
- **Project Inventory –** Generate a report summarizing the project elements, including the number and breakdown of pipes, the number of manholes, and so forth.
- **Plan View –** Generate plan view printable reports of the network for either the current drawing display (Current View) or the entire drawing extents (Full View).

2.2.8 Help Menu

The **Help** menu contains items that relate to on-line documentation for WaterCAD (which includes the information contained in the printed documentation, as well as updated information and built-in tutorials).

Help menu items can also be accessed from the **Help** button:

- **Contents** – Open the Table of Contents for the on-line Help.

  Toolbar Button: 

- **How to Use Help** – Access instructions for using the Help system.
- **AutoCAD Help Topics** – Access the AutoCAD on-line Help.
- **Release Notes** – Provide the latest information on the current version of WaterCAD. This topic, which takes the place of a ReadMe file, includes information about new features, tips, performance tuning, and other general information.
- **Services** – The services menu items will open an Internet browser on Haestad Methods internet site or a local page that provides an overview of the services and products offered by Haestad Methods. In the local page, accessed by selecting **Content**, there are links to Haestad Methods Internet sites, which are updated frequently.
- **Welcome Dialog –** (In Stand-Alone mode only). Open the **Welcome** dialog, which is also shown at program startup.
- **Tutorials –** Access the interactive tutorials, which guide you through many of the program's features. Tutorials are a great way to become familiar with new features.
- **Using WaterCAD** – Open a Help topic with an Introduction to WaterCAD and related elementary information.
- **How Do I** – Provide instructions for tasks commonly performed within the program, as well as frequently asked questions.
- **About WaterCAD –** Open a dialog displaying product and registration information.

2.3 WaterCAD Toolbars

The toolbar buttons are grouped based on functionality, so element creation tools are all together in the tool palette, results tools are all together on the tool pane, and so forth.

In AutoCAD mode, some tools are provided by AutoCAD itself (such as file, open and save, zoom, and so forth), so these tools are not included on the WaterCAD toolbars.

2.3.1 Tool Pane Summary

The tool pane contains buttons for project management, data management, and results presentation.

File Tools (Stand-Alone Only)

New – Create a new project.

Open – Open an existing project.

Save – Save the current project.

Print Preview – View a print preview of the current view.

Zoom Tools (Stand-Alone Only)

Zoom Extents – Zoom to the full extents of the drawing.

Zoom In

Zoom Out

Zoom Window – Zoom to an area selected by you.

Zoom Previous – Zoom to the previous view.

Calculation and Data Management Tools

GO – Open the **Calculation** dialog for the current scenario.

Tabular Reports- Open the **Table Manager** dialog.

Reporting Tools

Color Coding – Color code the network.

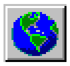
Annotation – Annotate elements with input or output data.

Profile – Open the **Profile** wizard to develop a network profile.

Contours – Open the **Contours** window to generate countours of various atributes.

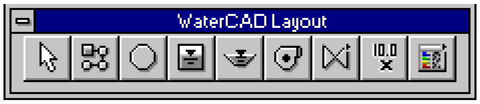
Quick View – Open the **Quick View** window for easy data viewing.

Updates and Help Tools


Globe – If you are connected to the Internet, this will take you to Haestad Methods' web site for product updates and other services.


Help – Access the on-line help system.

2.3.2 The Tool Palette



The tool palette contains a Select tool, Network Element tools, and Annotation tools.

- The Select tool allows you to select elements for group editing, detailed reporting, deleting, or moving elements.
- The Network Element tools allow you to add elements to your network. These tools can also be used to split pipes, morph nodes, and add spot elevation elements.
- The Annotation tools can be used to add legends to your drawing. In Stand-Alone mode, you can also add polylines, borders, and text to your drawing.

Click a tool to select it as the active tool. In Stand-Alone mode, when a tool is selected it will be highlighted, and the cursor appearance will change to reflect the active tool.

In Stand-Alone mode, right-click the tool palette to access the Prototype Manager for setting the default data for each type of network element.

2.3.3 Analysis Toolbar

The analysis toolbar displays the active scenario and provides a means for changing the current scenario and accessing the scenario manager. All input and output information displayed in the tables, profiles, element dialogs, and annotation will be related to the active scenario shown in the analysis toolbar.

You can change the current scenario from the list box. You can access the **Scenario Manager** by clicking the **Scenario Manager** button  and the **Cost Manager** by clicking the **Cost Manager** button .

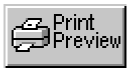
2.3.4 Other Toolbar Buttons

Some of the following toolbar buttons appear on secondary windows (such as the **Print Preview** window and the **Profile** window) available throughout the program:

- Print Preview Command
- Page Up/Down
- File

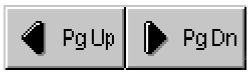
- Copy to Clipboard
- Print
- Options
- Close
- Help

Print Preview Command



Open a Print Preview on the contents of the current window.

Page Up/Down

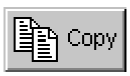


Navigate between pages of a multi-page report.

File Button

Export the data in the current window to a file format that can be used by other applications (such as DXF and ASCII text files).

Copy to Clipboard



Copy data to the clipboard, where it can be pasted into most Windows based spreadsheets, databases, and word processors.

Print

Print the contents of the current window.

Options

Options vary depending on the context. They may include things such as layer control or background color for the current window.

Close

Close the current window.

2.4 The Status Bar

The status bar (located along the bottom of the main application window) provides relevant information about the current state of the model and information about the drawing pane. The data displayed in the status bar includes:

- General Status Information
- DXF Background Status
- Cursor Location

- Calculation Results Status
- File Status

The AutoCAD status bar also contains relevant messages, but deals with your AutoCAD drawing status rather than your hydraulic project. For more information about AutoCAD's status bar, please refer to AutoDesk's documentation.

2.4.1 General Status Information

General status information includes any messages that relate to the user's current activities. These messages include information such as pull-down menu command descriptions, currently selected elements, and indications regarding the progress of an executing command.

2.4.2 DXF Background Status

This area of the status bar simply indicates whether or not a DXF background is currently visible for the active project.

2.4.3 Cursor Location

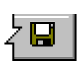
The status bar displays the current X and Y (or Northing and Easting) coordinates for the cursor's position within the drawing pane.

2.4.4 Calculation Results Status

In Stand-Alone mode, if the current calculation results are out-of-date or otherwise invalid, an indicator will appear in the status bar that signifies that the results no longer match the state of your input data. If the results are currently valid, no such indicator will appear.

2.4.5 File Status

If changes have been made since the last time the project file was saved, an image of a diskette appears in the status bar. If the file is currently in a saved state, no such image will appear.



Notes

Chapter 3

Quick Start Lessons

The purpose of Chapter 3 is to provide step-by-step lessons, in order to give you hands-on experience with many of the features and capabilities of WaterCAD. These detailed lessons will assist you in getting started with the exploration and use of the software. Before proceeding with the lessons, however, it is a good idea to run through the brief online tutorials, accessed from the **Help** menu. These interactive tutorials will take you rapidly through overviews of key program features.

Another way to become acquainted with WaterCAD is to run and experiment with the included sample files, located in the **Haestad\Wtrc\Sample** directory. Remember, you can right-click or press the **F1** key to access the context-sensitive online Help at any time.

In order to follow these lessons, you can either do them in sequence, since each lesson uses the results of the previous ones, or do the lessons in any order, using the catch up files located in Haestad\Wtrc\Lesson.

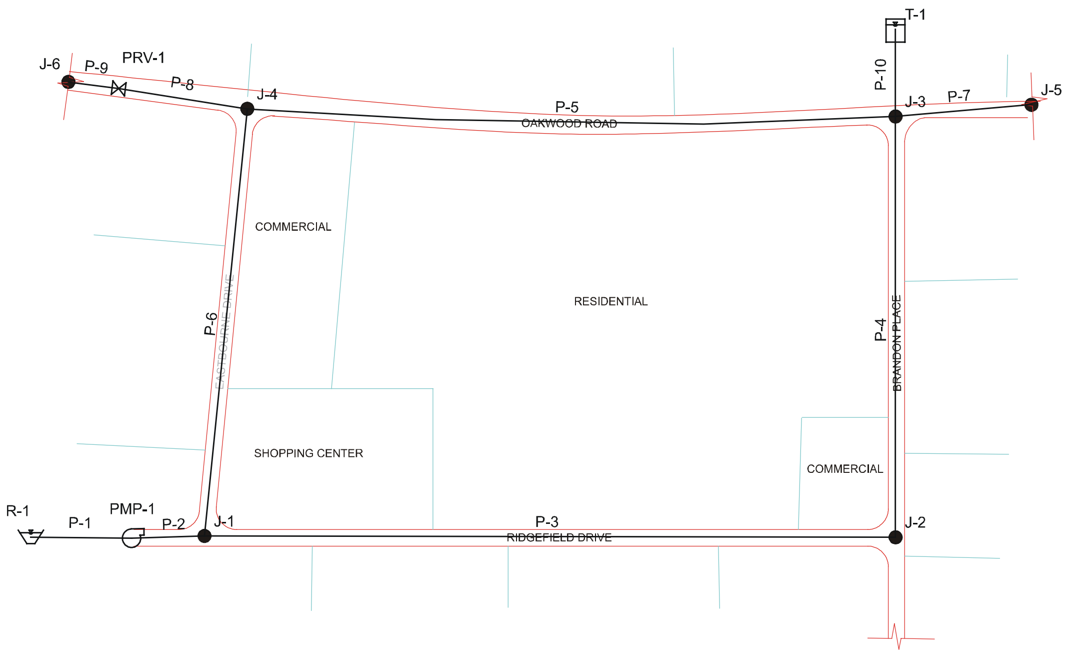
3.1 Lesson 1 – Constructing a Network, Steady State Analysis

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

In constructing a distribution network for this lesson, you do not need to be concerned with assigning labels to pipes and nodes, because WaterCAD will assign labels automatically. When creating a schematic drawing, pipe lengths are entered manually. In a scaled drawing, pipe lengths are automatically calculated from the position of the pipes' bends and start and stop nodes on the drawing pane.

In this lesson, you will create and analyze the network shown below. You will use a scaled background drawing for most of the network; however, four of the pipes are not to scale, and will have user-defined lengths.

Water Distribution Network



In this network, the modeling of a reservoir connected to a pump simulates a connection to the main water distribution system. Simplifying the network in this way can approximate the pressures supplied to the system at the connection under a range of demands. This type of approximation is not always applicable, and care should be taken when modeling a network in this way. It is more accurate to trace the network back to the source.

If at any time during this lesson the program prompts, "Do you wish to reset all calculated results to N/A?," click NO.

3.1.1 Part 1 – Creating a New Project File

 AutoCAD specific instructions

 Stand-Alone specific instructions

1. Double-click the WaterCAD desktop icon to start WaterCAD Stand-Alone. If the **Welcome to WaterCAD** dialog appears, select the **Close** button.

2. Open the **Global Options** tab, accessed from the **Tools\Options** pull-down menu. Since we will be working in metric units, click the **Unit System** selection box, and select **System International**. Click **OK**.

3. In the pull-down menus, select **File\New**. Click **No** when asked if you want to save the current project.

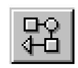
4. In the **Create Project File As** dialog, double-click the 'Lesson' folder, enter the file name 'MyLesson1.wcd' for your project, and click **Save**. The **Project Setup Wizard** will open.

1. Double-click the WaterCAD desktop icon to start WaterCAD for AutoCAD. Open the **Global Options** tab, accessed from the **Tools\Options** pull-down menu. Since we will be working in metric units, click the **Unit System** selection box, and select **System International**. Click **OK**.

2. Choose **Open** on the **File** pull-down menu. When prompted, do not save changes to the current drawing. If the **Select File** dialog opens, move to step 3. Otherwise, do the following:

   Press the **Esc** key. At the command prompt, type **filedia**, press the **Enter** key to enter the command, and then enter a new value of **1**. Choose **Open** from the **File** pull-down menu again, and do not save changes to the current drawing. Note that the fieldia variable controls whether some AutoCAD commands appear as dialogs or simply at the command prompt.

3. Select the existing AutoCAD file 'Lesson1.dwg' from the Wtrc\Lesson folder. With the drawing open, select **File\Save As** from the pull-down menu. In the **Save Drawing As** dialog, double-click the 'Lesson' folder, enter the filename as 'MyLesson1.dwg' and click **Save** to save the file in your \Wtrc\Lesson directory.

4. Now, select the **Layout Elements** tool  in the WaterCAD toolbar. Then, move the cursor onto the drawing pane and right-click to select **Reservoir** from the pop-up menu. Left-click once on the approximate location of reservoir R-1 (see diagram above). You will be prompted to set up the project. Click **Yes** to open the **Project Setup Wizard**.

The remaining commands are for both Stand-Alone and AutoCAD modes. You would proceed with these instructions after completing the instructions above for either the AutoCAD specific instructions or the Stand-Alone specific instructions.

1. In the Project Setup Wizard, title the project 'Lesson 1 – Steady State Analysis' and click the **Next** button.

2. Choose your desired settings. For this lesson, use the program default values. Click the **Next** button.

3. Select the **Scaled** radio button located under the **Drawing Scale** option. Set the horizontal scale to 1 mm = 4000 mm, and the vertical scale to 1 mm = 400 mm. **Stand-Alone users only**: Select the Browse button next to the Background Filename box. Select 'Lesson1.dxf' from the 'Lesson' folder and click **Open**. **All users**: Click the **Next** button to continue.

4. The element **prototype** buttons allow you to set default values for each element type. We will use the default prototype values in this lesson, so click the **Finished** button.

3.1.2 Part 2 – Laying Out the Network

Users in AutoCAD mode have already completed the first step below, and should proceed to step (2).

1. To draw the skeletonized water distribution network shown previously, select the **Layout Elements** tool  from the toolbar. Then, move the cursor onto the drawing pane and right-click to select **Reservoir** from the pop-up menu. Left-click once on the approximate location of reservoir R-1 (see the preceding Water Distribution Network diagram).

2. Next, move the cursor to the location of pump P-1. Right-click and select **Pump** from the pop-up menu. Left-click once to place it. Place junction J-1 by right-clicking, selecting **Junction** from the pop-up menu, and left-clicking on the appropriate location.

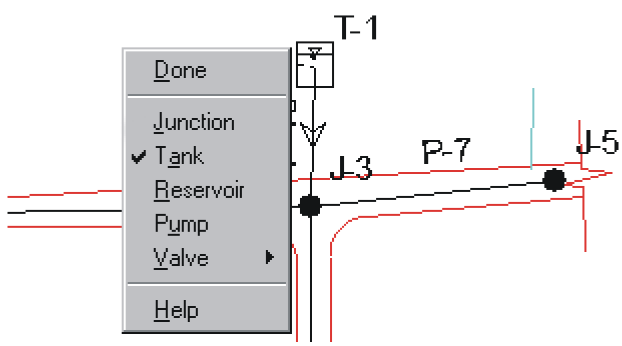
3. Proceed with laying out the network by placing junctions J-2, J-3, and J-4. Close the loop by selecting junction J-1. Right-click and select **Done** from the pop-up menu.

> **To construct a pipe with bends in the Stand-Alone version, hold down the Ctrl key and left-click on the location of the bend. Then, release the Ctrl key to enter the next element. In the AutoCAD version, right-click and select Bend from the pop-up menu, then insert the bend.**
>
> **You can insert bends after a pipe is constructed by right clicking the pipe in Stand-Alone mode and selecting Bend\Add Bend. Then, drag the new vertex to the appropriate location. In AutoCAD mode, select Edit\Modify Pipes\Insert Bend, click the pipe, and click to place the new vertex.**

4. Select the **Layout Elements** tool again, and left-click on junction J-3. Move the cursor to the location of J-5, and left-click to insert the element. Right-click and select **Done**.

5. Insert the PRV (**Valve\PRV** on the pop-up menu), junction J-6, and the tank by selecting the **Layout Elements** tool and placing the elements in their appropriate locations. Be sure to lay out the pipes in numerical order (P-7 through P-9), so that their labels correspond to the labels in the diagram. Right-click and select **Done** from the pop-up menu to terminate the **Layout Elements** command.

6. Insert the tank, T-1, and the pipe connecting it to node J-3. Right-click and select **Done**. The network layout is now complete.

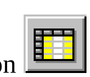


7. Save the WaterCAD network by clicking the Disk icon on the toolbar or by choosing **File\Save**.

3.1.3 Part 3 – Entering Data

There are four ways to enter and modify element data in WaterCAD:

- **Dialogs** – You can use the **Select** tool and double-click an element to bring up its editor dialog. In AutoCAD, click the element once with the **Select** tool to open the element's editor dialog.

- **FlexTables** – You can click on the **Tabular Reports** button  to bring up dynamic tables that allow you to edit and display the model data in a tabular format. You can edit the data as you would in a spreadsheet.

- **Database Connections** – The database connection feature allows you to import and export element data directly from external sources such as Excel spreadsheets, GIS, Jet Databases like Microsoft Access, or any other ODBC database. This process is further explained in the chapter on Database Connections, in the Database Connections tutorial, and in Lesson 7.
- **Alternative Editors** – Alternatives are used to enter data for different "what-if" situations used in Scenario Management. This topic is covered extensively in the Scenario Management chapter, in the Scenario tutorial, and in Lesson 3.

3.1.4 Part 4 – Entering Data through Dialogs

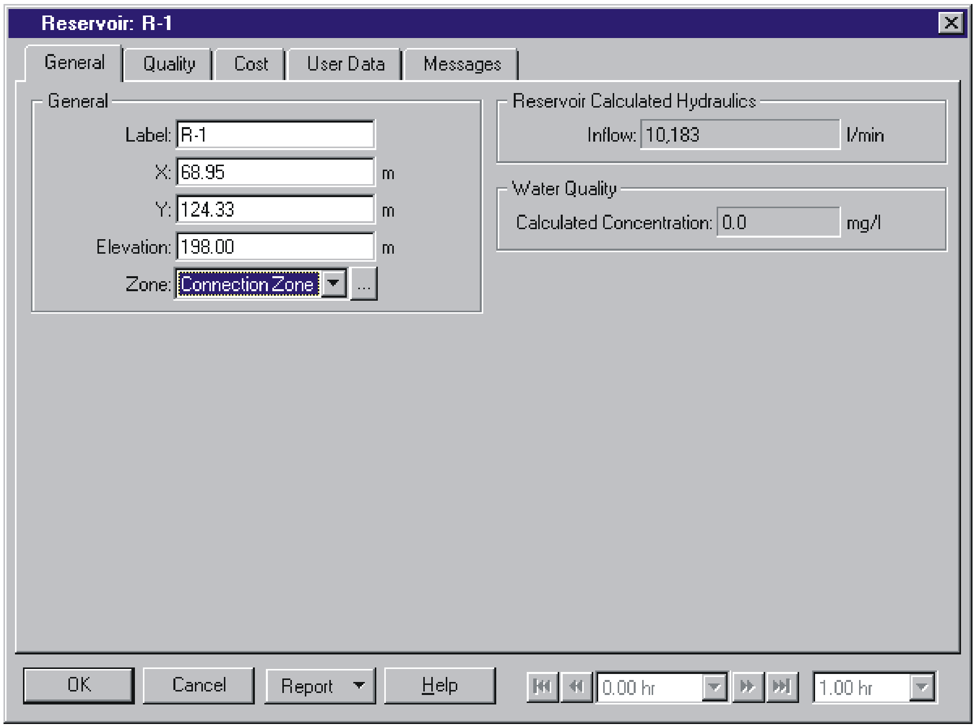
To access an element's dialog in Stand-Alone mode, simply double-click the element with the cursor. In AutoCAD, first click the **Select** tool on the toolbar, then click the element whose attributes you wish to modify.

1. Open the Editor dialog of the reservoir, R-1, and select the **General** tab. Enter the hydraulic grade line elevation from the following Reservoir Data table.

Reservoir Data

	General Tab	
Reservoir	**Elevation (m)**	**Zone**
R-1	198	Connection Zone

2. Click on the **ellipses (…)** button next to the **Zone** field. This action opens the **Zone Manager**. Click **Add**, then enter a label for the new pressure zone, 'Connection Zone'. Click **OK**, and **OK** again to exit the **Zone Manager**.
3. Finally, select the zone you just created from the **Zone** list box, and then click **OK** to close the **Reservoir** editor.



4. Open the element editor for the tank, T-1. Enter the data from the table below. Leave the other parameters set to their default values. Click **OK** to exit the dialog.

Tank Data

	General Tab	Section Tab					
Tank	**Zone**	**Section**	**Max. Elev. (m)**	**Initial Elev. (m)**	**Min. Elev. (m)**	**Base Elev. (m)**	**Diameter (m)**
T-1	Zone-1	Constant Area	226	225	220	200	8

5. Open the element editor for the pump, PMP-1. Select **Standard (3 Point)** from the **Pump Type** list box. Enter the pump elevation and the discharge curve as given in the Pump Data tables below; however, before entering the first discharge value (3800 l/min), make sure to change the discharge units from m^3/min to l/min. Do this by right clicking in the **Design Discharge** box, selecting **Design Properties**, and selecting **l/min** from the **Units** list box. Also, notice the pump has an upstream pipe and a downstream pipe to define the direction. If the pump is ever going in the wrong direction, simply click the **Reverse** button to change it. In this example, the upstream pipe should be P-1, and the downstream pipe should be P-2. Click **OK** to exit the dialog.

Pump Data

	General Tab	
Pump	**Elevation (m)**	**Pump Type**
PMP-1	193	3 Point

	General Tab	
	Head (m)	**Discharge (l/min)**
Shutoff:	30.0	0
Design:	27.4	3800
Max. Operating	24.8	7500

6. Enter the element editor for the valve, PRV-1. Use the information given in the PRV Data table below. Leave the other parameters set to their default values. Click **OK** to exit the dialog.

PRV Data

	General Tab				
Valve	**Elevation (m)**	**Diameter (mm)**	**Status**	**Settings**	**Pressure (kPa)**
PRV-1	165	150	Active	Pressure	390

7. Enter the data for the junctions as outlined in the following Junction Node data table. However, before entering the demand data, right-click in the **Demand** column, and select **Demand Properties** from the pop-up menu. From the **Units** list box, select **l/min** and click **OK**. Leave all other fields set to their default values.

Use the procedure described in steps 2 and 3 above to create the new Zone, 'Zone-2'.

Junction Node Data

	General Tab		Demand Tab
Junction	**Ground Elevation (m)**	**Zone**	**Demand (l/min)**
J-1	184	Zone-1	38
J-2	185	Zone-1	31
J-3	184	Zone-1	34
J-4	183	Zone-1	38
J-5	185.5	Zone-1	350
J-6	165	Zone-2	356

8. Finally, you need to specify user-defined lengths for pipes P-1, P-7, P-8, P-9 and P-10, since the reservoir, tank, PRV, and nodes J-5 and J-6 are only shown in approximate locations. Select pipe P-1 to open the Pipe dialog. Click the box labeled **User Defined Length** to activate this feature. Then, enter a value of .01 m in the **Length** field. Because you are using the reservoir and pump to simulate the connection to the main distribution system, you want headloss through this pipe to be negligible. Therefore, the length is very small and the

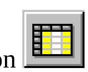
diameter will be large. Repeat this procedure for pipes P-7, P-8, P-9 and P-10, using the user-defined lengths in the table below.

Pipe Data

Pipe	Material	Diameter (mm)	User-Defined Length (m)
P-1	Ductile Iron	1000	0.01
P-2	Ductile Iron	150	N/A
P-3	Ductile Iron	150	N/A
P-4	PVC	150	N/A
P-5	Ductile Iron	150	N/A
P-6	Ductile Iron	150	N/A
P-7	PVC	150	400
P-8	Ductile Iron	150	500
P-9	Ductile Iron	150	31
P-10	Ductile Iron	150	100

3.1.5 Part 5 – Entering Data through FlexTables

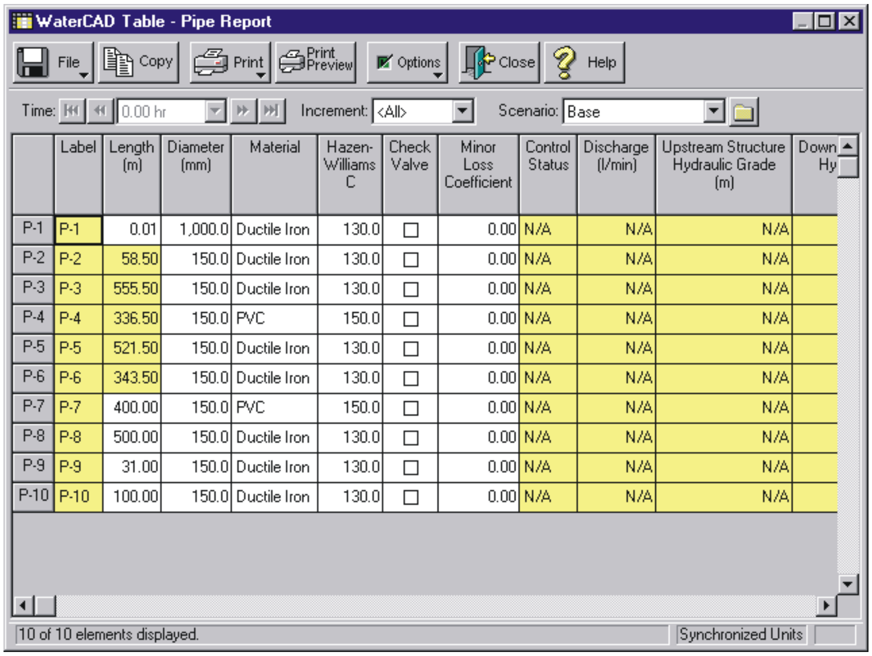
It is often more convenient to enter data for similar elements in tabular form, rather than to individually open a dialog for an element, enter the data into the dialog, and then select the next element. Using tabular reports, you can enter the data as you would enter data into a spreadsheet.

1. To access the tabular report, click the **Tabular Reports** button  in the toolbar.

2. Left-click the Pipe Report and click **OK** to access the report. Note that the white fields are editable, but the yellow fields are not. The pipes may not be in alpha-numeric order in the table. To sort the table by pipe label, right-click the **Label** column heading. Select **Sort\Ascending** from the pop-up menu that appears.

3. For each of the ten pipes, enter the section size and the pipe material as outlined in the Pipe Data table above. Notice that the C values for the pipes will be automatically assigned to preset values based on the material; however, these values could be modified if a different coefficient were required.

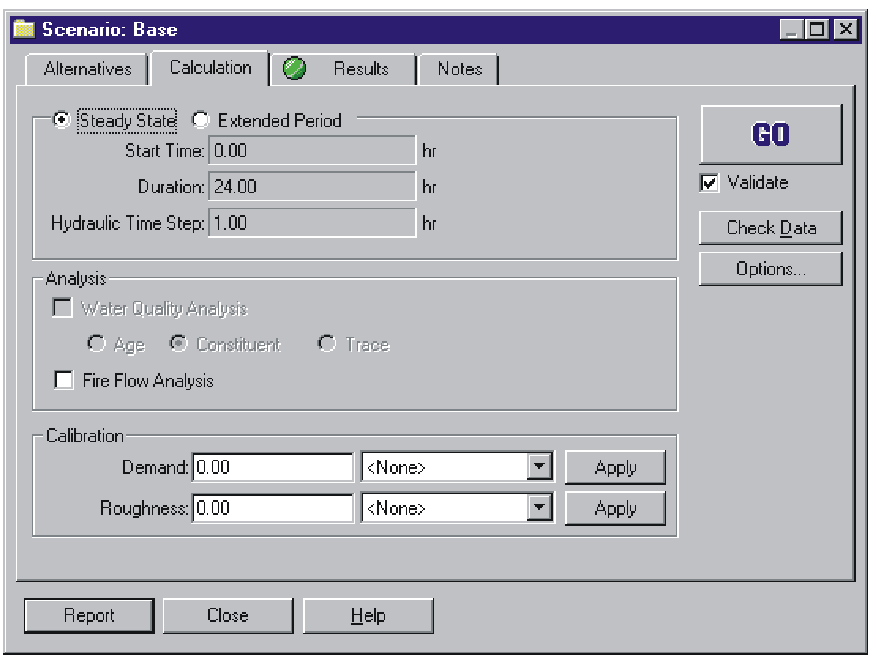
4. Leave other data set to their default values. Click the **Close** button to exit the table when you are finished.



	Label	Length (m)	Diameter (mm)	Material	Hazen-Williams C	Check Valve	Minor Loss Coefficient	Control Status	Discharge (l/min)	Upstream Structure Hydraulic Grade (m)	Down Hy
P-1	P-1	0.01	1,000.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-2	P-2	58.50	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-3	P-3	555.50	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-4	P-4	336.50	150.0	PVC	150.0	☐	0.00	N/A	N/A	N/A	
P-5	P-5	521.50	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-6	P-6	343.50	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-7	P-7	400.00	150.0	PVC	150.0	☐	0.00	N/A	N/A	N/A	
P-8	P-8	500.00	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-9	P-9	31.00	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	
P-10	P-10	100.00	150.0	Ductile Iron	130.0	☐	0.00	N/A	N/A	N/A	

3.1.6 Part 6 – Performing a Steady State Analysis

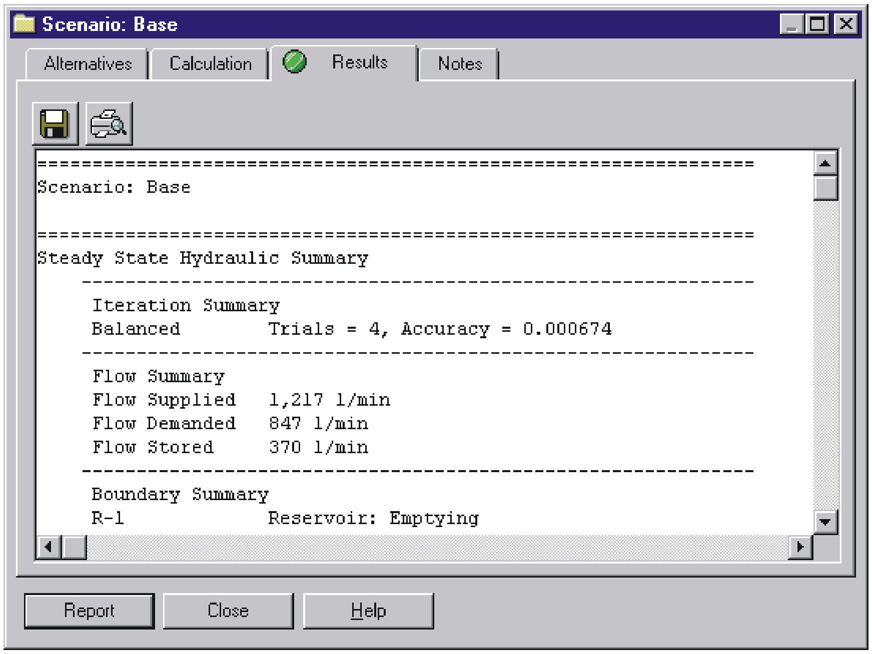
1. Click the **GO** button to bring up the **Calculation** dialog. Make sure that the **Calculation Type** is marked as **Steady State.**



2. Click the **GO** button on the dialog to analyze the model. When calculations are completed, a **Results** report is displayed.

3. The **Results** tab displays a summary of model results. Scroll through the summary to get an idea of the results that are given. There should be a green light displayed on the **Results** tab of the dialog. You can quickly tell if there were warnings or failures with a glance at the

light. A green light indicates no warnings or failures, a yellow light indicates warnings, while a red light indicates problems.



4. Click **Close** when you are done. After a model run, all the calculated fields in dialogs and tabular reports will display results. See Lesson 4 for an overview of the output options available.

Before proceeding to the next lesson, save this project (for instance as 'MyLesson1.wcd' in Stand-Alone mode or 'MyLesson1.dwg' in AutoCAD mode).

3.2 Lesson 2 – Extended Period Simulation

This lesson will illustrate how WaterCAD can model the behavior of a water distribution system through time using an extended period simulation (EPS). An EPS can be conducted for any duration specified by the user. System conditions are computed over the given duration at a specified time increment. Some of the types of system behaviors that can be analyzed using an EPS include how tank levels fluctuate, when pumps are running, whether valves are open or closed, and how demands change throughout the day.

If, at any time during this lesson, the program asks "Do you wish to reset all calculated results to N/A?" click NO.

This lesson is based on the project created in Lesson 1. If you have not completed Lesson 1, then open the project 'Lesson2.wcd' (Lesson2.dwg in the AutoCAD version) from the Wtrc\Lesson directory. If you did complete Lesson 1, then you can use the 'MyLesson1' file you created.

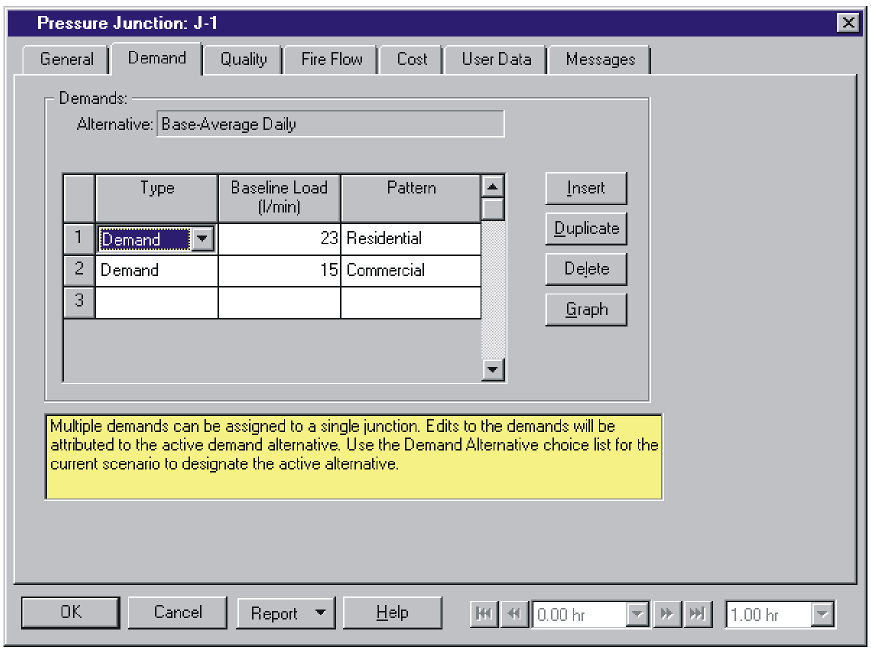
After you have opened the file, select **File\Save As** from the pull-down menu. Type the filename 'MyLesson2' and click **Save**. Select **File\Project Summary**, and change the **Project Title** to 'Lesson 2 – Extended Period Simulation'. Click **OK**.

3.2.1 Part 1 – Creating Demand Patterns

Water demand in a distribution system fluctuates over time. For example, residential water use on a typical weekday is higher than average in the morning before people go to work, and is usually highest in the evening when residents are preparing dinner, washing clothes, etc. This variation in demand over time can be modeled using demand patterns. Demand patterns are simply multipliers that vary with time and are applied to a given base demand, most typically the average daily demand.

In this lesson, you will be dividing the single ‘Fixed’ demands for each junction node in Lesson 1 into two individual demands with different demand patterns. One demand pattern will be created for residential use, and another for commercial use. You will enter demand patterns at the junction nodes through the junction editor dialogs.

1. Open the editor for Junction J-1 and select the **Demand** tab. By default, the demand pattern is set to ‘Fixed.’ In the **Demands** table, leave the first row set to Demand, and set the baseline load to 23 l/min. Click the corresponding cell in the Pattern column, and select the **ellipses (…)** button that appears. This action opens the **Pattern Manager**. Click the **Add** button to create a new pattern for this model.



2. In the **Pattern** dialog, enter the name **Residential** in the **Label** field. Leave the **Start Time** to 0.0 (midnight) and set the **Starting Multiplier** to 0.5. Under **Format**, select the radio button labeled **Stepwise**. The resulting demand pattern will have multipliers that remain constant until the next pattern time increment is reached.

3. In the **Pattern** table, enter the times and multipliers from the table below. Click **OK** when you are finished to return to the Pattern Manager.

Residential Pattern Data

Time from Start (hr)	Multiplier
3	0.4
6	1.0
9	1.3
12	1.2
15	1.2
18	1.6
21	0.8
24	0.5

Note that the multiplier for the last time given (24 hr) must be the same as the Starting Multiplier (0.5). These values are equal because the demand curve represents a complete cycle, with the last point the same as the first.

4. While we are in the Pattern Manager, we will go ahead and create a pattern for commercial demands. Select the **Add** button, and create a pattern labeled **Commercial**, having a **Start Time** of 0.0 and a **Starting Multiplier** of 0.4. Enter the data below into the **Pattern** table. This pattern will also be stepwise. Click **OK** when you are finished to return to the Pattern Manager.

Commercial Pattern Data

Time from Start (hr)	Multiplier
3	0.6
6	0.8
9	1.6
12	1.6
15	1.2
18	0.8
21	0.6
24	0.4

5. Click **OK** to return to the Junction J-1 Editor. In the **Pattern** list box for the first row, select ‘Residential’ from the drop-down list. In the second row, enter a demand of 15 l/min. Select ‘Commercial’ as the pattern for this row. Click **OK** to exit the junction J-1 editor.

6. Go into the **Demands** table in the editor dialogs for junctions J-2, J-3, J-4, J-5 and J-6 and enter the demand data from the table below. We will use the Residential and Commercial demand patterns already created, so just select the appropriate pattern from the list box.

	Demand Tab	
Junction	Residential Demand (l/min)	Commercial Demand (l/min)
J-2	23	8
J-3	23	11
J-4	23	15
J-5	350	N/A
J-6	280	76

7. Now, we will set up an additional demand pattern to simulate a three-hour fire at node J-6. In the **Demand** tab of the junction J-6 Editor, insert an additional demand of 2000 l/min in row 3 of the **Demands** table.

8. Click in the Pattern column for row 3 and select the **ellipses (…)** button to open the **Pattern Manager**. Select the **Add** button to add a new demand pattern. In the **Pattern** dialog, enter a **Label** of '3-Hour Fire', a **Start Time** of 0.00 hr, and a **Starting Multiplier** of 0.00. Select the radio button for **Stepwise** format.

9. Enter the information given below into the pattern table.

3-Hour Fire Pattern Data

Time from Start (hr)	Multiplier
18	1.00
21	0.00
24	0.00

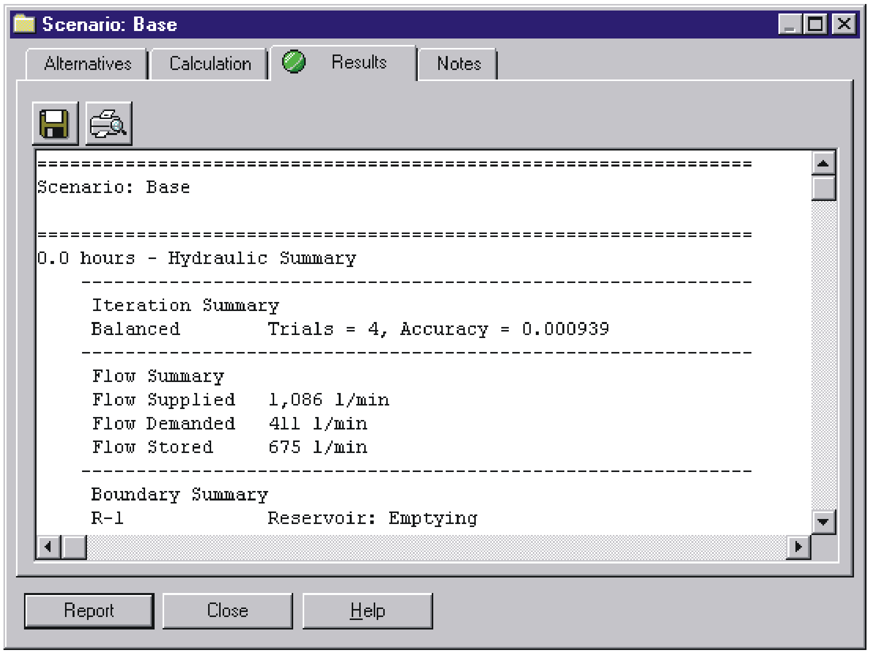
10. After you have filled in the table, select the **Report** button at the bottom of the dialog box. Choose **Graph** from the menu to display a graph of the demand pattern. Notice that the value of the multiplier is zero, except for the period between 18 and 21 hours, when it is 1.0. Since we input the demand as 2000 l/min, the result will be a 2000 l/min fire flow at junction J-6 between hours 18 and 21.

11. Click **Close** to exit the graph, **OK** to exit the Pattern dialog and **OK** again to exit the Pattern Manager. Finally, select your new pattern, '3-Hour Fire', from the **Pattern** selection box in row 3 of the **Demands** table, and click **OK** to exit the Junction Editor.

3.2.2 Part 2 – Running an Extended Period Simulation

1. To run the Extended Period Simulation, click on the **GO** button on the toolbar. Select the radio button for **Extended Period**. Accept the defaults of a **Start Time** of 0.0, a **Duration** of 24 hours, and a **Hydraulic Time Step** of 1 hour. Then, click the **GO** button to run the analysis.

2. After the model runs, the green light on the **Results** tab indicates that there are not any warnings for the analysis and WaterCAD was able to compute a balanced solution for the distribution network. Click the **Close** button.



3. You can view results by opening individual element dialog boxes, clicking the **Report** button to generate detailed reports and graphs for the individual elements, as well as through output tables, color-coding, profiling, contouring, and annotation. For example, open the editor for the PRV and click the Report button. Select Detailed Report from the menu. Scroll and page through the report to view the Calculated Results Summary. Notice that the PRV is throttling, except for hours 18 to 20, when the fire occurs and the downstream pressure setting can no longer be maintained. See Lesson 5 on reporting results for more information.

4. Save this project before proceeding to Lesson 3.

3.3 Lesson 3 – Scenario Management

One of WaterCAD's many powerful and versatile project tools is Scenario Management. Scenarios allow you to calculate multiple "what-if" situations in a single project file. You may wish to try several designs and compare the results, or analyze an existing system using several different demand alternatives and compare the resulting system pressures. A scenario is a set of alternatives, while alternatives are groups of actual model data. Both scenarios and alternatives are based on a parent/child relationship where a child scenario or alternative inherits data from the parent scenario or alternative.

In Lessons 1 and 2, you constructed the water distribution network, defined the characteristics of the various elements, entered demands and demand patterns, and performed steady-state and extended period simulations. In this lesson, you will set up the scenarios needed to test four "what if" situations for our water distribution system. These "what-if" situations will involve changing demands and pipe sizes. At the end of the lesson, you will compare all of the results using the Scenario Comparison tool.

If at any time during this lesson the program asks, "Do you wish to reset all calculated results to N/A?" click NO.

This lesson is based on the project created in Lesson 2. If you completed Lesson 2, open the project you created previously, MyLesson2 in the **Wtrc\Lesson** directory. Otherwise, open the file called lesson3.wcd (lesson3.dwg in the AutoCAD version).

After you have opened the file, select **File\Save As** from the pull-down menu. Type the filename 'MyLesson3' and click **Save**. Select **File\Project Summary**, and change the **Project Title** to 'Lesson 3 – Scenario Management'. Click **OK**.

3.3.1 Part 1 – Creating a New Alternative

First, you need to set up the required data sets, or Alternatives. An Alternative is a group of data that describes a specific part of the model. There are ten Alternative types: Physical, Demand, Initial Settings, Operational, Age, Constituent, Trace, Fire Flow, Cost and User Data. In this example, you need to set up a different Physical or Demand Alternative for each design trial you want to evaluate. Each Alternative will contain different pipe size or demand data.

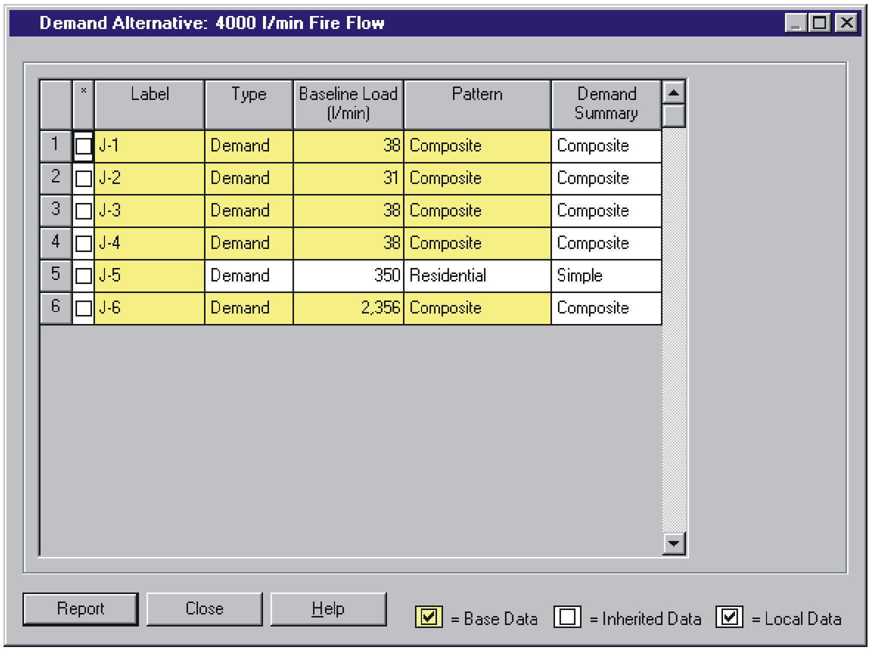
In WaterCAD, we create families of Alternatives from Base Alternatives. Base Alternatives are Alternatives that do not inherit data from any other Alternative. Child Alternatives can be created from the Base Alternative. A Child Alternative inherits the characteristics of its Parent, but specific data can be overridden to be local to the Child. A Child Alternative can, in turn, be the Parent of another Alternative.

1. Select **Analysis\Alternatives** from the pull-down menu. Select the **Demand** tab from the top of the dialog. Currently, there is only one Demand Alternative listed. The Base-Demand Alternative contains the demands for the current distribution system.

2. You can change the default 'Base-Average Daily' demand name to be something more descriptive of our data. Click on the **Rename** button, and enter the new name, 'Average Daily with 2000 l/min Fire Flow'.

3. You would like to add a Child of the Base-Demands Alternative because the new Alternative will inherit most data. Then, you can locally change the data that you want to modify. You will modify the existing demand data by increasing the fire flow component at node J-6 from 2000 l/min to 4000 l/min. Click the **Add Child** button and enter the name '4000 l/min Fire Flow' for the new Alternative, and then click **OK**.

4. The Demand Alternative editor for the new Alternative will appear showing you the data that was inherited from the Parent Alternative. Notice the key at the bottom describing the check boxes. As the key indicates, all of your data is inherited. If you change any piece of data, the check box will become checked because that record is now local to this Alternative and not inherited from the Parent.



5. Click in the **Demand Summary** column for node J-6. A button appears. Click this button to open the **Demand** table for this node. Change the 2000 l/min fire demand to 4000 l/min, and click **OK**. Click **Close** to exit the **Alternative Editor** and return to the **Alternative Manager**, and **Close** again to exit back to the drawing pane.

3.3.2 Part 2 – Editing and Creating Scenarios

Alternatives are the building blocks of a Scenario. A Scenario is a set of one of each of the ten types of Alternatives, plus all of the calculation information needed to solve a model.

As there are Base, Parent, and Child Alternatives, there are also Base, Parent, and Child Scenarios. The difference is that instead of inheriting model data, Scenarios inherit sets of Alternatives. To change the new Scenario, change one or more of the new Scenario's Alternatives. For this lesson, we will create a new Scenario for each different set of conditions we need to evaluate.

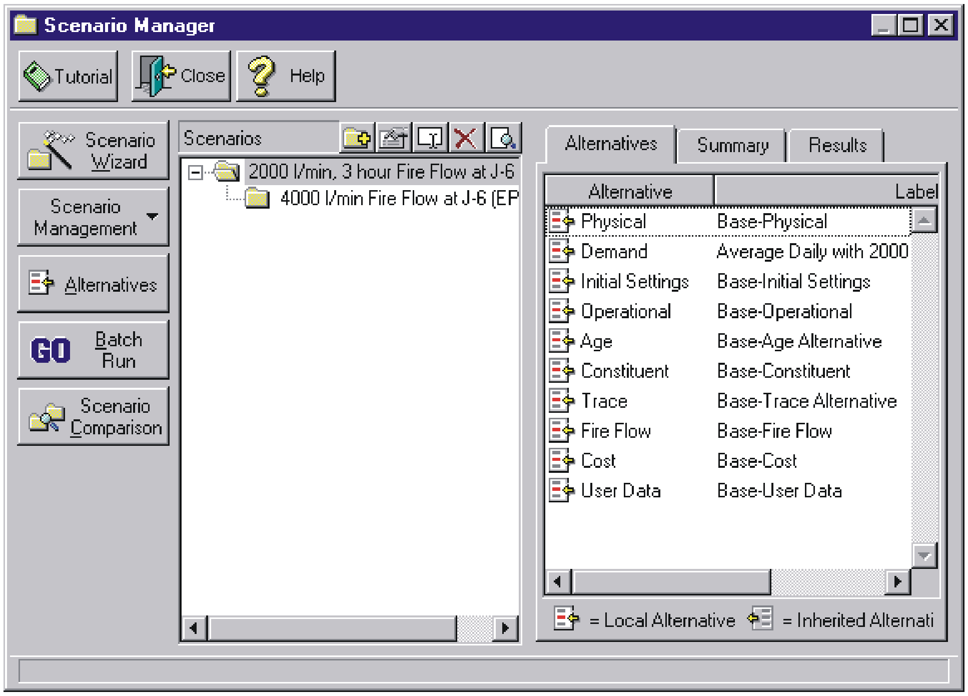
1. Select **Analysis\Scenarios** from the pull-down menus. You are now in the **Scenario Manager**. There is always a default Base Scenario that is comprised of the ten base Alternatives listed in the right pane. The left pane of the Scenario Manager contains a list of the Scenarios. Only the Base is available initially, because we have not created any new Scenarios.

2. You should first rename the Base Scenario as something more descriptive. Click the **Scenario Management** button and select **Rename** from the pull-down menu. The Scenario name in the left pane will become editable. Type in a descriptive name for the Scenario, such as '2000 l/min, 3-hour Fire Flow at J-6 (EPS)' and then press the Enter key.

3. Now, you will create a Child Scenario from our existing Base Scenario, to incorporate our new Demand Alternative. Click the **Scenario Management** menu button, and select **Add\Child Scenario** from the menu. Type in a Scenario name of '4000 l/min Fire Flow at J-6 (EPS)' and click **OK**. A dialog box for the new Scenario appears listing the Alternatives as inherited from the Base Scenario.

4. Your new Child Scenario initially consists of the same Alternatives as its Parent Scenario, except the Demand Alternative should be the new Alternative you created, '4000 l/min Fire Flow'. Check the box next to **Demand**. A selection box for this Alternative appears. Click

on the box, and select the new Alternative from the list. The new Alternative is no longer inherited from the Parent, but is local to this Scenario. Click **Close** to exit the Scenario.

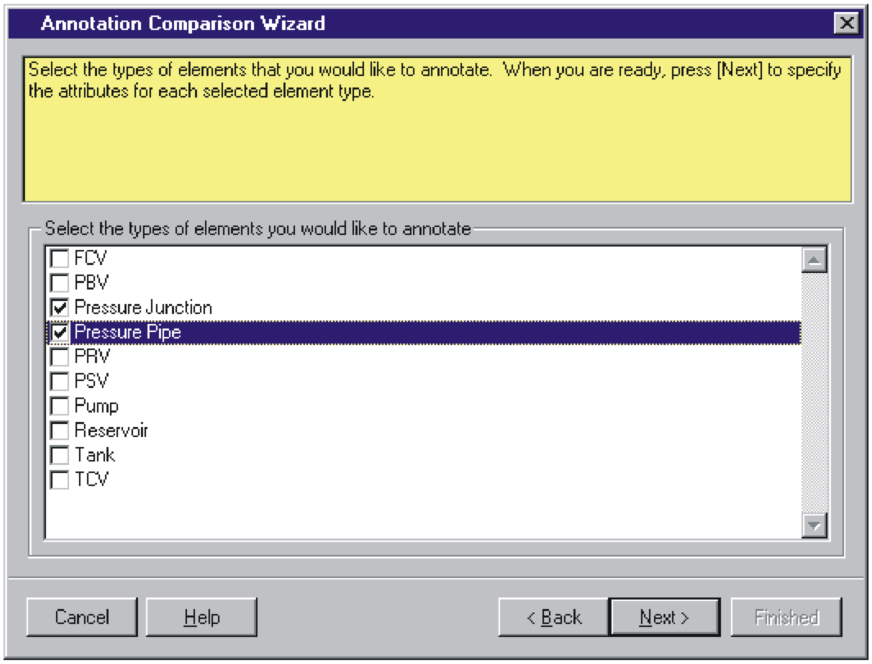


3.3.3 Part 3 – Calculate and Compare

1. You are going to calculate both of the Scenarios at the same time using the **Batch Run** tool. Click the **Batch Run** button on the left side of the **Scenario Management** dialog. Check the boxes next to both Scenarios, and then click the **Batch** button. Click **Yes** at the prompt to run the batch for two Scenarios. When they have finished computing, click **OK**.

2. You can see the results for each Scenario by highlighting it in the Scenario list. Click the **Results** tab at the top right in the dialog box to see the selected Scenario's results. We can see that the Scenarios are different, but what exactly is different about them? We will use the **Scenario Comparison** tool to compare the results. Click the **Scenario Comparison** button to start the **Annotation Comparison Wizard**. Select the '2000 l/min, 3-hour Fire Flow at J-6 (EPS)' Scenario in the first list box and the '4000 l/min Fire Flow at J-6 (EPS)' in the second list box. Click the **Next** button.

3. We will compare the results for pressures at the junctions and velocities in the pipes, so click the check boxes next to **Pressure Junction** and **Pressure Pipe**, and then click **Next**.



4. Select **Pressure** from the list box under the **Attributes** column for **Pressure Junction Annotation**. Edit the entry in the **Mask** column by deleting the label name, so that only the pressure and unit variables (**%v %u**) remain. Click **Next**.

5. Select **Velocity** from the list box under the **Attributes** column for **Pressure Pipe Annotation**. Again, you can edit the label in the **Mask** column. Click **Next**. Check to make sure your annotation is correct, and then click **Finished**.

6. A plan view of the system with annotation displaying the difference between the two Scenarios will appear. To better view the data, maximize the window and use the zoom buttons from the upper toolbar to look at different areas of the model. The difference between the two is found by subtracting Scenario 1 from Scenario 2. For example, say Scenario 1 has a pressure of 50 kPa at a junction, and Scenario 2, which represents a future scenario, has a pressure of 45 kPa at the same junction. Comparing these pressures for Scenario 1 and Scenario 2 would result in annotation stating a difference of -5 kPa. When you have more than two scenarios, you can select different combinations of the scenarios from the two list boxes and click the **Update** button to view the differences between the two. You could also click the **Auto Update** check box and the differences will automatically update every time you change the combination of scenarios and time increments in the list boxes.

7. Check the **Auto Update** box and use the VCR-style buttons to scroll through different increments. Look at the difference in pressures at junction J-6 during the fire flow (between 18 and 21 hours). There is a very large pressure drop due to the substantial increase in demand. In the next part of this lesson, we will look at scenarios to reduce this decrease in pressure.

3.3.4 Part 4 – Physical Alternative

You need to further examine what is going on in the system as a result of the fire flow, and find solutions to any problems that might have arisen in the network as a result. You can review output tables to quickly see what the pressures and velocities are within the system, and create new Alternatives and Scenarios to capture your modifications.

1. Click the **Close** button to exit the **Scenario Comparison Window**. Click **Close** again to exit the **Scenario Manager** and return to the drawing pane.

2. Select the **Tabular Reports** button from the toolbar. Highlight **Junction Report** in the list, and click **OK**. Select '4000 l/min Fire Flow at J-6 (EPS)' in the **Scenario** list box, and set the **Time** to 18.00 hr. Most of the system pressures look acceptable at this time increment; however, the pressure at J-6 is actually negative.

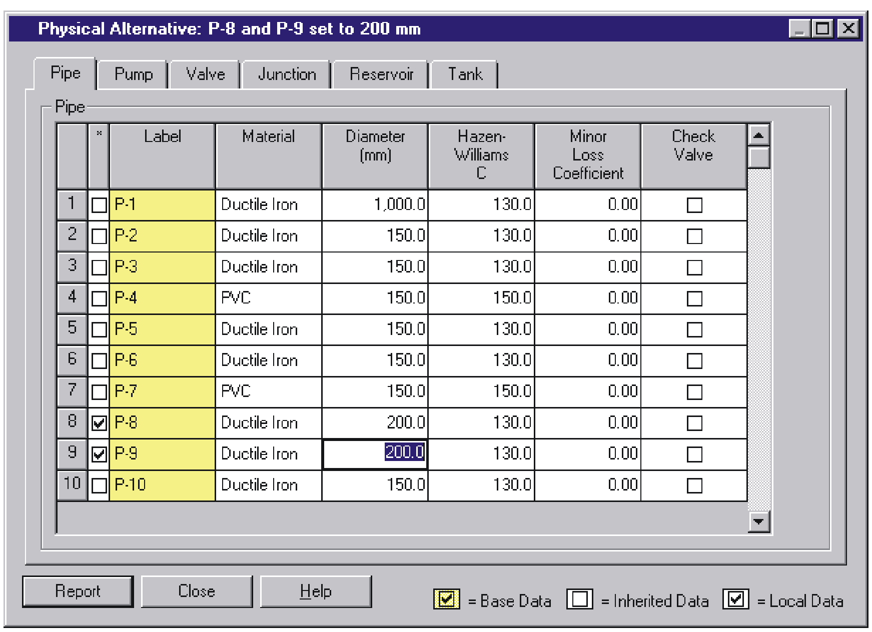
3. Click the **Options** button and select **Table Manager** from the menu. Highlight **Pipe Report**, and click **OK**. In the Pipe Report table, notice that the headloss gradient for pipes P-8 and P-9 are significantly higher than in the rest of the system. We can reduce the headloss gradient by increasing the sizes of these pipes. The pressure at J-6 should increase as a consequence.

4. Click **Close** to exit the table. We will create a new Scenario having a new Physical Alternative with the pipe sizes for P-8 and P-9 increased to 200 mm. From the pull-down menus, select **Analysis\Scenarios**. Highlight '4000 l/min Fire Flow at J-6 (EPS)' in the list of Scenarios, click the **Scenario Management** button, and select **Add\Child Scenario** from the menu.

5. Name the new Scenario 'P-8 and P-9 Set to 200 mm', and click **OK**.

6. Under the **Alternatives** tab of the **Scenario** dialog, check the box for **Physical**. Click the **ellipses (…)** button to open the **Physical Properties Alternatives** dialog. Click the **Add Child** button, and name the new Alternative 'P-8 and P-9 Set to 200 mm'. Click **OK**.

7. Under the **Pipe** tab for this Alternative, change the pipe sizes in the table for pipes P-8 and P-9 from 150 mm to 200 mm. Click the **Close** button to exit the editor, and click **Close** again to exit the **Physical Properties Alternatives** dialog.



	*	Label	Material	Diameter (mm)	Hazen-Williams C	Minor Loss Coefficient	Check Valve
1	☐	P-1	Ductile Iron	1,000.0	130.0	0.00	☐
2	☐	P-2	Ductile Iron	150.0	130.0	0.00	☐
3	☐	P-3	Ductile Iron	150.0	130.0	0.00	☐
4	☐	P-4	PVC	150.0	150.0	0.00	☐
5	☐	P-5	Ductile Iron	150.0	130.0	0.00	☐
6	☐	P-6	Ductile Iron	150.0	130.0	0.00	☐
7	☐	P-7	PVC	150.0	150.0	0.00	☐
8	☑	P-8	Ductile Iron	200.0	130.0	0.00	☐
9	☑	P-9	Ductile Iron	200.0	130.0	0.00	☐
10	☐	P-10	Ductile Iron	150.0	130.0	0.00	☐

8. Select the new Physical Alternative from the list box for the scenario, and click **Close** to return to the Scenario Manager. Select **Batch Run** to run the model. Turn off the check boxes for the first two scenarios, and turn on the check box for 'Pipes P-8 and P-9 Set to 200 mm'. Click **Batch** and select **Yes** to confirm and run the Scenario. Click **OK** when the run is complete.

9. **Close** the Scenario Manager and return to the drawing pane. Select the **Tabular Reports** button from the toolbar, and open the **Junction Report**. In the Scenario list box, select the new Scenario, and examine the pressures at J-6 for 18, 19, and 20 hours. The pressures for this node are now at acceptable levels.

If you would like to learn more about the various results presentation methods available in WaterCAD, see Lesson 4.

Close the open dialogs and save this project before proceeding with Lesson 4, Reporting Results.

3.4 Lesson 4 – Reporting Results

An important feature in all water distribution modeling software is the ability to present results clearly. This lesson outlines several of WaterCAD'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.
- **Profiles**, to graphically show, in a profile view, how a selected attribute, such as hydraulic grade, varies along an interconnected series of pipes.
- **Contouring**, to show how a selected attribute, such as pressure, varies throughout the distribution system.
- **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.

If at any time during this lesson the program asks, "Do you wish to reset all calculated results to N/A?" click NO.

For this lesson, you will use the system from Lesson 3, saved as 'MyLesson3 in the Wtrc\Lesson directory. If you did not complete Lesson 3, you may use the file lesson4.wcd (lesson4.dwg in AutoCAD). After you have opened the file, select **File\Save As** from the pull-down menu. Type the filename 'MyLesson4' and click **Save**. Select **File\Project Summary**, and change the **Project Title** to 'Lesson 4 – Reporting Results'.

3.4.1 Part 1 – Reports

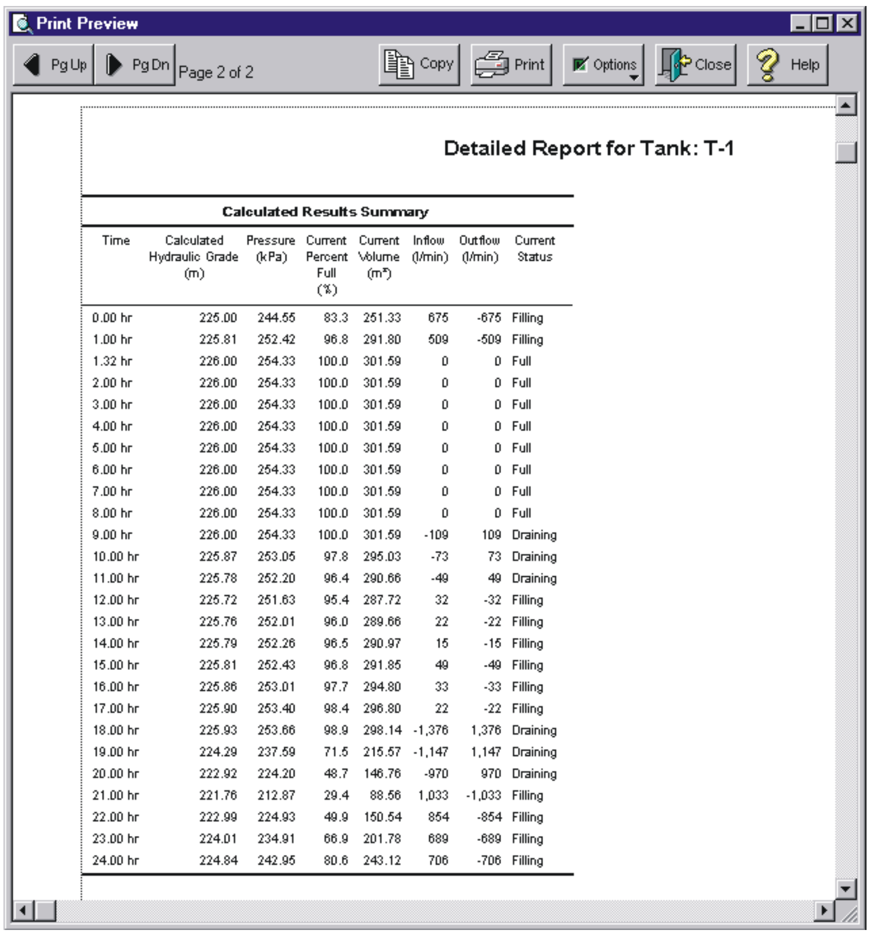
1. Select the '2000 l/min, 3 hour fire flow at J-6 (EPS)' Scenario from the **Scenario** toolbar. Click the **GO** button to open the run dialog, and click **GO** to analyze this Scenario.

2. When the **Results** dialog appears, notice that the **Results** report can be saved to a file or printed using the buttons in the top left corner. This report displays key system characteristics on a formatted page. In an EPS analysis such as this one, these characteristics are displayed

for each time increment. If there were any warnings or problems, they would also appear here.

3. Click **Close**. Note that the results for the current Scenario (the Scenario appearing in the list box in the toolbar) can be accessed at any time by clicking the **GO** button in the toolbar, and then clicking on the **Results** tab.

4. Select the Tank, T-1, and open its **Editor** dialog. Click on the **Report** button at the bottom of the dialog and select **Detailed Report** from the pull-down menu to view a formatted summary report for the tank. On page 2, notice that you can see the tank's status (draining or filling) at each time increment.



Detailed Report for Tank: T-1

Calculated Results Summary

Time	Calculated Hydraulic Grade (m)	Pressure (kPa)	Current Percent Full (%)	Current Volume (m^3)	Inflow (l/min)	Outflow (l/min)	Current Status
0.00 hr	225.00	244.55	83.3	251.33	675	-675	Filling
1.00 hr	225.81	252.42	96.8	291.80	509	-509	Filling
1.32 hr	226.00	254.33	100.0	301.59	0	0	Full
2.00 hr	226.00	254.33	100.0	301.59	0	0	Full
3.00 hr	226.00	254.33	100.0	301.59	0	0	Full
4.00 hr	226.00	254.33	100.0	301.59	0	0	Full
5.00 hr	226.00	254.33	100.0	301.59	0	0	Full
6.00 hr	226.00	254.33	100.0	301.59	0	0	Full
7.00 hr	226.00	254.33	100.0	301.59	0	0	Full
8.00 hr	226.00	254.33	100.0	301.59	0	0	Full
9.00 hr	226.00	254.33	100.0	301.59	-109	109	Draining
10.00 hr	225.87	253.05	97.8	295.03	-73	73	Draining
11.00 hr	225.78	252.20	96.4	290.66	-49	49	Draining
12.00 hr	225.72	251.63	95.4	287.72	32	-32	Filling
13.00 hr	225.76	252.01	96.0	289.66	22	-22	Filling
14.00 hr	225.79	252.26	96.5	290.97	15	-15	Filling
15.00 hr	225.81	252.43	96.8	291.85	49	-49	Filling
16.00 hr	225.86	253.01	97.7	294.80	33	-33	Filling
17.00 hr	225.90	253.40	98.4	296.80	22	-22	Filling
18.00 hr	225.93	253.66	98.9	298.14	-1,376	1,376	Draining
19.00 hr	224.29	237.59	71.5	215.57	-1,147	1,147	Draining
20.00 hr	222.92	224.20	48.7	146.76	-970	970	Draining
21.00 hr	221.76	212.87	29.4	88.56	1,033	-1,033	Filling
22.00 hr	222.99	224.93	49.9	150.54	854	-854	Filling
23.00 hr	224.01	234.91	66.9	201.78	689	-689	Filling
24.00 hr	224.84	242.95	80.6	243.12	706	-706	Filling

5. Every element can generate a report in the same general format, which includes the name of the calculated scenario and a series of tables describing the element's properties and results in detail. You can print this report or copy it to the clipboard using the buttons at the top of the dialog. The report is "What you see is what you get" (WYSIWYG), so it will print or paste into a word processor in the exact format seen on the screen. Click the **Close** button on the report, and then click **OK** to exit the tank dialog.

6. You can also print detailed reports for several elements at one time. In the stand-alone version, use the **Select** tool to draw a window around the elements you want to report, or hold down the shift key while selecting the elements individually. Then, select **Report\Element Details** from the pull-down menus to bring up the Detailed Reports dialog. In the AutoCAD version, start by selecting **Report\Element Details** from the pull-down menus. The crosshairs change into a pickbox. Using the pickbox, select elements from the drawing space

that you want to report (individually or using a window), and then right-click once to bring up the **Detailed Reports** dialog.

7. When the **Detailed Reports** dialog opens, the elements selected from the layout view are already highlighted for output. From this dialog, you can edit the element selection set (hold down the **Ctrl** key to select multiple elements from the list), and then print the reports. Click the **Select** button to go to the **Selection Set** dialog. If you wished to select multiple elements based on some criteria, you could do that here. Click **Cancel** to return to the **Detailed Reports** dialog. You can click the **Print** button to print all of the reports for the selected elements. Click **Cancel** to exit the dialog.

8. Another type of report available is the Element Results Report. The elements to be included in this report are selected the same way as the elements for the Detailed Report, except that **Elements Results** is selected from the **Report** pull-down menu. The result is a single report containing calculated analysis results for each of the elements selected. From the **Elements Results Report** dialog, you can print or copy/paste the report for any element(s). The Elements Results Report contains all of the results calculated for the selected element(s). Go ahead and select elements to be included in this report, and click **Preview** to view the resulting output. Click **Close** when you are finished to exit the preview and the Analysis Results Report dialog.

9. Select **Scenario Summary** from the **Reports** pull-down menu. This report summarizes the alternatives and options selected in the current scenario. Click **Close**.

10. Click the **Report** pull-down menu again and select the **Project Inventory** option. This report will tell you the total number of each type of element and the total length of pipe in the system. Click **Close**.

11. Finally, click the Cost Manager button  in the toolbar. Assuming Cost Estimating calculations had already been run, clicking on the Report button located above the **Scenario Costs** tree view would generate a Capital Cost Analysis Report. Click **Close**.

3.4.2 Part 2 – Tabular Reports (FlexTables)

Tabular Reports are an extremely powerful tool in WaterCAD. These reports are not only good presentation tools, they are also very helpful in data entry and analysis. When data must be entered for a large number of elements, clicking each element and entering the data can be tedious and time consuming. Using the tabular reports, elements can be changed using the global edit tool, or filtered to display only the desired elements. Values that are entered into the table will be automatically updated in the model. The tables can also be customized to contain only the desired data. Columns can be added or removed, or you can display duplicates of the same column with different units. The tabular reports can save you an enormous amount of time and effort.

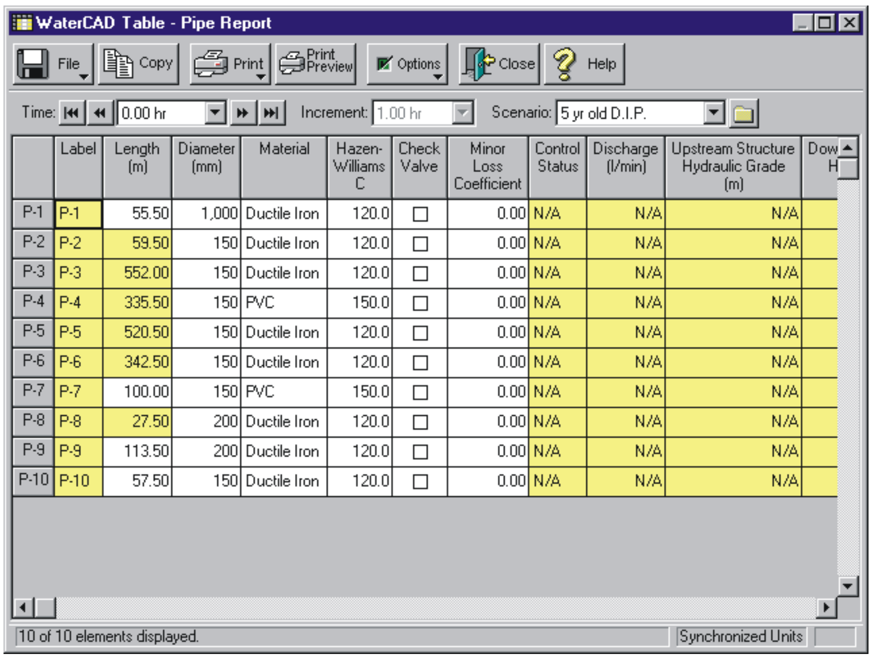
Tabular reports are dynamic tables of input values and calculated results. White columns are editable input values, and yellow columns are calculated values (not editable). These tables can be printed or copied into a spreadsheet program.

Two very powerful features in these tables are **Global Edit** and **Filtering**. Suppose we decide to evaluate how our network might operate in five years. Assume that the C factor for 5-year old ductile iron pipe reduces from 130 to 120. It would be repetitive to go through and edit the pipe roughness through the individual pipe dialogs, particularly when dealing with a large system. Instead, you will use the filter tool in this example to filter out the PVC pipes, and then use global edit tool to change the pipe roughness on the ductile iron pipes only.

1. Begin by setting up a new Alternative and Scenario to capture the changes to the C values. Select **Analysis\Scenarios** from the drop-down menus. Highlight the Scenario 'P-8 and P-9 Set to 200 mm' and click the **Scenario Management** button. Add a Child for this Scenario called '5-yr-old D.I.P.'

2. Because we want to create a new Physical Alternative, check the box labeled **Physical** and click the **New** button. Name the new Alternative '5-yr-old D.I.P.' and click **OK**. Click **Close** to exit the Alternative editor, and again to exit the Scenario Manager and return to the drawing pane.

3. To open a tabular report, select the **Tables** option from the **Report** pull-down menu or click the **Tabular Report** button on the toolbar. Select the **Pipe Report** from the list and click **OK**.



	Label	Length (m)	Diameter (mm)	Material	Hazen-Williams C	Check Valve	Minor Loss Coefficient	Control Status	Discharge (l/min)	Upstream Structure Hydraulic Grade (m)	Dow H
P-1	P-1	55.50	1,000	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-2	P-2	59.50	150	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-3	P-3	552.00	150	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-4	P-4	335.50	150	PVC	150.0	☐	0.00	N/A	N/A	N/A	
P-5	P-5	520.50	150	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-6	P-6	342.50	150	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-7	P-7	100.00	150	PVC	150.0	☐	0.00	N/A	N/A	N/A	
P-8	P-8	27.50	200	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-9	P-9	113.50	200	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	
P-10	P-10	57.50	150	Ductile Iron	120.0	☐	0.00	N/A	N/A	N/A	

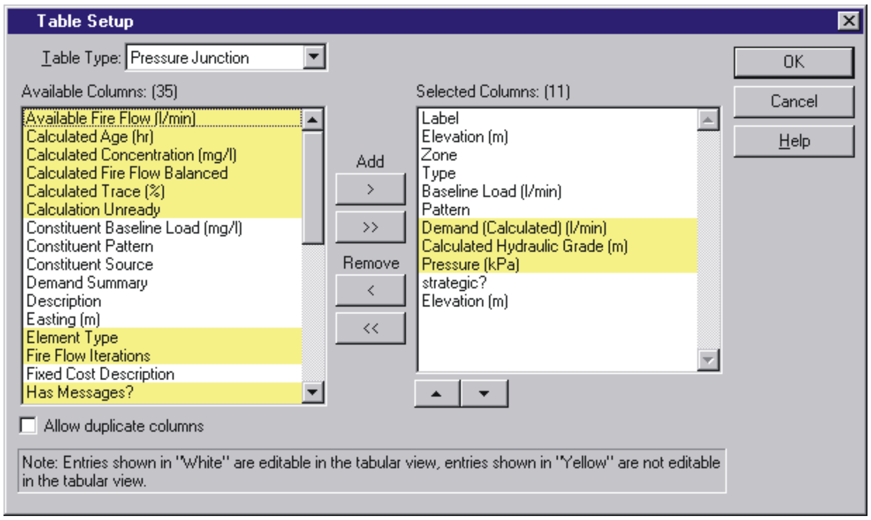
4. Click the **Scenario** list box and select the new Scenario from the list. Right-click on the **Material** column and choose **Filter\Quick Filter** from the pop-up menu. You want to display only ductile iron pipes in the table. To do so, set the **Column** field to **Material**, set the **Operator** to =, and set the **Value** field to **Ductile Iron**. Click **OK**.

5. Now, you will use the **Global Edit** tool to modify all of the roughness values in the table. Right-click on the **Hazen-Williams C** column and select **Global Edit**. Select **Set** from the **Operation** list and enter **120** into the **Global Edit** box. Click **OK**. All of the values are now set to 120.

6. To deactivate the filter, right-click anywhere in the dialog and select **Filter\Reset** from the pop-up menu. Click **Yes** to reset the filter.

7. You may also wish to edit a table by adding or removing columns using the **Table Manager**. Click the **Options** button at the top of the table dialog box and select **Table Manager**. Select the **Junction Report** from the list, then click the **Table Management** button. Notice in the Table Management pull-down menu there is a **New** option that would allow you to create your own table from scratch. Any tables you add will be saved for use with other projects. For now, click the **Edit** option to edit the existing Junction Report.

8. Scroll through the list on the left side to see the types of data available for placement in the table. You can highlight a particular item, then use the **[<]** and **[>]** buttons to add or remove that column from your table. For this example, we can display junction elevations in both meters and feet by checking the **Allow Duplicate Columns** box, highlighting **Elevation** (shown in gray) in the list of available columns, and clicking the **[>]** button. Elevation now appears twice under Selected Columns. You can adjust the order in which the columns will be displayed using the arrows below the right hand list, or simply drag items up and down. Click **OK**, and **OK** again to exit the Table Manager.



9. The revised table appears with two Elevation columns, both in meters. Drag and drop the new column heading to relocate it next to the original Elevation column. To apply separate units to the two columns, click the **Options** button and select **Use Local Units**. This option allows the tabular report to have units independent of the project and local to the table. In other words, without this option switched on, changing the units from meters to feet in the table would change the unit for pressure throughout the project. The **Use Local Units** option is ideal for displaying the same variable with multiple units within the same tabular report. Click **Yes** in the warning box that appears indicating that you wish to switch to local units. To assign units of feet to one of the columns, right-click on the column heading and select **Elevation Properties** from the pop-up menu. In the **Set Field Options** dialog, select **ft** from the **Units** list box. Click **OK** to return to the updated table.

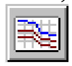
10. Click **Close** to exit the table when you are finished. Click **GO** to open the run dialog, and click **GO** to compute the network for '5-yr-old D.I.P.', and then click **Close**.

3.4.3 Part 3 – Create a Plan and Profile

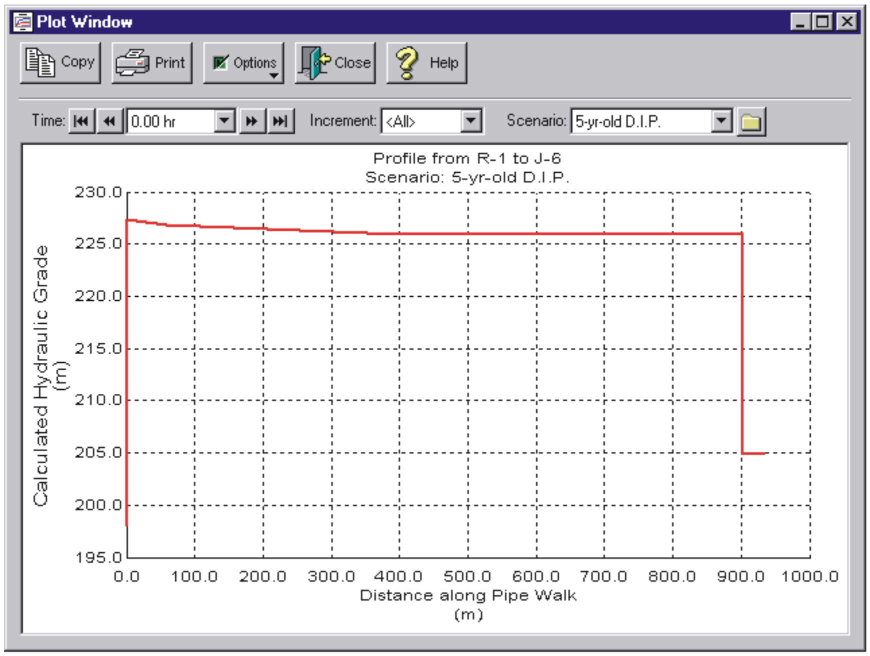
1. To create a plan view of the distribution system, click the **Report** pull-down menu and select **Plan View\Full View**. The **Full View** option will create a plan of the entire system regardless of what the screen shows, while the **Current View** option will create a plan of exactly what is displayed in the window at that moment.

2. The **Plan View** will be put into a separate window, which can then be printed or copied to the clipboard. If you click the **Copy** button, you can then paste the plan view into a word processor. Click **Close**.

3. You can also create a plan view in the Stand-Alone version for export to AutoCAD or other compatible software. Simply click on the **File** menu and select **Export\DXF File**. This action will create a DXF file of your network that you can then import into AutoCAD. In AutoCAD, use plan views as a quick way to develop simple scaled views of your primary network.

4. To create a profile view, select the **Profiling** option from the **Tools** pull-down menu, or click the **Profile** button  in the toolbar. The **Profile Setup** opens. From the dialog, choose the attribute you wish to profile from the **Attribute** list box. For this example, choose **Calculated Hydraulic Grade**.

5. Next, click the **Select From Drawing** button. The dialog disappears, and the crosshairs change to a pick box in AutoCAD. Click on pipes P-1, P-2, P-6, P-8 and P-9, selecting a continuous path, or "walk", through the network. Right-click when you are finished (and select **Done** in Stand-Alone). The **Profile Setup** dialog reappears, with the selected elements appearing, in order, in the list. Click the **Profile** button to view the profile.



6. Once the profile is created, you can make adjustments to its appearance, if desired, by clicking the **Options** button and selecting **Graph Options**. A dialog opens in which you can change the titles, fonts, scaling, and line types used in the graph. Leave everything set to the defaults for this example, and click **Cancel** to exit the dialog.

7. When you have finished setting up the graph, it can be printed or copied to the clipboard using the buttons at the top of the **Plot** window.

8. Click **Close** to exit the **Plot** window when you are finished, and close again to exit the **Profile Setup** dialog.

3.4.4 Part 4 – Contouring

The contouring feature in WaterCAD enables you to generate contours for reporting attributes such as elevation, pressure, and hydraulic grade. You can specify the contour interval, as well as color code the contours by index values or ranges of values. In this lesson, we will contour based on hydraulic grade elevations.

1. To create a plan view of the water distribution system with contours, click the Contour button from the toolbar. Within the Contour Manager, choose **Calculated Hydraulic Grade** from the **Contour** list box.

2. Create a Selection Set of elements you will use in contouring by clicking the ellipses button next to the **Selection Set** list box. Click **Add**, name the set ‘Contour by HGL’, and click **OK**.

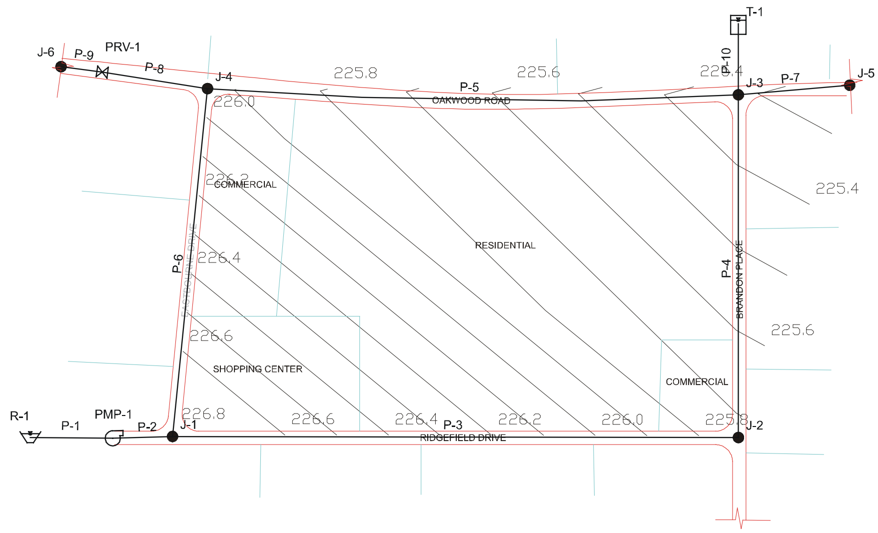
3. Define your selection set so that it consists of all junctions in Zone-1. Click **Select\By Filter\Pressure Junctions**. Under **Column**, select **Zone**; for **Operator**, select **[=]**; and for **Value**, select **Zone-1**. Click **OK** to return to the Selection Set dialog.

4. Notice that the selected elements are now highlighted under Available Items. Click **[>]** to move the elements to the Selected Items list, and click **OK**. Click **OK** again to exit the Selection Set Manager.

5. Select ‘Contour by HGL’ in the **Selection Set** list box, and click **Initialize** to update the Minimum and Maximum HGL elevations. Enter an **Increment** of 0.1 m, and an **Index Increment** of 0.2 m. Make sure the radio button for **Color by Index** is selected, and click **OK**.

6. A plan view of the water distribution model is opened in a separate window, along with contours that interpolate between the elevations of the selected network components.

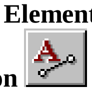
7. To improve the appearance of the contouring, press the **Options** button and choose **Smooth Contours**.



8. Click **Close** to return to the drawing pane.

3.4.5 Part 5 – Element Annotation

When you want to label network attributes in the plan view, use the Annotation feature. With it, you can control which values are displayed, how they are labeled, and how units are expressed. For this example, we will annotate demands at the junctions, and flow and velocity in the pipes.

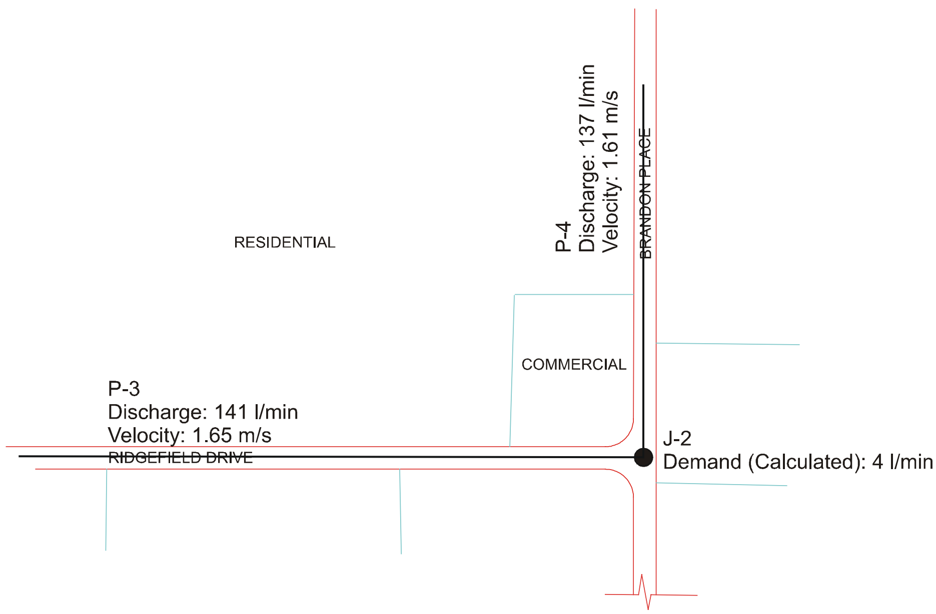
1. Click the **Element Annotation** option located in the **Tools** pull-down menu or click the **Annotation**  button on the toolbar to open the **Annotation Wizard**.

2. Select the elements you wish to annotate. In this example, we will add annotation to the pressure junctions and pipes. Select these elements from the list and click **Next**.

3. The next dialog allows you to choose the attributes you wish to annotate for the specified element type. The **Attributes** column is used for selecting the attribute you would like to annotate. The **Mask** is a template of how the annotation will appear on the screen. The %v and %u options are added to display and control the value and units associated with the attribute. The **Preview** column shows an example of the annotation with values for the variables. For this example, we will add annotation for the demand summary at each junction.

4. In the first row of the attribute column select **Demand (Calculated)** from the list and click **Next**. Add **Discharge** and **Velocity** annotations for the pipes in the same manner and click **Next**.

5. The final dialog of the **Annotation Wizard** contains a summary where you can check your annotations. If there are any errors, click the **Back** button to go backwards in the wizard and make any necessary changes. Click **Finished**.



6. The drawing will now display all of the annotations. You can try changing the properties of an element and recalculating. The annotations will update automatically to reflect any changes in the system.

7. If the annotation is crowded, you can click and drag the annotation to move it. In the AutoCAD version, click on the annotation and then click the grip to move it, or use an AutoCAD command such as Move or Stretch. Alternatively, you could go back into the Annotation Wizard, and abbreviate or remove the **Mask** labels. A third option is to decrease the **Annotation Height**. This option is found under the **Drawing** tab in the **Tools\Options** menu.

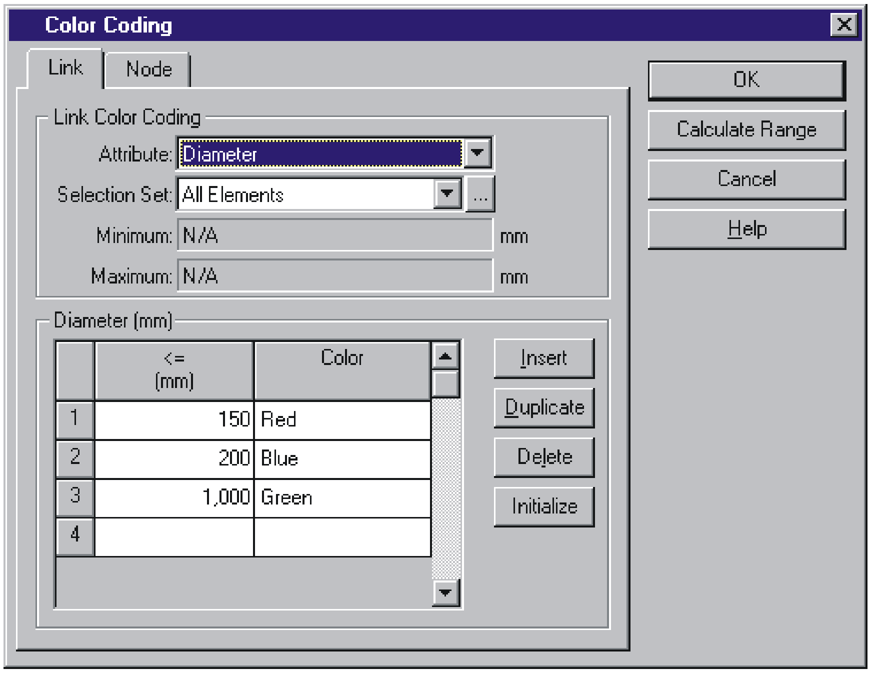
8. If you wish to delete the annotations, click the **Annotation** toolbar button, and uncheck any checked boxes. Click **Next**, and then **Finished**.

3.4.6 Part 6 – Color Coding

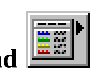
You can also review results in the plan view by color coding the elements based on attributes or ranges of values.

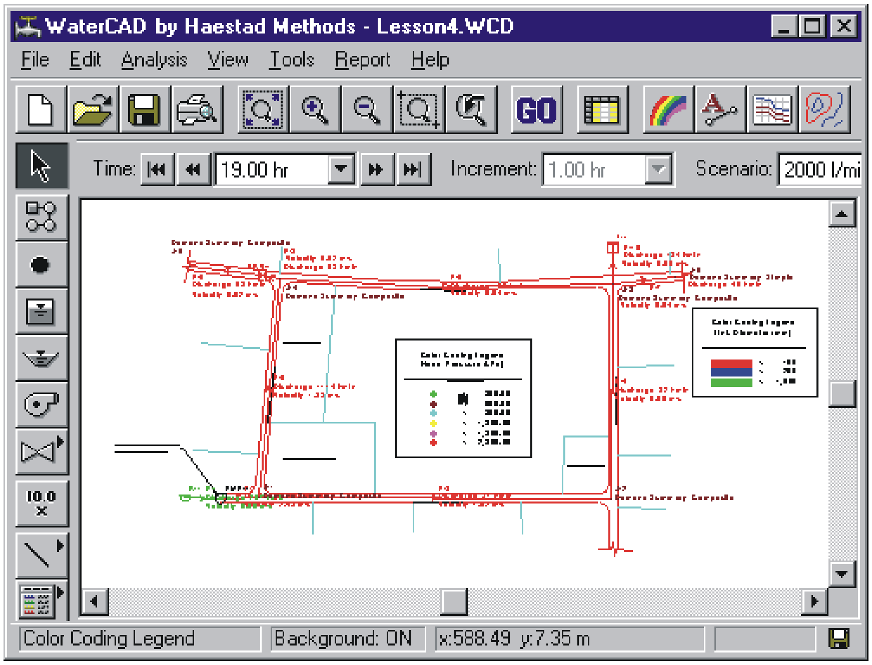
1. Select the **Color Coding** option under the **Tools** pull-down menu or click the **Color Coding** button  on the toolbar.

2. The **Color Coding** dialog allows you to set the color coding for links, nodes, or both. We will color code by diameter (link attribute) and pressure (node attribute) in this example. Click on the **Link** tab, and from the **Color Coding** list box, select the **Diameter** attribute. In the table, enter values of 150, 200 and 1000 mm with colors of red, blue, and green, respectively.



3. Next, click on the node tab, and select **Pressure** from the list box. Click the **Calculate Range** button to get the minimum and maximum values for the variable displayed at the top of the dialog. Then, click the **Initialize** button and the model will select the color coding ranges in the table automatically. Finally, click OK to generate the Color Coding.

4. We can add a legends to the drawing by clicking the **Insert Legend**  button, choosing the **Link Legend** or **Node Legend** option, and clicking anywhere on the drawing space to place the legend. In AutoCAD, accept the defaults when prompted about scale and legend.



5. Color coding is dynamic; that is, it updates automatically as you change time increment or Scenario using the list boxes above the drawing pane. Notice that as you switch from the first Scenario to 'P-8 and P-9 set to 200 mm', the color of these pipes changes from red to blue.

6. To turn the color coding off, simply open the Color Coding dialog and set the **Attribute** to **<None>** on both the Link and Node tabs. To delete a legend, select it by clicking once, and hit the **Delete** key.

Close the open dialogs and save this project before proceeding with Lesson 5.

From these four lessons, you have had a brief introduction to the capabilities of WaterCAD. Feel free to continue to play with the program. Use this model to explore and become familiar with all of the features. If you do not know what a button does, just try it.

3.5 Lesson 5 – Automated Fire Flow Analysis

One of the primary goals of a water distribution system is to provide adequate capacity to fight fires. WaterCAD's automated fire flow analysis can be used to determine if the system can meet the fire flow demands while maintaining minimum pressure constraints. Fire flows can be computed for all nodes in the system, or you can create a selection set consisting of specific nodes where you wish to test available flow.

Fire flows are computed by iteratively assigning demands and computing system pressures. The model assigns the fire flow demand to a node and checks the model, checking to see if all pressure constraints are met at that demand. It will assign a new demand and automatically re-check the system pressures if a constraint is violated. Iterations continue until the constraints are met, or until the maximum number of iterations is reached.

The purpose of this example is to walk you through the steps to create, calculate, and analyze a fire flow scenario. This lesson again uses the distribution system from the previous lessons.

3.5.1 Part 1 – Inputting Fire Flow Data

1. Start WaterCAD and open the file 'Lesson5', found in the \Haestad\Wtrc\Lesson folder. Or, if you have previously completed Lesson 3 or 4, you can use your 'MyLesson3' or 'MyLesson4' file. Select **Project Summary** from the **File** menu, and change the title of the project to 'Lesson 5—Fire Flow Analysis'. Click **OK**.

2. Previously, we ran an analysis with a fire flow at node J-6 by manually adding a large demand to the individual node. Before running the automated fire flow analysis, we will create a new Demand Alternative, removing that demand. Select **Analysis\Alternatives** from the pull-down menus and click the **Demand** tab. Highlight 'Average Daily with 2000 l/min Fire Flow', and click **Duplicate**.

3. In the **Demand Summary** column for node J-6 (row 6), click the word 'Composite', and it changes to a button. Click the **Composite** button to open the editor for the composite demand. Click the 2000 l/min demand, and click **Delete**. Select **Yes** to confirm, **OK** to close the editor, and **Close** to exit the Demand Alternative.

4. You will now see the Demand Alternative 'Copy of Average Daily with 2000 l/min Fire Flow' in the list. Highlight it and click the **Rename** button. Rename this Alternative 'Base-Average Daily'.

5. You are going to analyze the fire flows by adding to the Maximum Day Demands, which are 1.5 times the Average Day Demands. Set up the Maximum Day Demand Alternative by highlighting the Base – Average Daily Alternative and selecting **Duplicate**. Right click the **Demand** column, and select **Global Edit**. Set the **Operation** to multiply, and enter a value of **1.5**. Click **OK**.

6. Click **Close** to exit the new Demand Alternative, and **Rename** this Alternative 'Max. Day'. Then, click the **Fire Flow** tab in the Alternative Manager.

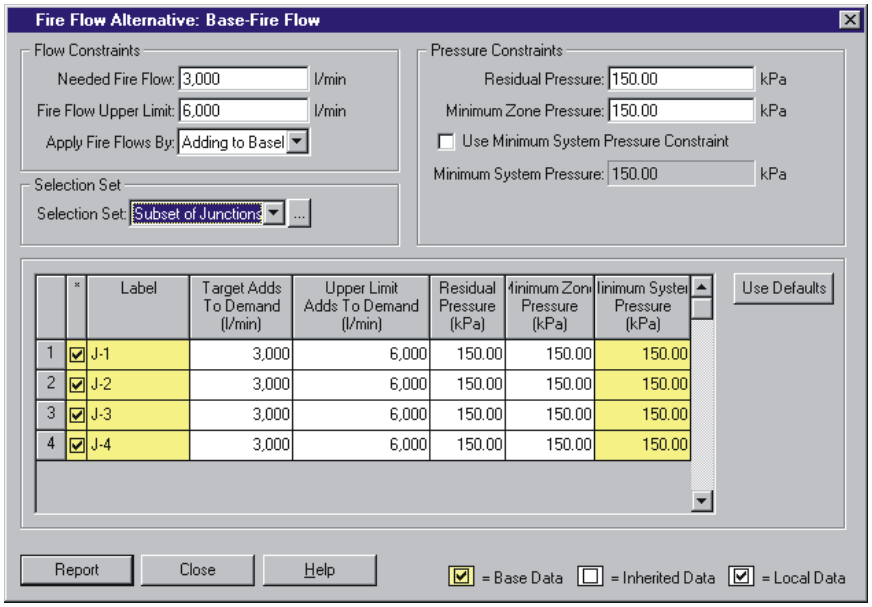
7. Click the **Edit** button to set up the Base-Fire Flow Alternative. Enter a **Needed Fire Flow** of 3,000 l/min, a **Fire Flow Upper Limit** of 6,000 l/min, and **Apply Fire Flows By Adding to Baseline Demand**. This selection means that in performing the analysis, the fire flow will be added to any demands already assigned to the junction. Alternatively, you could have selected to replace these demands, so that the fire flow would represent the total demand at the node.

8. For Pressure Constraints, set both the **Residual Pressure** and **Minimum Zone Pressure** to 150 kPa. Leave the box for **Use Minimum System Pressure Constraint** unchecked, so that the minimum pressure will only be checked for the zone a particular node is in. Note that if you had multiple zones within your project and wanted to insure that a minimum system-wide pressure constraint was met, you could check the **Use Minimum System Pressure Constraint** box and type it in the box provided. This box is grayed out until the check box is activated.

9. Using the Selection Set list box, you can choose to apply the fire flow to **All Junctions** or a **Subset of Junctions**. For this example, choose the **Subset of Junctions** option. Then, click the ellipses **(…)** button to open the Selection Set dialog.

10. For this example, a fire flow analysis is only needed for the junctions at the four street corners in our drawing. Choose the **Select From Drawing** button and click on nodes J-1, J-2, J-3, and J-4 (in Stand-Alone, you must hold down the **Shift** key while making multiple selections). When you are finished, right-click (and select **Done** in Stand-Alone).

11. The nodes you selected are now in the **Selected Items** list. Click **OK** to exit the **Selection Set** dialog. Note that the Selection Set table is now filled with your Selection Set. Click the **Close** button to exit the **Fire Flow Alternative** dialog, and **Close** again to exit the **Alternative Manager** dialog.



3.5.2 Part 2 – Calculating a Fire Flow Analysis

1. You must now set up a new Scenario to utilize the appropriate alternatives and set up the calculation options. Select **Analysis\Scenarios** to open the Scenario Manager, and choose **Scenario Management\Add\Base Scenario**. Name your new Scenario 'Automated Fire Flow Analysis' and click **OK**.

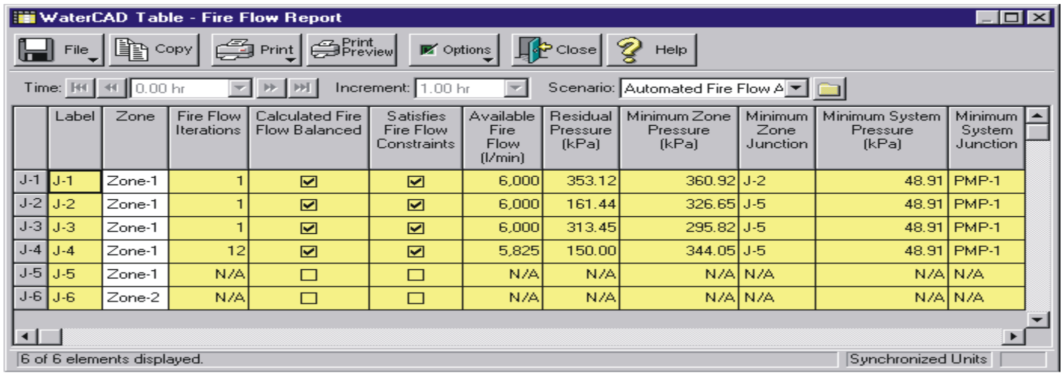
2. In the **Alternatives** tab of the Scenario dialog, change the **Physical** Alternative to 'P-8 and P-9 Set to 200 mm'. Change the **Demand** to 'Max. Day' and leave all other Alternatives set to their defaults.

3. Click the **Calculation** tab, make sure that **Steady State** is selected, and check the **Fire Flow Analysis** box. Note that all fire flow calculations must be performed under steady-state conditions. Click **Close** to exit the Scenario dialog.

4. To run the Scenario, click **GO Batch Run**. Check the box for 'Automated Fire Flow Analysis', and uncheck the other Scenarios. Click **Batch** to run the analysis, and **Yes** at the confirmation prompt. When the calculation is complete, click **OK** and **Close** to exit the **Scenario Manager**.

3.5.3 Part 3 – Viewing Fire Flow Results

1. Click the Tabular Reports button, located to the right of the **GO** button on the WaterCAD toolbar, to open the **Table Manager**. Highlight the **Fire Flow Report** and click the **OK** button to view the report. Make sure that 'Automated Fire Flow Analysis' is selected in the **Scenario** list box.



	Label	Zone	Fire Flow Iterations	Calculated Fire Flow Balanced	Satisfies Fire Flow Constraints	Available Fire Flow (l/min)	Residual Pressure (kPa)	Minimum Zone Pressure (kPa)	Minimum Zone Junction	Minimum System Pressure (kPa)	Minimum System Junction
J-1	J-1	Zone-1	1	☑	☑	6,000	353.12	360.92	J-2	48.91	PMP-1
J-2	J-2	Zone-1	1	☑	☑	6,000	161.44	326.65	J-5	48.91	PMP-1
J-3	J-3	Zone-1	1	☑	☑	6,000	313.45	295.82	J-5	48.91	PMP-1
J-4	J-4	Zone-1	12	☑	☑	5,825	150.00	344.05	J-5	48.91	PMP-1
J-5	J-5	Zone-1	N/A	☐	☐	N/A	N/A	N/A	N/A	N/A	N/A
J-6	J-6	Zone-2	N/A	☐	☐	N/A	N/A	N/A	N/A	N/A	N/A

2. In the **Satisfies Fire Flow Constraints** column, all of the boxes are checked, except for the nodes that you did not analyze, because the specified needed flow of 3,000 l/min was available and minimum pressures were exceeded. For nodes J-1 and J-3, pressures were computed for the Fire Flow Upper Limit of 6,000 l/min, because none ever dropped below specified minimum pressures. Nodes J-2 and J-4 reached their minimum residual pressures at flows slightly below the maximum of 6,000 l/min. Notice also that the report contains the Minimum System Pressure (excluding the current node being flowed) and its location.

3. When you are finished reviewing the report, click **Close** in the WaterCAD **Fire Flow Report** dialog and save your file as Mylesson5.

Another good way to review an automated fire flow analysis is to use color coding. If you have a situation where all nodes do not meet the pressure constraints for the needed fire flow, you can color code these nodes in the plan view for easy identification. See Lesson 4, Part 6 in this chapter for more information on color coding.

3.6 Lesson 6 – Water Quality Analysis

In conjunction with Extended Period simulations, WaterCAD is capable of performing a water quality analysis to compute water age, constituent concentration, or percentage of water from a given node (trace analysis). Using these features, you can look at factors such as residence time in tanks, chlorine residuals throughout the system, and which tank or reservoir is the primary water source for different areas in your system.

This lesson uses the file called 'Lesson6.wcd' ('Lesson6.dwg' in the AutoCAD version), located in the \Haestad\Wtrc\Lesson directory. Open this file, and then select **File\Save As** from the pull-down menu. Type the filename 'MyLesson6' and click **Save**.

The water distribution system has already been set up for you. It has one reservoir and one tank. The system serves primarily residential areas, with some commercial water use as well. There are two pumps connected to the reservoir; however, under normal conditions, only one pump will be in use. A background drawing has been included for reference. If you would like to turn off the dxf background in the Stand-Alone version, select **Tools\Options** from the menus, click the **Drawing** tab, and uncheck the **Show Background** box. In the AutoCAD version, you can simply turn off the desired layers.

If at any time during this lesson the program asks, "Do you wish to reset all calculated results to N/A?" click NO.

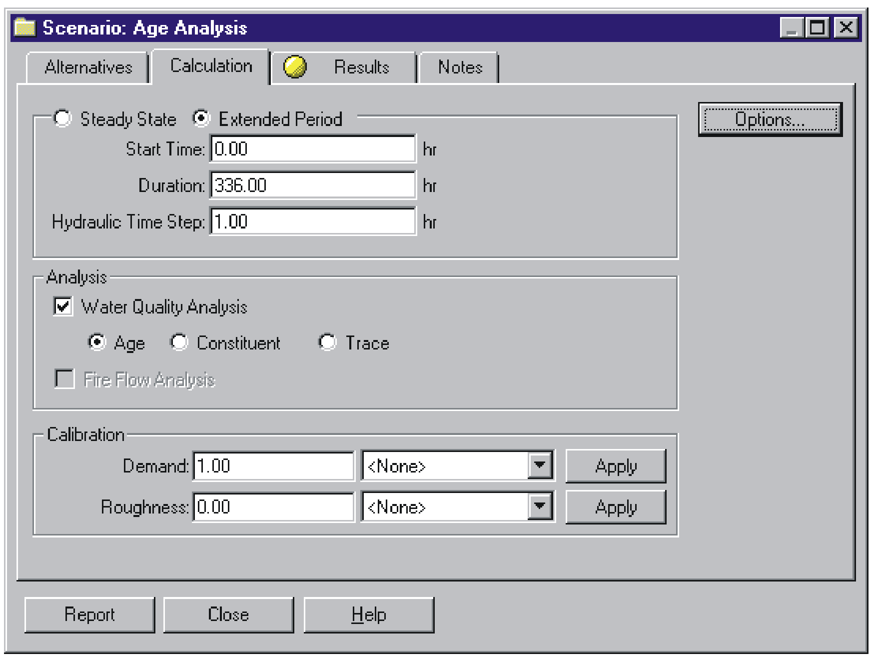
3.6.1 Part 1 – Computing Water Age

You will begin by running an age analysis for water in the system, assuming an initial age of 0 for all nodes. The water from the reservoir will be an infinite supply of new water, so the age of water elsewhere in the system will be a reflection of time from the start of the run and how long ago the water left the reservoir. The analysis will be run for a 2-week period (336 hours), in order to determine the equilibrium point of the system.

1. Start by adding a new Alternative for the age analysis. Select **Analysis\Alternatives** from the pull-down menus, and click the **Age** tab to access the Age Alternatives. Click the **Add** button to add a new Alternative, and name it 'Initial Age = 0'. Since you are assuming an initial age of 0 everywhere in the system, you do not need to enter any initial ages. Close out of the table and the Alternative Manager.

2. Next, set up a new Scenario to run an Extended Period Simulation incorporating the new Alternative. Select **Analysis\Scenarios** from the pull-down menus. You will see that there is one Scenario already set up called 'Existing – Avg Day'. Highlight this Scenario and click **Scenario Management\Add\Child Scenario**. Type in 'Age Analysis' as the new Scenario name, and click **OK**. Under the Alternatives tab, click to check the box labeled **Age**, and select the Alternative you just created, 'Initial Age = 0', from the list box.

3. Click the **Calculation** tab to view the calculation settings for this Scenario. **Extended Period Analysis** should already be selected. Enter a Start Time of 0, Duration of 336 hours, and Hydraulic Time Step of 1 hour. Check the **Water Quality Analysis** box, and select the **Age** radio button.



4. Click the **Options** button. In this dialog, you will find options for Maximum Pipe Segments (the largest number of pipe segments a pipe can be broken into during a water quality analysis), Minimum Pipe Travel Time, and a place to enter a user-defined Water Quality Time Step. You will leave these options set to their default values, so click **Cancel** to exit the dialog box, and **Close** to exit the **Scenario Editor**.

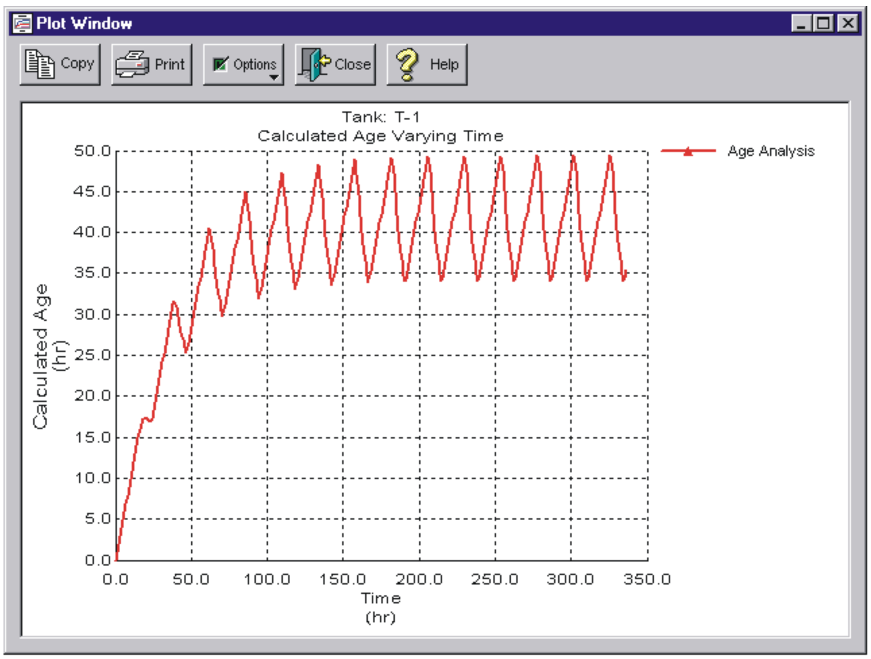
5. You are now ready to run the Scenario. Click the **Batch Run** button and check the box for 'Age Analysis'. Click **Batch**, and then **Yes** at the verification prompt. When the run is complete, click **Close** to exit the **Scenario Manager**. Select 'Age Analysis' from the **Scenario** toolbar list box.

6. A good way to view changes in water quality attributes over time is to use color coding. Click the **Color Coding** button on the toolbar. Under the **Link** tab, set the **Attribute** list box to **Calculated Age**, and then click the **Initialize** button to set up a default color scheme. Accept this default scheme, and repeat this procedure for the **Node** tab. When you are finished, click **OK**.

7. Once back in the drawing pane, click the **Legend** button on the toolbar, and select either the **Node Legend** or **Link Legend** (in this case, the color coding should be the same for both). Click in your drawing to place the legend. If you don't place it correctly the first time, you can always drag it and drop it in a new location.

8. Set the time increment in the toolbar to 4 hours. Use the VCR-style buttons to scroll through time in your analysis, and observe the color changes in your network reflecting water age.

9. A good way to check if your network has had sufficient time to reach an equilibrium point is to look at Age vs. Time graphs for your elements. Open the editor dialog for Tank T-1, click the **Report** button at the bottom of the editor, and select **Graph**. Set up the Independent Variable as Time, and the Dependent Variable as Calculated Age. Make sure 'Age Analysis' is checked in the **Available Scenarios** box and click **OK**.



10. From the graph, you can see that once a repeating pattern is reached, the age of the water fluctuates between approximately 34 and 49 hours in 24-hour periods. Looking at these equilibrium ranges for various nodes can help guide you in setting up initial water age values in subsequent runs.

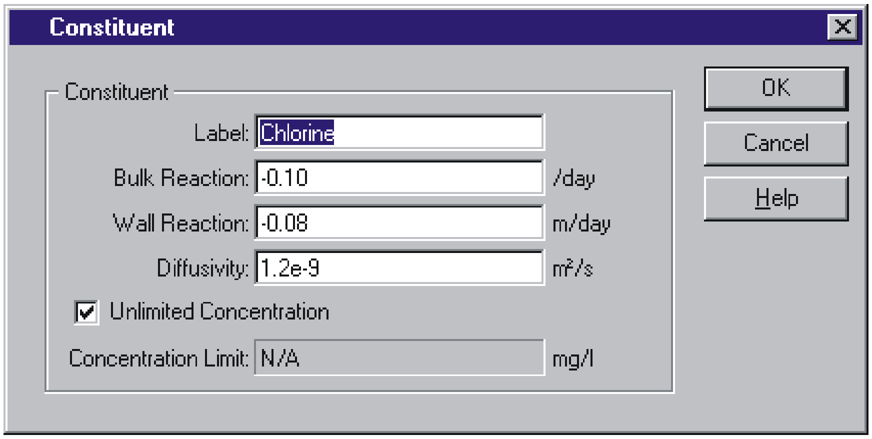
3.6.2 Part 2 – Analyzing Constituent Concentrations

In this portion of the lesson, you will look at chlorine residuals in the system over time. WaterCAD stores information on constituent characteristics in a file called a constituent library. You will add information for chlorine to this library, set up initial concentrations in the system, and run the simulation.

1. Select **Analysis\Alternatives** from the pull-down menus. Select the **Constituent** tab, and click the **Add** button. Name the new Alternative 'Chlorine Injection' and click **OK**.

2. Click the **ellipses (…)** button next to the **Constituent** list box to open the **Constituent Library**, and click the **Insert** button. An entry for ‘Unlabeled’ appears in the table. Click the **Edit** button, and enter the data below into the dialog.

Label:	Chlorine
Bulk Reaction:	-0.10/day
Wall Reaction:	-0.08 m/day
Diffusivity:	1.2e-9 m^2/s



3. Leave the **Unlimited Concentration** box checked, click **OK**, and then click **Close** to exit the Constituent Library. You should now be back in the editor for the Constituent Alternative.

4. Select **Chlorine** from the **Constituent** list box. Notice that the Bulk Reaction in the table is automatically updated. For the **Pump** and **Valve**, set the pumps and valves to an initial concentration of 1 mg/l. Click the **Junction** tab, and initialize the chlorine concentrations by entering a value of 1 mg/l at each junction node. Under the **Reservoir** tab, enter a value of 2.0 mg/L for the reservoir, and for the **Tank**, enter a value of 0.5 mg/l.

To quickly enter the initial concentrations for an element type, use the Global Edit feature.

5. Close out of the Editor and the Alternative Manager. Now, open the Scenario Manager and set up a new Scenario in order to run the Constituent Analysis. Create a new Child off of the ‘Age Analysis’ Scenario by highlighting it and clicking **Scenario Management\Add\Child Scenario**. Type in ‘Chlorine Analysis’ as the new Scenario name, and click **OK**. Under the **Alternatives** tab, click to check the box labeled **Constituent**, and select the ‘Chlorine Injection’ Alternative from the list box.

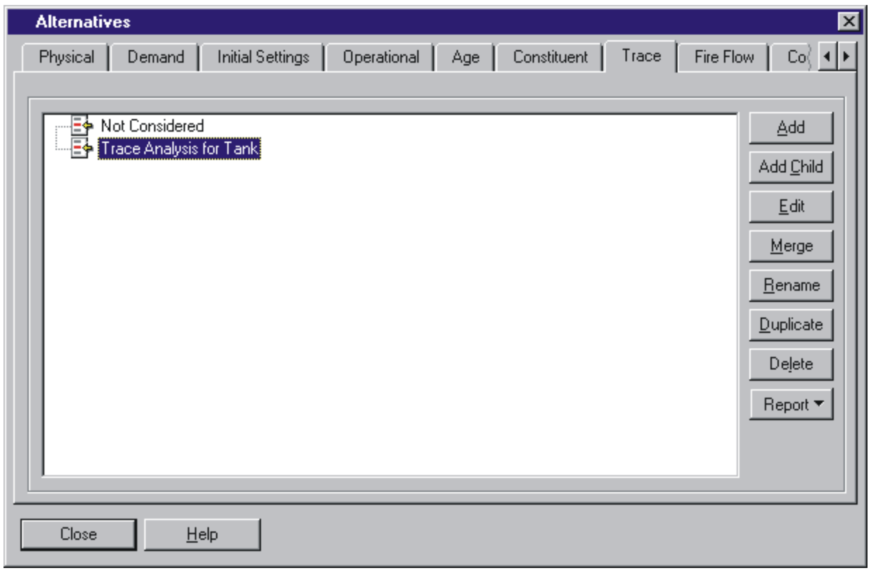
6. Click the **Calculation** tab, select the **Constituent** radio button in the **Analysis** section, and leave everything else set to the inherited values. Click **Close** to exit the dialog, click **GO Batch Run**, uncheck 'Age Analysis', check 'Chlorine Analysis', then click **Batch** to run the model.

7. Make sure ‘Chlorine Analysis’ is visible as the current Scenario and set up color coding using the procedure from Part 1, but color code by Calculated Concentration instead of Calculated Age. Scroll through the time steps to view how the concentrations change throughout the network. When you look at your results using color coding, tables and graphs, try to discover what better initial values for chlorine concentration might be.

3.6.3 Part 3 – Performing a Trace Analysis

A trace analysis determines the percentage of water at all nodes and links in the system from a specific source node (the trace node). In systems with more than one source, it is common to perform multiple trace analyses using the various source nodes as the trace nodes in successive analyses. For this run, we will perform a trace analysis to determine the percentages of water coming from the tank.

1. Select **Analysis\Alternatives** from the pull-down menus. Select the **Trace** tab, and click the **Add** button. Name the new Alternative 'Trace Analysis for Tank' and click **OK**. In the **Trace Node** list box, select the tank, **T-1**. Leave the initial percentages set to zero, and close the editor and the Alternative Manager.



2. Next, set up a new Scenario to run an Extended Period Simulation incorporating the new Alternative. Select **Analysis\Scenarios** from the pull-down menus. Create a new Child off of the 'Age Analysis' Scenario by highlighting it and clicking **Scenario Management\Add\Child Scenario**. Type in 'Trace Analysis' as the new Scenario name, and click **OK**. Under the Alternatives tab, click to check the box labeled **Trace**, and select the 'Trace Analysis for Tank' Alternative from the list box.

3. In the calculation tab, select the **Trace** radio button in the Analysis section, and leave everything else set to the inherited values. Click **Close** to exit the dialog, click **GO Batch Run**, and run the analysis for the new Scenario.

4. Use color coding (by Calculated Trace), tables, and graphs to view the results of this run. As you scroll through the time periods, notice how the colors spread outward from the tank during periods when the tank is draining, and recede when the tank begins to fill. For more information on reporting features, see Lesson 4.

Close the open dialogs and save this project before proceeding with Lesson 7.

3.7 Lesson 7 – Data from External Sources

WaterCAD supports several methods of exchanging data with external applications, preventing duplication of effort and allowing you to save time by reusing data already present in other locations. For instance, you can exchange data with databases or a GIS system, or you can convert existing CAD linework to a pipe network.

There are multiple ways of importing data from outside sources into WaterCAD. You can set up one or more database connections to bring in information stored in many standard database and spreadsheet formats. GIS information can be brought in through connections to ESRI Shapefiles. If you have existing drawings of your network in a DXF format (DWG format in the AutoCAD version), you can have WaterCAD convert your lines and/or blocks into distribution system elements, setting up preferences for handling situations such as T-intersections and line endpoints, and creating tolerances to allow for drawing imperfections. Or, you can display a DXF file as a background drawing for use in laying out a scaled network (stand-alone version only). Finally, WaterCAD will automatically import networks created in EPANet, KYPipe, and previous versions of Cybernet/WaterCAD.

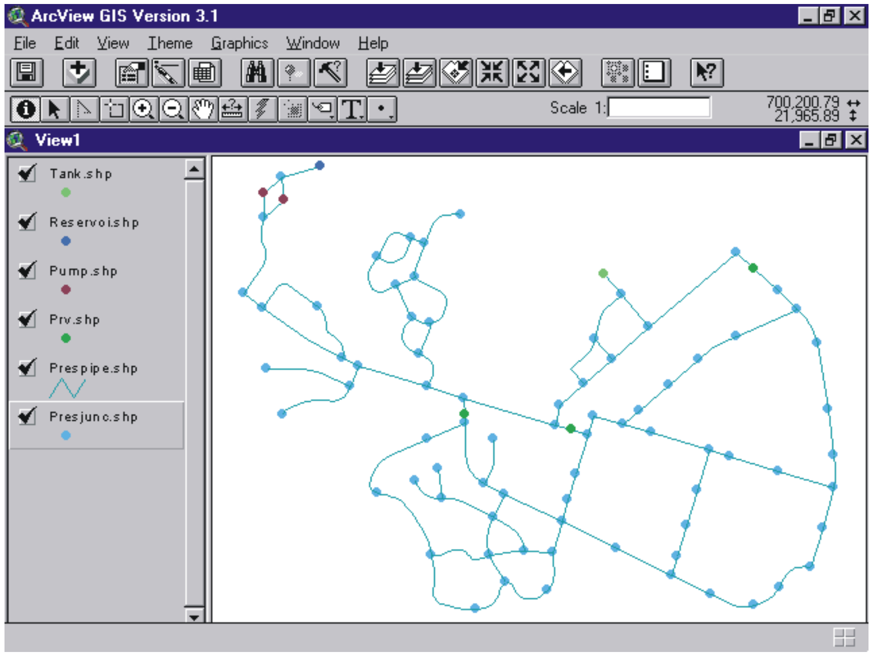
WaterCAD again utilizes database and Shapefile connections to export data from the model for use externally. You can also copy tables, reports, and graphs and paste them into other Windows applications, or save plan and profile views in DXF format for use when creating construction documents in CAD.

This lesson covers the three main methods of building your network using external data, summarized in the following table.

Method	Description	Advantages	Disadvantages
Database Connection	Create connections to import and export model data using common database and spreadsheet formats.	Extremely versatile. Allows exchange of most any model data with a wide variety of applications, including anything with an ODBC format. A topographic representation of the network can be created by using node coordinates and assigning "to" and "from" nodes to pipes. Once a connection is established, it can be saved for later use, and multiple connections can be created and synchronized simultaneously.	Pipes will be depicted as straight lines connecting the "to" and "from" nodes, so pipe bends will not be displayed.
Shapefile Connection	Create connections to import and export model data in ESRI Shapefile format.	Advantages are similar to those of Database Connections, except the topographic data exchange is automatic and pipe bends are accounted for.	More proprietary. You have to have software that supports ESRI Shapefiles in order to utilize the data.
Polyline to Pipe Conversion	Convert existing lines, polylines, and blocks in DXF/DWG format into pipes and other network elements.	Enables you to use legacy CAD drawings to build your network. You can set up tolerances to allow for drawing imperfections, and preferences for how nodes will be created.	Elements are assigned default labels as they are created. Only topographic data can be brought in, not model values. Requires user to carefully review the drawing for accuracy.

3.7.1 Part 1 – Importing Shapefile Data

In this part of the lesson, you will import ESRI Shapefiles to construct the distribution network in WaterCAD from existing GIS data. If you have ArcView, ArcInfo, or other application that can open a Shapefile, then you can, if you choose, view the files externally prior to importing them. However, you will still be able to perform the workshop problem even if you don't have one of these applications. This lesson uses the network from Lesson 6.



The ESRI Shapefile actually consists of three separate files that combine to define the spatial and non-spatial attributes of a map feature. The three required files are as follows:

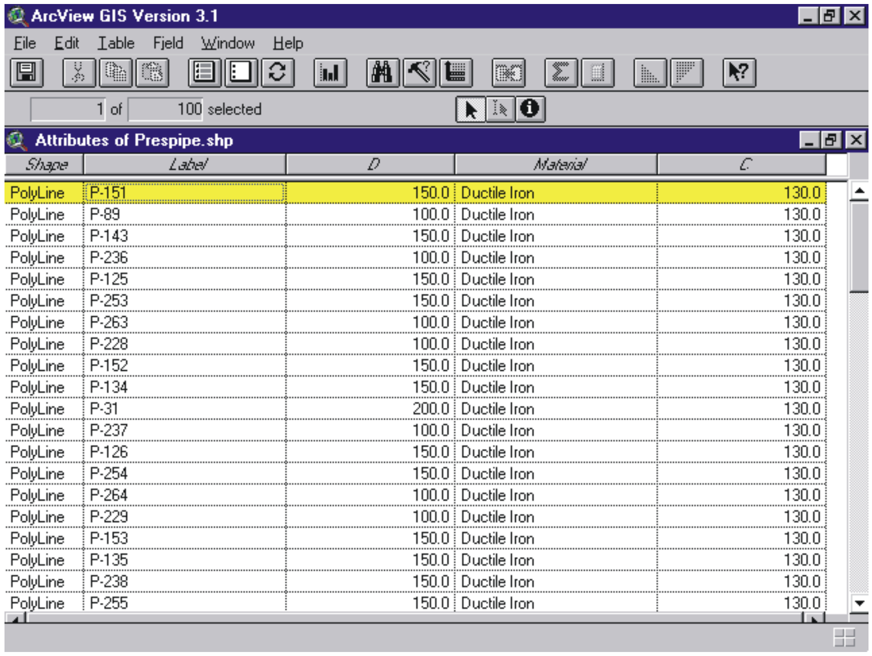
- **Main File** – The main file is a binary file with an extension of SHP. It contains the spatial attributes associated with the map features. For example, a polyline record contains a series of points, and a point record contains x and y coordinates.
- **Index File** – The index file is a binary file with an extension of SHX. It contains the byte position of each record in the main file.
- **Database File** – The database file is a dBase III file with an extension of DBF. It contains the non-spatial data associated with the map features.

All three files must have the same file name with the exception of the extension, and be located in the same directory.

Listed below are the files you will be importing. Only the main files are listed; however, corresponding SHX and DBF are present as well.

- PresJunc.shp
- PresPipe.shp
- PRV.shp
- Pump.shp
- Reservoi.shp
- Tank.shp

If you have a program such as ArcView that allows you to view Shapefiles, begin by setting up a View with all of the Shapefiles (Themes) listed above turned on. If you completed Lesson 6, you should recognize the layout from that lesson. You can look at the data table for each of the Themes to see what we will be importing. When you have finished reviewing the Shapefiles, close the application.



Shape	Label	D	Material	C
PolyLine	P-151	150.0	Ductile Iron	130.0
PolyLine	P-89	100.0	Ductile Iron	130.0
PolyLine	P-143	150.0	Ductile Iron	130.0
PolyLine	P-236	100.0	Ductile Iron	130.0
PolyLine	P-125	150.0	Ductile Iron	130.0
PolyLine	P-253	150.0	Ductile Iron	130.0
PolyLine	P-263	100.0	Ductile Iron	130.0
PolyLine	P-228	100.0	Ductile Iron	130.0
PolyLine	P-152	150.0	Ductile Iron	130.0
PolyLine	P-134	150.0	Ductile Iron	130.0
PolyLine	P-31	200.0	Ductile Iron	130.0
PolyLine	P-237	100.0	Ductile Iron	130.0
PolyLine	P-126	150.0	Ductile Iron	130.0
PolyLine	P-254	150.0	Ductile Iron	130.0
PolyLine	P-264	100.0	Ductile Iron	130.0
PolyLine	P-229	100.0	Ductile Iron	130.0
PolyLine	P-153	150.0	Ductile Iron	130.0
PolyLine	P-135	150.0	Ductile Iron	130.0
PolyLine	P-238	150.0	Ductile Iron	130.0
PolyLine	P-255	150.0	Ductile Iron	130.0

1. Double-click the WaterCAD desktop icon to start WaterCAD Stand-Alone. If the **Welcome to WaterCAD** dialog appears, select the **Close** button.

2. Open the **Global Options** tab, accessed from the **Tools\Options** pull-down menu. Since we will be working in metric units, click the **Unit System** selection box, and select **System International**. Click **OK**.

3. In the pull-down menus, select **File\New**. Click **No** when asked if you want to save the current project. In the **Create Project File As** dialog, double-click the 'Lesson' folder, enter the file name 'gisprob.wcd' for your project, and click **Save**. The **Project Setup Wizard** will open.

4. In the Project Setup Wizard, title the project 'Lesson 7, Part 1 – Importing GIS Data'. Click **Next**. Click the **Next** button again to leave this dialog set to its default values. In this dialog, set up the drawing as **Scaled**, with a horizontal scale of 1:**5000** and a vertical scale of 1:**500**. Change the three **Annotation Multipliers** (**Symbol Size**, **Text Height** and **Annotation Height**) to **2.8**. Click **Next**, leave the Prototypes set to their default values, and click **Finished**.

1. Double-click the desktop icon to start WaterCAD for AutoCAD. Open the **Global** tab, accessed from the **Tools\Options** pull-down menu. Since we will be working in metric units, click the **Unit System** selection box, and select **System International**. Click **OK**.

2. Choose **New** on the **File** pull-down menu and select **No** when prompted to save the existing drawing. If the Create New Drawing dialog opens, move to step 4; otherwise, do the following:

3. Click the **Esc** key. Then, type **filedia** at the command prompt and press **Enter**. Type the value **1** and press **Enter**. Then, choose **New** on the **File** pull-down menu again, and don't save changes to the existing drawing.

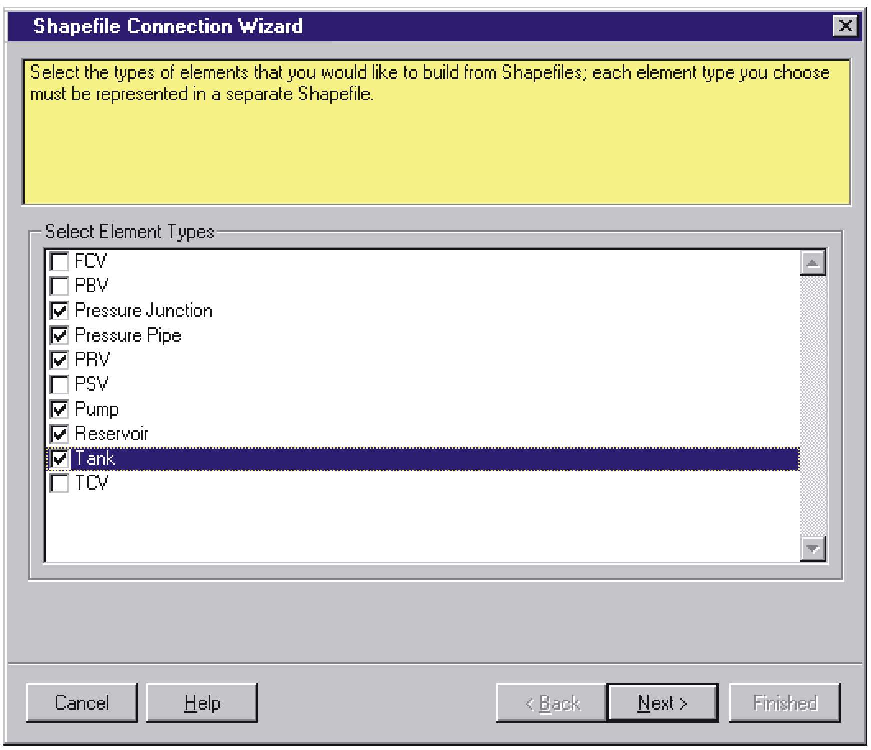
4. When the Create New Drawing dialog appears, make sure that **Metric** is selected, and click **OK**. Answer **Yes** when asked if you want to set up the project. In the Project Setup Wizard, title the project ‘Lesson 7, Part 1 – Importing GIS Data’ and click **Next**. Click **Next** again to accept the defaults on the second screen.

5. In this dialog, set up the drawing as **Scaled**, with a horizontal scale of 1:**5000** and a vertical scale of 1:**500**. Change the three **Annotation Multipliers** (**Symbol Size**, **Text Height** and **Annotation Height**) to **2.8**. Click **Next**, leave the Prototypes set to their default values, and click **Finished**.

The remaining commands are for both Stand-Alone and AutoCAD modes.

1. Select **File\Synchronize\Shapefile Connections** from the pull-down menus. If you have not defined any Shapefile connections in WaterCAD yet, you will be asked if you want to create a Shapefile connection; answer **Yes** to start the **Shapefile Connection Wizard**. Or, if you have already defined Shapefile Connections in any other WaterCAD project, start the **Shapefile Connection Wizard** by clicking **Add** in the Shapefile Connection Manager that appears. Enter the **Connection Label** ‘Lesson 7, Part 1’ for this connection, and click the **Next** button.

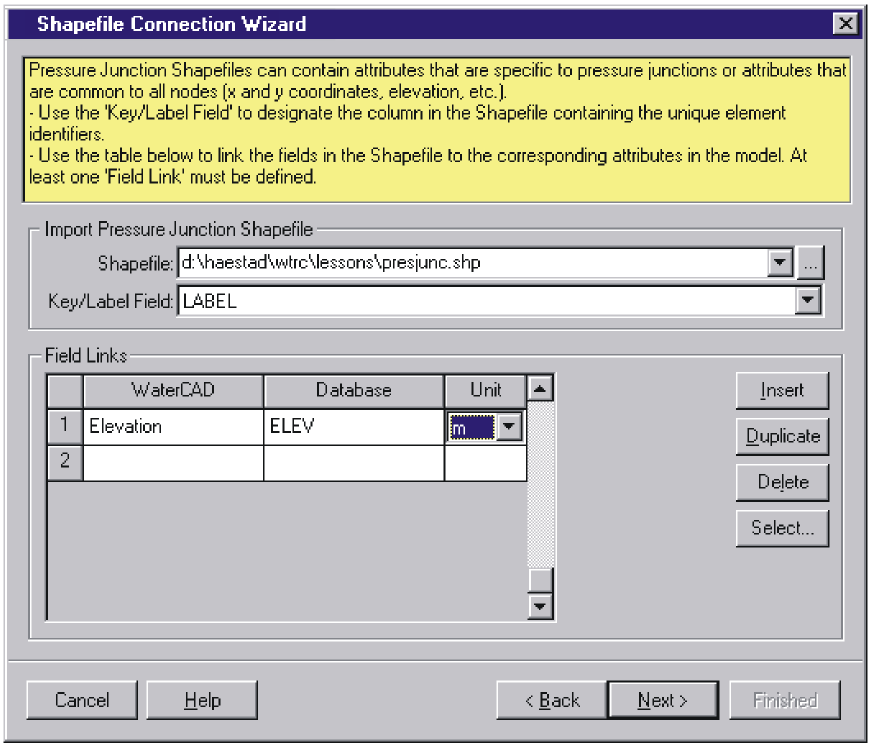
2. Now, you need to check the boxes for the types of elements you will be importing. For this connections, check the boxes for **Pressure Junction**, **Pressure Pipe**, **PRV**, **Pump**, **Reservoir**, and **Tank**. Click **Next**.



3. Leave the **Shapefile Unit** set to **m**, and check the box to establish missing connectivity data from spatial data, and then click **Next** again.

4. Click the **ellipses (…)** button next to the **Shapefile** field. Browse and select the file ‘PresJunc.shp’ from the \Wtrc\Lesson directory, then click **Open**. Set the **Key/Label Field** to **LABEL**. This item designates the field that WaterCAD matches with its own element labels, so that data will be assigned to the correct place.

5. Using the **Field Links** table, you must now match the data types available in WaterCAD to the data types you will be bringing in from the Shapefile. In row 1, select **Elevation** from the **WaterCAD** column, and **ELEV** from the **Database** column. Set the **Unit** to **m** to let WaterCAD know that the coordinate it is reading from the Shapefile is in meters. If the units in your Shapefile were different than the units set up in WaterCAD, the program would automatically make the necessary conversions.



6. Fill in the next row, so that your entries correspond to the table below. Click **Next** when you are finished.

Pressure Junction Shapefile Connection

WaterCAD	Database	Unit
Elevation	ELEV	m
Demand	DEMAND	l/min

7. The wizard now takes you to the dialog for setting up the Pressure Pipe connections. Continue by filling in the information below for the Pressure Pipe and clicking **Next** to proceed to the next dialog. The Shapefile for each type of element will be located in the \Haestad\wtrc\lesson directory (for instance, select the prespipe.shp file for the pressure pipe connection), and the entry for **Key\Label Field** will always be **LABEL**. Your **Field Links** tables should look like the tables that follow.

Pressure Pipe Shapefile Connection

WaterCAD	Database	Unit
Diameter	D	mm
Material	MATERIAL	
Hazen-Williams C	C	

PRV Shapefile Connection

WaterCAD	Database	Unit
Elevation	ELEV	m
Diameter	D	mm
Initial HGL	HGL	m
Initial Valve Status	INITIAL_ST	

Pump Shapefile Connection

WaterCAD	Database	Unit
Elevation	ELEV	m
Shutoff Head	SHUT_H	m
Design Head	DES_H	m
Design Discharge	DES_Q	l/min
Maximum Operating Head	MAX_H	m
Maximum Operating Discharge	MAX_Q	l/min
Initial Pump Status	INITIAL_ST	

Reservoir Shapefile Connection

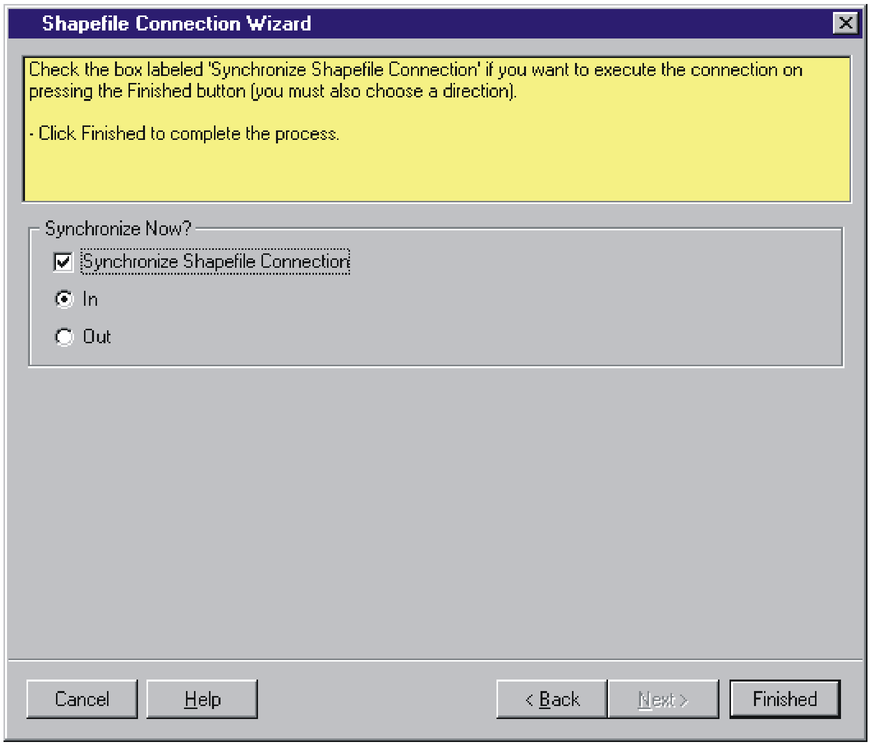
WaterCAD	Database	Unit
Elevation	ELEV	m

Tank Shapefile Connection

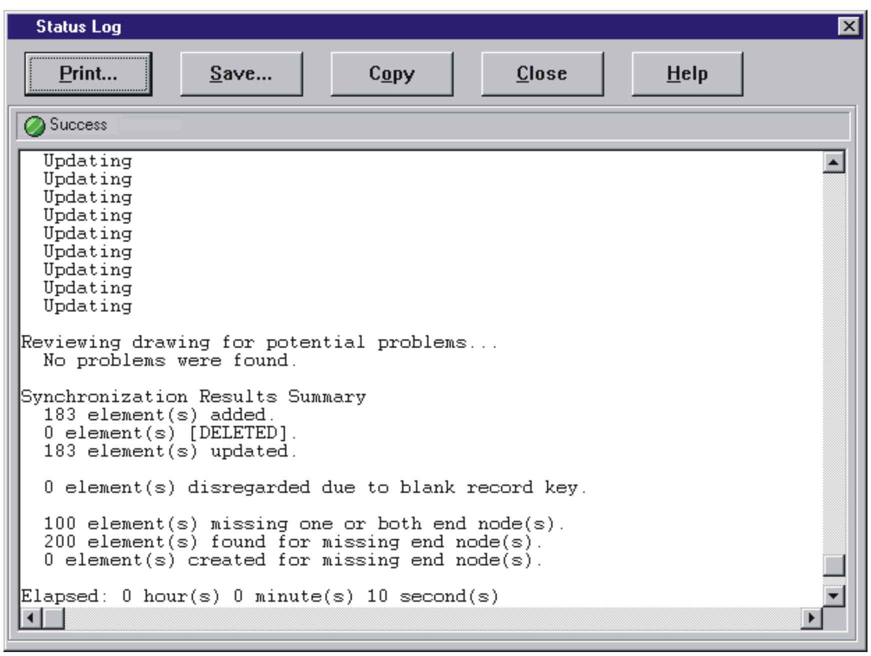
WaterCAD	Database	Unit
Tank Diameter	TANK_D	m
Base Elevation	BASE_ELEV	m
Minimum Elevation	MIN_ELEV	m
Initial HGL	INITIAL_HG	m
Maximum Elevation	MAX_ELEV	m

8. When you are finished setting up the Shapefile connections, click **Next** to proceed. The **Synchronize Now?** box will appear. The **Synchronize Shapefile Connection** box should be checked, and **In** selected because we will be reading data from the Shapefiles.

9. Click **Finished**, and **Yes** when prompted if you want to proceed.



10. A Status Log is generated showing the elements as data is read into the model. When the import is complete, you should get a yellow light in this window, indicating that the synchronization was successful, but that there are warnings. If there were no warnings, you would get a green light, and if there were errors, a red light. In this case, the warnings are due to the fact that we instructed WaterCAD to generate our network connectivity from the GIS spatial data. The log indicates where connectivity is being established, which is fine. **Close** the Status Log and click **OK** to return to the drawing pane.



11. Now, examine the network that you imported. Notice that it looks like the network from Lesson 6, and many of the pipes have bends and curves in them. Since you have topographic

information stored in the Shapefile, these bends can be imported. Because we created a scaled drawing, the pipe lengths will be read from the layout.

12. Notice also that the default Scenario, **Base**, is currently displayed as the current Scenario. Whenever data is brought in through a database or Shapefile connection, it is automatically written into the Alternatives referenced by the current Scenario. Similarly, whenever data is exported, the data associated with the current Scenario will be used.

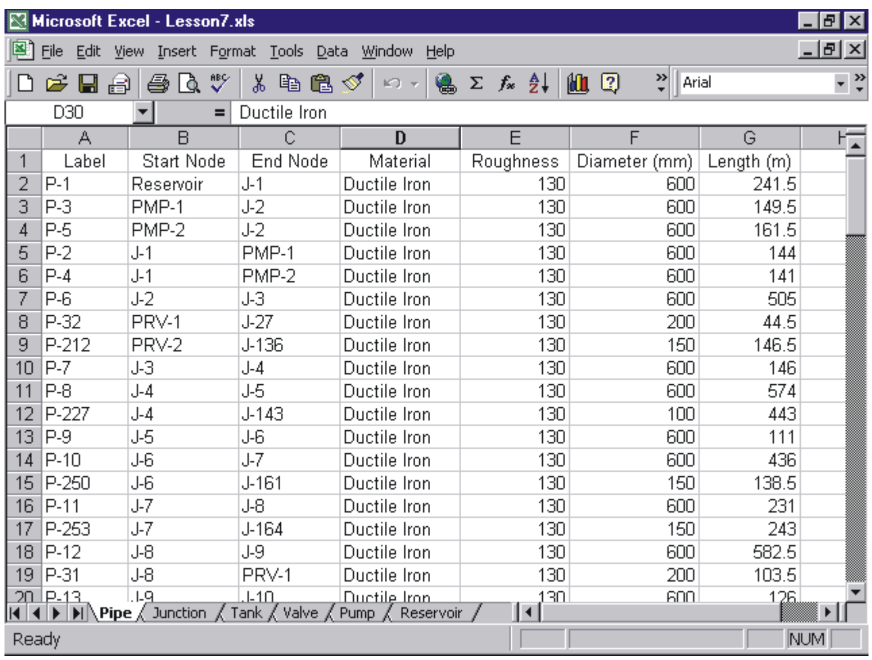
13. To run the model, click the **GO** button in the toolbar, and then click **GO** in the dialog. Now that you have calculated data, you could export the new data to your GIS database by going into the database and creating a new label for it. In Part 2 of this lesson, we will use an almost identical procedure to export pressures using database connections.

14. When you are finished, close the Scenario Editor. Proceed to Part 2 of this lesson or save your file as MyLesson7 and exit WaterCAD.

3.7.2 Part 2 – Importing Data from a Database

This portion of the lesson will take you through the steps to set up a connection to a database, in order to create a new water distribution network from existing data.

The necessary data has been included as a Microsoft Excel 5.0 spreadsheet. If you do not have software that can read this file type, you will still be able to perform the workshop, but you won't be able to open the data to view it externally.



	A	B	C	D	E	F	G
1	Label	Start Node	End Node	Material	Roughness	Diameter (mm)	Length (m)
2	P-1	Reservoir	J-1	Ductile Iron	130	600	241.5
3	P-3	PMP-1	J-2	Ductile Iron	130	600	149.5
4	P-5	PMP-2	J-2	Ductile Iron	130	600	161.5
5	P-2	J-1	PMP-1	Ductile Iron	130	600	144
6	P-4	J-1	PMP-2	Ductile Iron	130	600	141
7	P-6	J-2	J-3	Ductile Iron	130	600	505
8	P-32	PRV-1	J-27	Ductile Iron	130	200	44.5
9	P-212	PRV-2	J-136	Ductile Iron	130	150	146.5
10	P-7	J-3	J-4	Ductile Iron	130	600	146
11	P-8	J-4	J-5	Ductile Iron	130	600	574
12	P-227	J-4	J-143	Ductile Iron	130	100	443
13	P-9	J-5	J-6	Ductile Iron	130	600	111
14	P-10	J-6	J-7	Ductile Iron	130	600	436
15	P-250	J-6	J-161	Ductile Iron	130	150	138.5
16	P-11	J-7	J-8	Ductile Iron	130	600	231
17	P-253	J-7	J-164	Ductile Iron	130	150	243
18	P-12	J-8	J-9	Ductile Iron	130	600	582.5
19	P-31	J-8	PRV-1	Ductile Iron	130	200	103.5
20	P-13	J-9	J-10	Ductile Iron	130	600	126

This lesson uses the network from Lesson 6.

1. Open the spreadsheet file 'Lesson7.xls' and take a look at it. As you can see from the worksheet tabs, the data is organized into six worksheets, one for each type of element in the network. When setting up a spreadsheet yourself, you may organize and group data however you like. Just make sure that the different types of data are sorted into columns, with a descriptive heading in the topmost cell, and include a column for your labels.

1. Double-click the WaterCAD desktop icon to start WaterCAD Stand-Alone. If the **Welcome to WaterCAD** dialog appears, select the **Close** button.

2. Open the **Global Options** tab, accessed from the **Tools\Options** pull-down menu. Since we will be working in metric units, click the **Unit System** selection box, and select **System International**. Click **OK**.

3. In the pull-down menus, select **File\New**. Click **No** when asked if you want to save the current project. In the **Create Project File As** dialog, double-click the 'Lesson' folder, enter the file name 'dbprob.wcd' for your project, and click **Save**. The **Project Setup Wizard** will open.

4. In the Project Setup Wizard, title the project 'Lesson 7, Part 2 – Importing Data from a Database'. Click **Next**. Click the **Next** button again to leave this dialog set to its default values. In this dialog, set up the drawing as **Schematic**, and change the three **Annotation Multipliers** (**Symbol Size**, **Text Height** and **Annotation Height**) to **25**. Click **Next**, leave the Prototypes set to their default values, and click **Finished**.

1. Double-click the WaterCAD desktop icon to start WaterCAD for AutoCAD. Open the **Global Options** tab, accessed from the **Tools\Options** pull-down menu. Since we will be working in metric units, click the **Unit System** selection box, and select **System International**. Click **OK**.

2. Choose **New** on the **File** pull-down menu and select **No** when prompted to save the existing drawing. If the Create New Drawing dialog opens, move to step 3; otherwise, do the following:

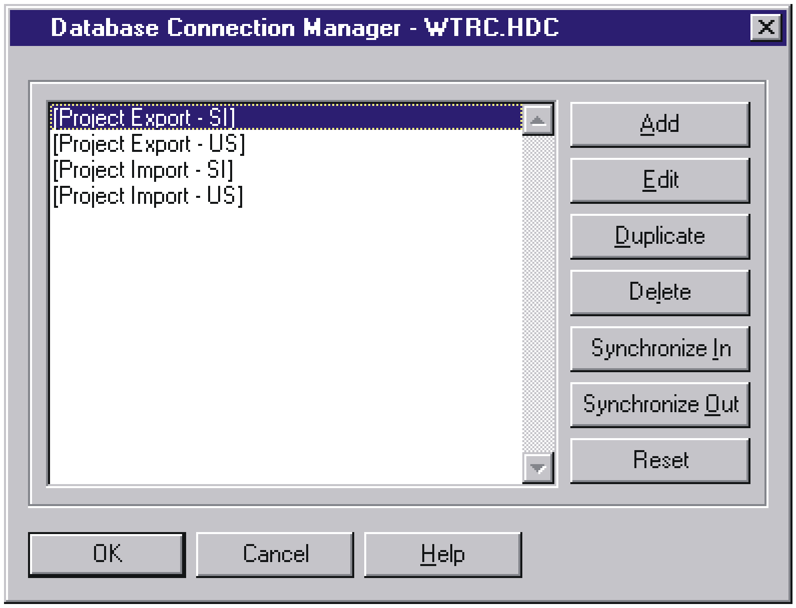
   Click the **Esc** key. Then, type **filedia** at the command prompt and press **Enter**. Type the value **1** and press **Enter**. Then, choose **New** on the **File** pull-down menu again, and don't save changes to the existing drawing. Note that the filedia variable controls whether some AutoCAD commands appear as dialogs or simply at the command prompt.

3. When the Create New Drawing dialog appears, make sure that **Metric** is selected, and click **OK**. Answer **Yes** when asked if you want to set up the project. In the Project Setup Wizard, title the project 'Lesson 7, Part 2 – Importing Data from a Database' and click **Next**. Click **Next** again to accept the defaults on the second screen.

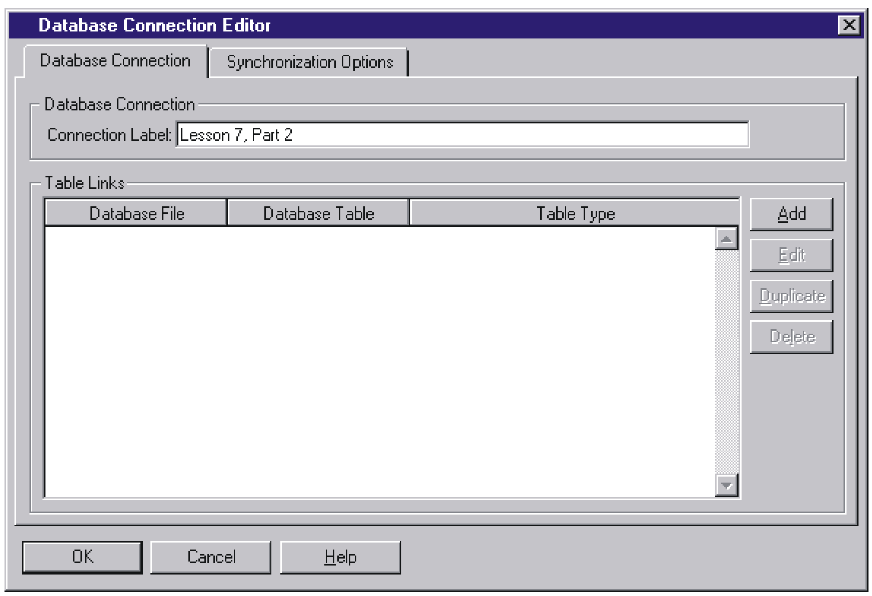
4. In this dialog, set up the drawing as **Schematic**, and change the three **Annotation Multipliers** (**Symbol Size**, **Text Height** and **Annotation Height**) to **25**. Click **Next**, leave the Prototypes set to their default values, and click **Finished**.

The remaining commands are for both Stand-Alone and AutoCAD modes.

1. Select **File\Synchronize\Database Connections** from the pull-down menus. Click **Add**.



2. Enter the **Connection Label** ‘Lesson 7, Part 2’ for this connection, and click the **Add** button.



3. Set the **Database Type** to **Excel 5.0**. Click the ellipses button next to the **Database File** field, and browse to select the ‘Lesson7.xls’ file from the \Haestad\Wtrc\Lesson directory.

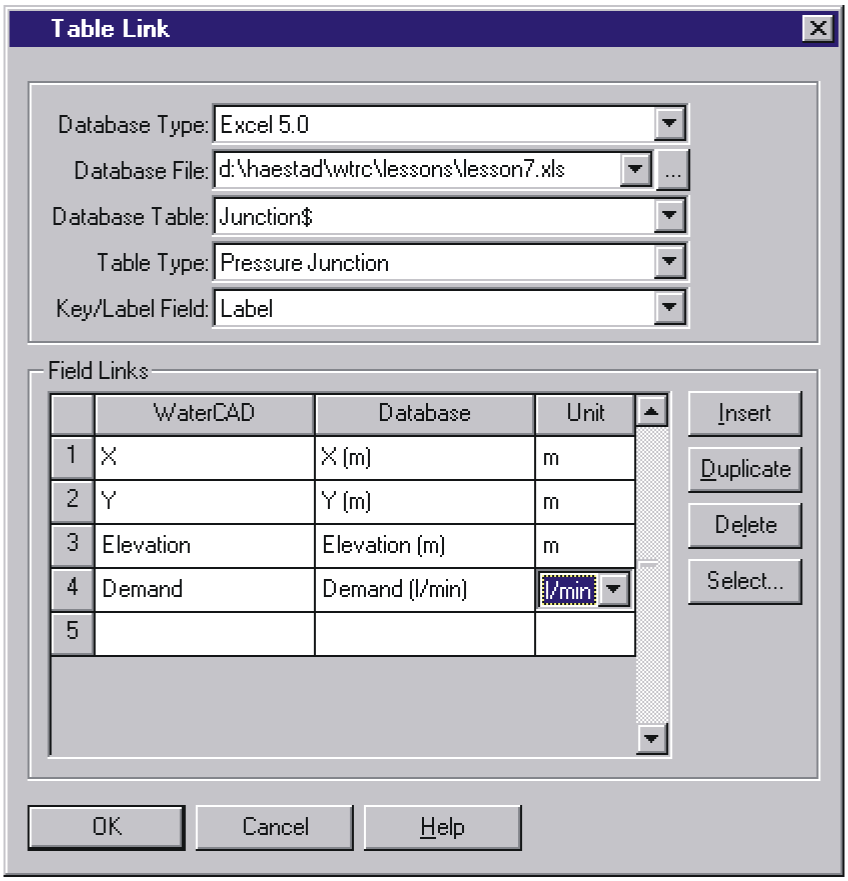
4. Click the **Database Table** list box. Notice that the items in the list correspond to the different worksheet tabs in your spreadsheet file. Select **Junction$** from the list, and **Pressure Junction** for the **Table Type**. Set the **Key/Label** field to **Label**. This item designates the field that WaterCAD matches with its own element labels, so that data will be assigned to the correct place.

5. Using the **Field Links** table, you must now match the data types available in WaterCAD to the data types you will be bringing in from the spreadsheet. In row 1, select **X** from the **WaterCAD** column, and **X (m)** from the **Database** column. Set the **Unit** to **m** to let WaterCAD know that the coordinate it is reading from the spreadsheet is in meters. If the units in your database were different than the units set up in WaterCAD, the program would automatically make the necessary conversions.

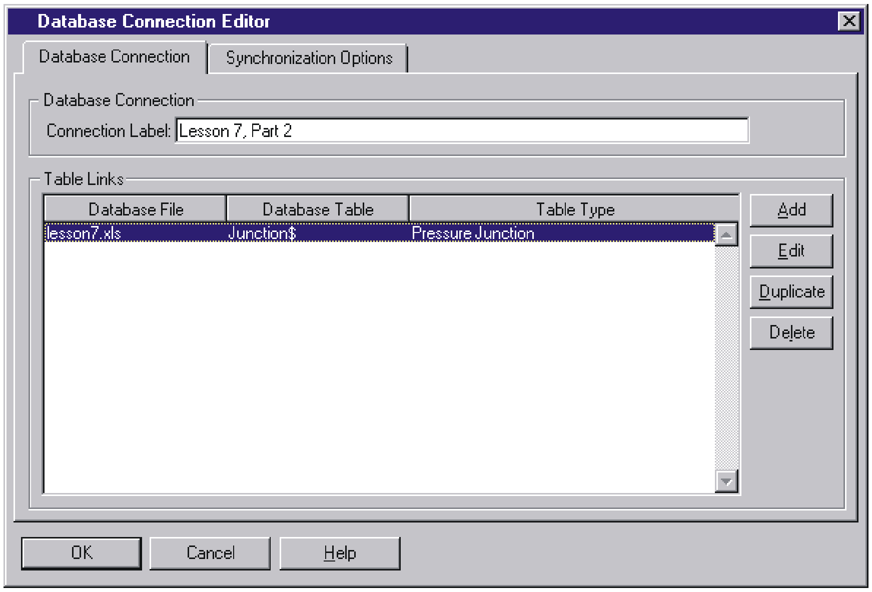
6. Fill in the remaining rows, so that your entries correspond to the table below. Click **OK** when you are finished.

Junction Database Connection

WaterCAD	Database	Unit
X	X (m)	m
Y	Y (m)	m
Elevation	Elevation (m)	m
Demand	Demand (l/min)	l/min



7. You should now be back in the **Database Connection** dialog. Click the **Add** button, and set up your database connection for pipe data. Use the same spreadsheet file you used for the junction data, but set the **Database Table** and **Table Type** to **Pipe$** and **Pressure Pipe**, respectively. Your **Key/Label Field** is again **Label**. Then, set up the following Pipe Database connection.



Pipe Database Connection

WaterCAD	Database	Unit
+Start Node	Start Node	
+Stop Node	End Node	
Diameter	Diameter (mm)	mm
Material	Material	
Hazen-Williams C	Roughness	
Length	Length (m)	m

8. Repeat the above procedure to set up connections for Pump, Reservoir, Tank and Valve connections, using information from the following tables.

Pump Database Connection

WaterCAD	Database	Unit
X	X (m)	m
Y	Y (m)	m
Elevation	Elevation (m)	m
Shutoff Head	Shutoff Head (m)	m
Design Head	Design Head (m)	m
Design Discharge	Design Q (l/min)	l/min
Maximum Operating Head	Max# Head (m)	m
Maximum Operating Discharge	Max# Q (l/min)	l/min
Initial Pump Status	Initial Status	

Reservoir Database Connection

WaterCAD	Database	Unit
X	X (m)	m
Y	Y (m)	m
Elevation	Elev# (m)	m

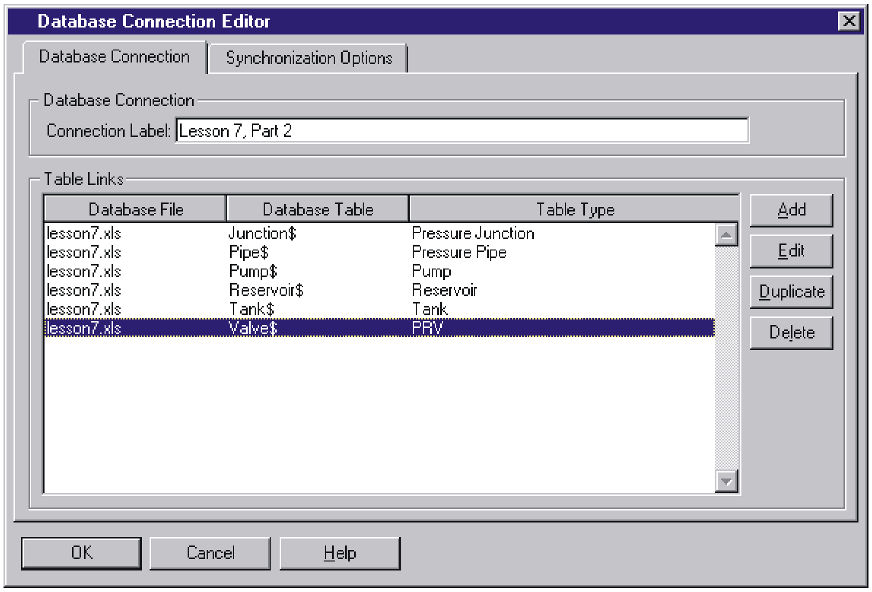
Tank Database Connection

WaterCAD	Database	Unit
X	X (m)	m
Y	Y (m)	m
Tank Diameter	Tank Diameter (m)	m
Base Elevation	Base Elev# (m)	m
Minimum Elevation	Minimum Elev# (m)	m
Initial HGL	Initial Elev# (m)	m
Maximum Elevation	Maximum Elev# (m)	m

PRV Database Connection

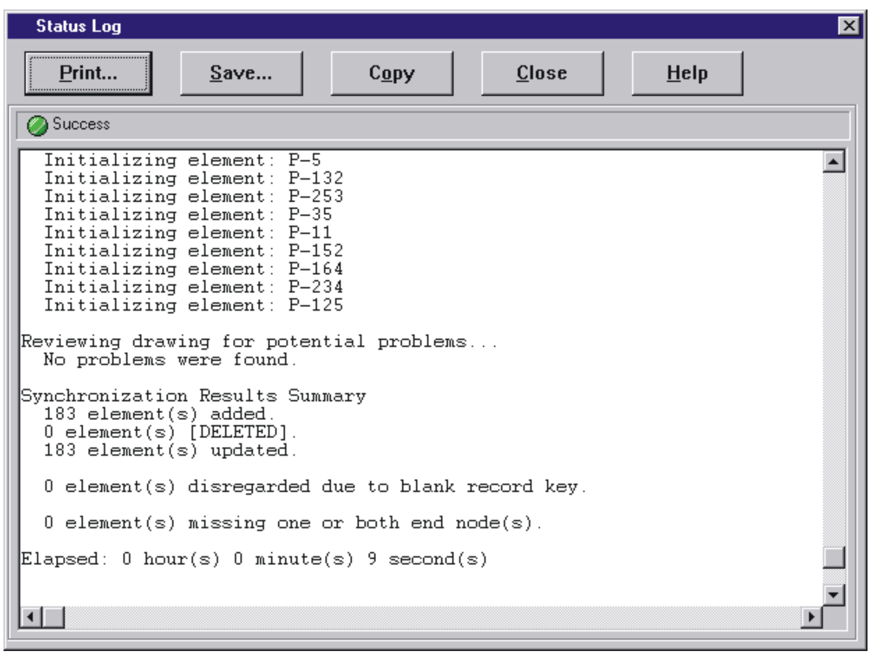
WaterCAD	Database	Unit
X	X (m)	m
Y	Y (m)	m
Elevation	Elevation (m)	m
Diameter	Diameter (mm)	mm
Initial HGL	Initial Grade Setting (m)	m
Initial Valve Status	Initial Status	

The Table Type for this connection is PRV.

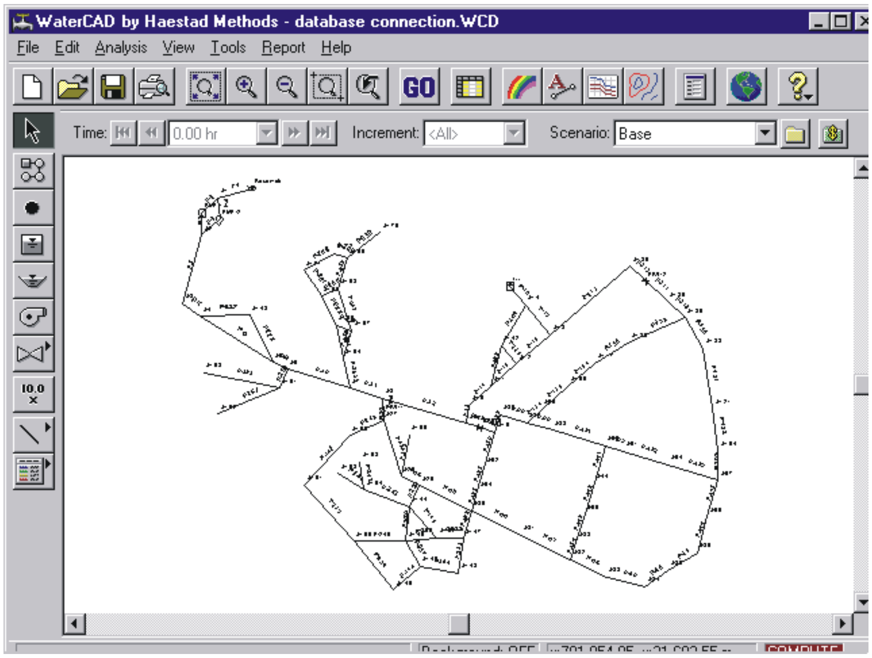


9. When you are finished setting up the database connections, click **OK** to close the **Database Connection** editor, and then click the **Synchronize In** button. WaterCAD will give you a confirmation prompt. Click **Yes** to proceed. You will then get a message asking if you want to add an element. Click **Yes to All**.

10. A Status Log is generated showing the elements as data is read into the model. When the import is complete, you should get a green light in this window. If there were warnings or errors you would get a yellow light or red light, respectively. You could then scroll through the log to see where any problems might be occurring. Click **Close** to exit the Status Log, and **OK** to exit the **Database Connection Manager**.



11. You should now be able to see the imported network in the drawing pane, but the symbol and label sizes are very small. Select **Tools\Options** from the pulldown menus, and then click the **Drawing** tab. Set all three **Annotation Multipliers** to 25, and click **OK**.

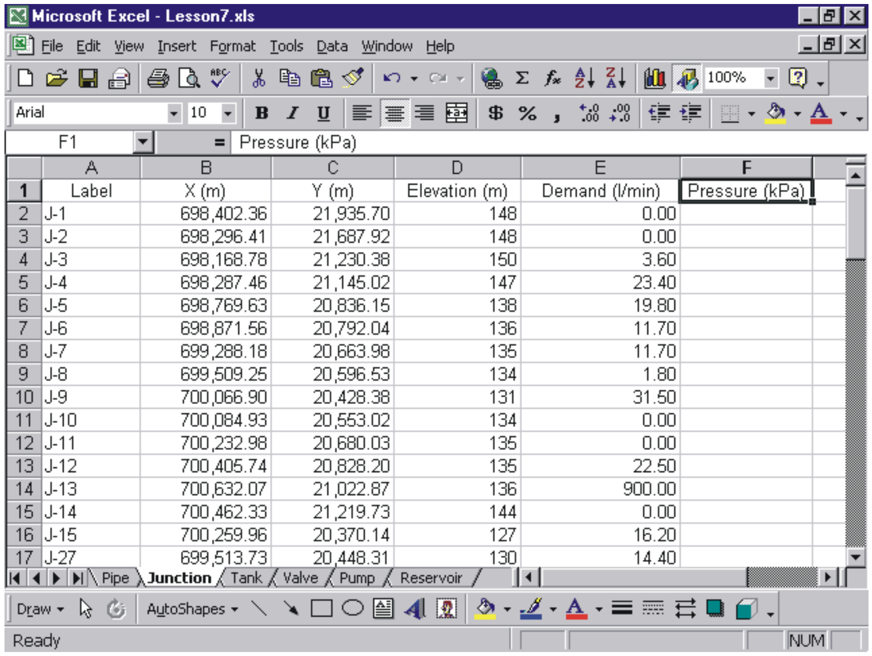


12. Now, examine the network that you imported. Notice that it is different in appearance from the same network imported using a shapefile in Part 1 of this lesson. The difference stems from the fact that in a database connection, a pipe's layout is defined only by the location of its end nodes. Therefore, pipes appear without bends, making a straight line connection between nodes. Hydraulically, your model will not be affected, since the pipe lengths are user-defined, not scaled from the layout.

13. Notice also that the default Scenario, **Base**, is currently displayed as the current Scenario. Whenever data is brought in through a database or Shapefile connection, it is automatically written into the Alternatives referenced by the current Scenario. Similarly, whenever data is exported, the data associated with the current Scenario will be used.

14. Click the **GO** button, and then click **GO** again to run the model. Now that you have calculated data, we can export it back to our database. For this example, we will only export pressures at the junction nodes. Go ahead and close the **Scenario Editor**.

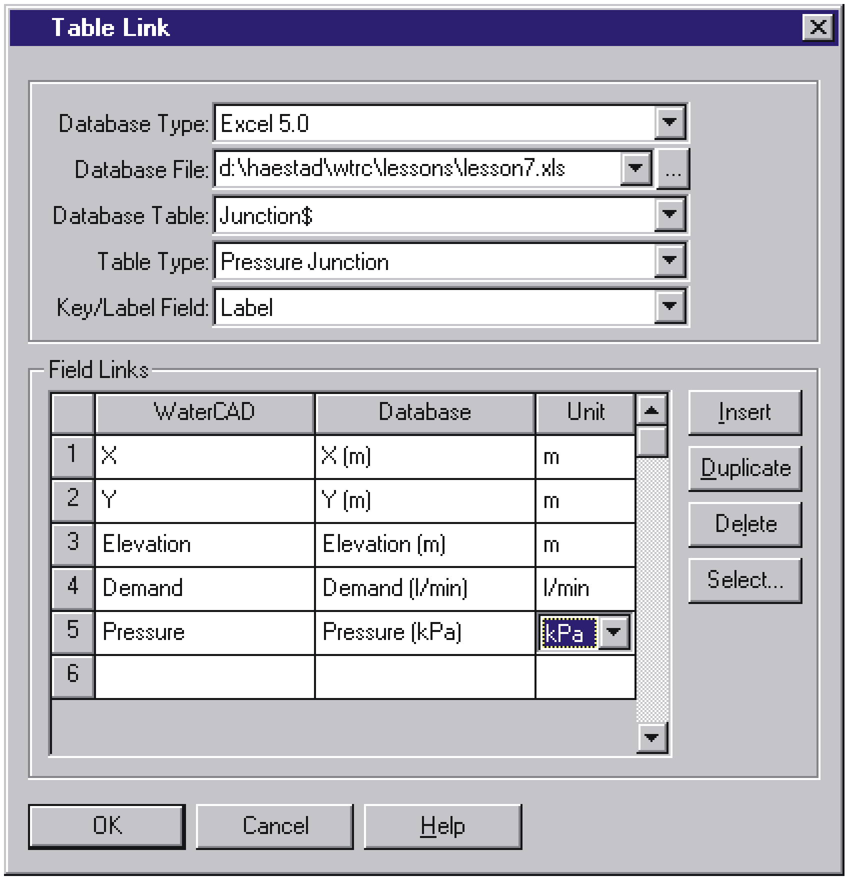
15. Use Excel to open 'Lesson7.xls' in another window. Click on the tab for the **Junction** worksheet, and add a new column heading in cell F1 called 'Pressure'. Save and Close the file.



	A	B	C	D	E	F
1	Label	X (m)	Y (m)	Elevation (m)	Demand (l/min)	Pressure (kPa)
2	J-1	698,402.36	21,935.70	148	0.00	
3	J-2	698,296.41	21,687.92	148	0.00	
4	J-3	698,168.78	21,230.38	150	3.60	
5	J-4	698,287.46	21,145.02	147	23.40	
6	J-5	698,769.63	20,836.15	138	19.80	
7	J-6	698,871.56	20,792.04	136	11.70	
8	J-7	699,288.18	20,663.98	135	11.70	
9	J-8	699,509.25	20,596.53	134	1.80	
10	J-9	700,066.90	20,428.38	131	31.50	
11	J-10	700,084.93	20,553.02	134	0.00	
12	J-11	700,232.98	20,680.03	135	0.00	
13	J-12	700,405.74	20,828.20	135	22.50	
14	J-13	700,632.07	21,022.87	136	900.00	
15	J-14	700,462.33	21,219.73	144	0.00	
16	J-15	700,259.96	20,370.14	127	16.20	
17	J-27	699,513.73	20,448.31	130	14.40	

16. Go back to the WaterCAD window, and choose **File\Synchronize\Database Connections** from the menu. Highlight 'Lesson 7, Part 2', and click the **Edit** button. Select the junction table from the list, and click **Edit** again.

17. In Row 5 of the Field Links table, link WaterCAD's **Pressure** to the Database's **Pressure**. The Unit should be set to **kPa**. Click **OK** and **OK** again to get back to the Database Connection Manager. Click the **Synchronize Out** button to send the information back to the spreadsheet.



18. Finally, if you reopen the ‘Lesson7.xls’ file in Excel, you will see that the pressure values have now been added.

3.7.3 Part 3 – Converting CAD Drawing Entities

The Polyline to Pipe tool enables you to take existing CAD entities and use them to quickly construct a water distribution network. Although this feature is called Polyline to Pipe, Line and Block entities can be converted as well (Polylines and Lines can be converted to pipes; Blocks can be converted to any available node type).

Building a model based on graphical elements can be an error-prone process. Difficulties can arise due to the fact that a drawing may appear to be correct visually, but may contain problems that are not readily apparent. For example, what appears to be a single line in a drawing could in fact be made up of many line segments – or it could be made up of 2 lines, one directly on top of another.

The Polyline to Pipe Wizard will guide you through the conversion process, enabling you to set up options relating to tolerances, node creation, and handling T-intersections. To help alleviate some of the problems that you may encounter during the import process, a comprehensive drawing review is also performed. During conversion, the network is analyzed, and potential problems are flagged for review. After performing the conversion, the Drawing Review window will allow you to navigate to and fix any problems that may be encountered.

For users in Stand-Alone mode only:

1. Open WaterCAD and choose **Tools\Options** from the pulldown menus. In the **Global Options** tab, make sure that the **Unit System** is set to **System International**, and then click **OK**. Next, select **File\Import\Polyline to Pipe**. You will be asked if you want to set up the project. Click **Yes** to start the Project Setup Wizard.

For users in AutoCAD mode only:

1. Start WaterCAD for AutoCAD and open the file 'Lesson7.dwg' in the \Haestad\Wtrc\Lesson directory. Select **Edit\Change Entities to Pipes** from the menus. The AutoCAD command line will prompt you to 'Select objects'. Draw a selection window around all of the objects in the drawing by clicking the upper left and lower right corners, then click the right mouse button. Click **Yes** when asked if you wish to set up the project.

The following commands are for both Stand-Alone and AutoCAD modes.

1. In the Wizard, enter 'Lesson 7 -- Polyline to Pipe' as the project title, click **Next**, and **Next** again to accept the default settings. Make sure that you are set up for a **Scaled** drawing, with a horizontal scale of 1:**5000** and a vertical scale of 1:**500**. Set the three **Annotation Multipliers** to 2.8. Click **Next** again.

2. In order to minimize your data input later, create prototypes for common element characteristics. The most common type of pipe in the model you will be creating is 150 mm ductile iron with a C of 130. Make sure these characteristics coincide with the prototype values, and click **OK**.

3. Since you have two identical pumps, go ahead and set up a prototype for them, using the data below. Be sure to change the units to **l/min** before entering the discharge values. Click **OK** when you are finished.

Elevation (m)	Pump Type
148	3 Point

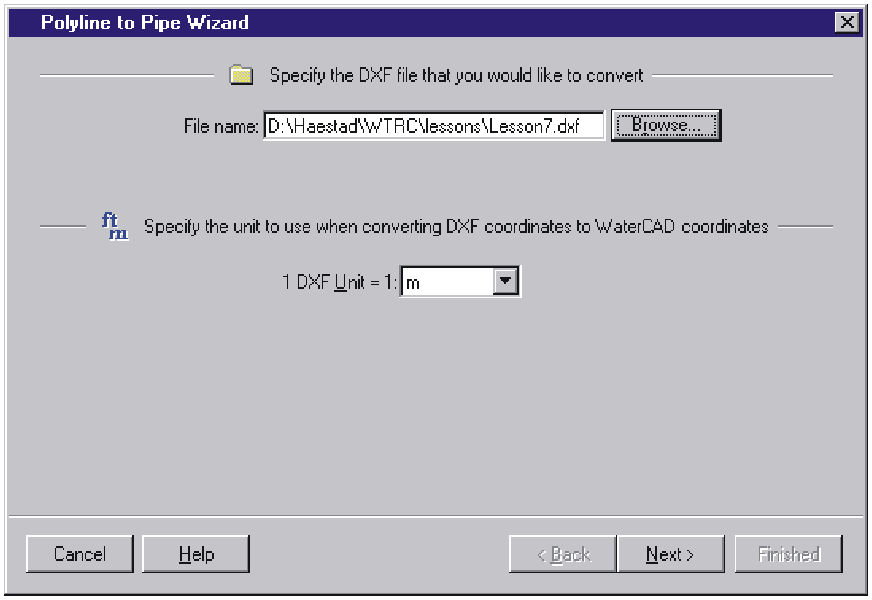
Pump Data

	Head (m)	Discharge (l/min)
Shutoff:	70.0	0
Design:	50	1200
Max. Operating	35	2000

4. Create one more prototype, this time for the PRV's. They both have an elevation of 129 m and an HGL setting of 185.2 m. Click **OK**, and then **Finished**. The Polyline to Pipe Wizard now opens.

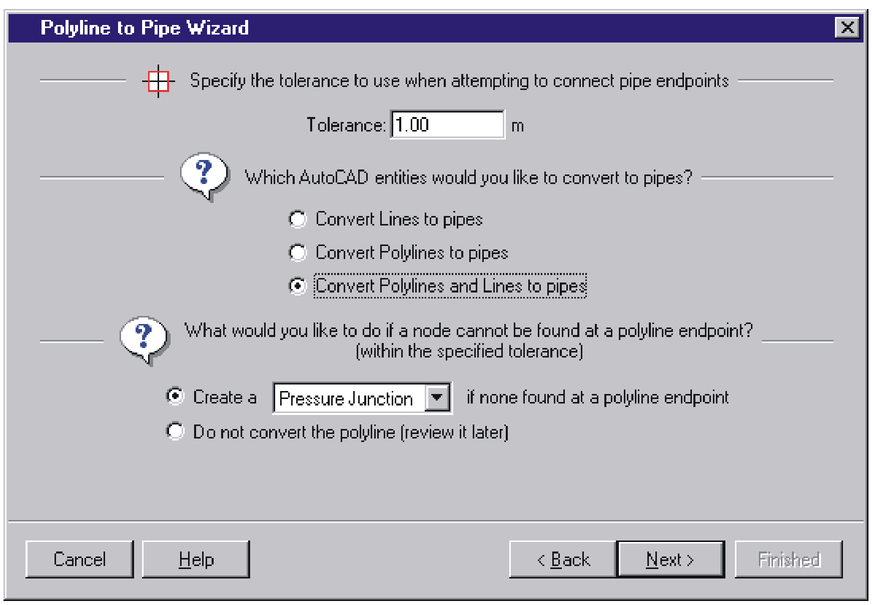
For users in Stand-Alone mode only:

1. Browse to open the file 'Lesson7.dxf', located in the Haestad\Wtrc\Lesson directory. Leave the DXF unit set to meters, and click **Next**.

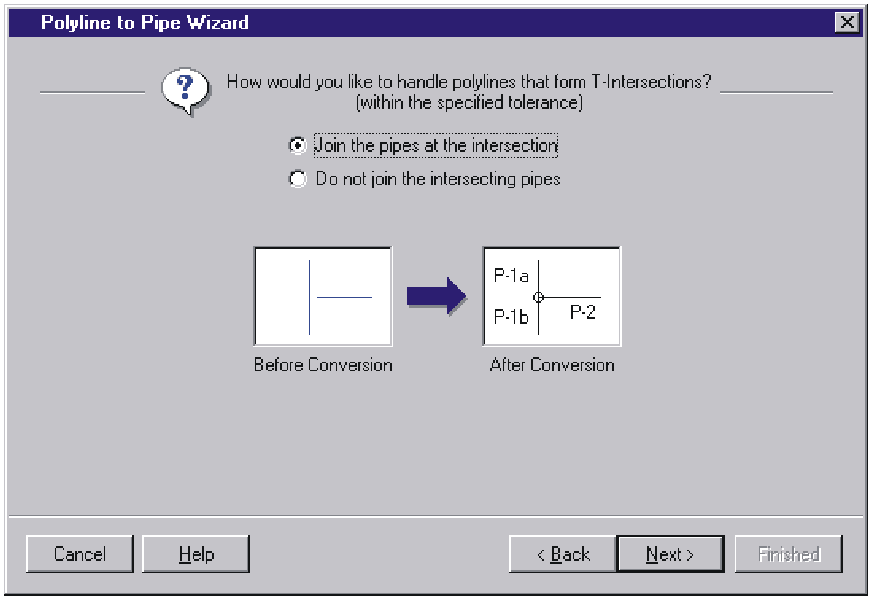


For all users:

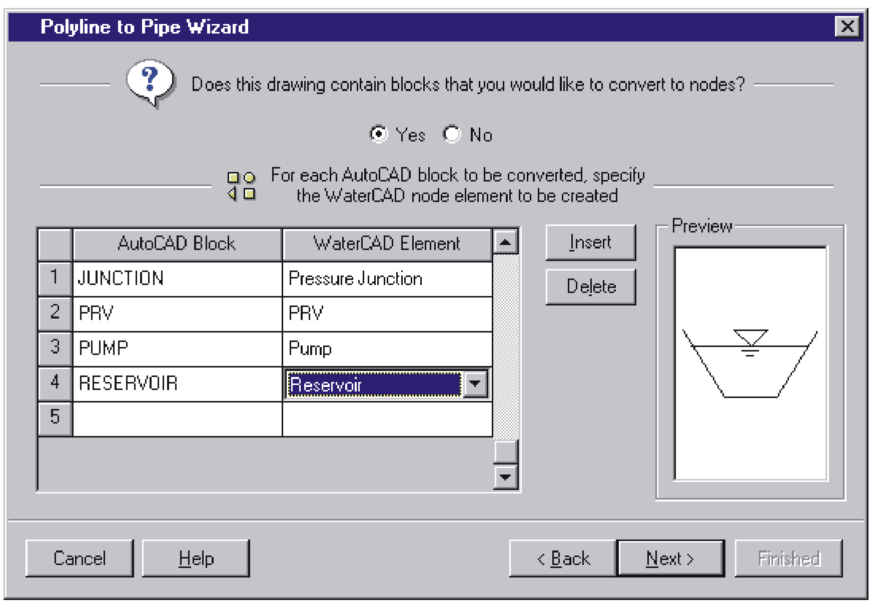
1. Now, set up the options WaterCAD will use when performing the conversion. Change the **Tolerance** to 1 m, so that pipe endpoints that come within a meter of one another will be assumed to be connected. Select the radio button for **Convert Polylines and Lines to pipes**, and tell WaterCAD create a **Pressure Junction** if no node is found at a polyline endpoint. Click **Next**.



2. Select the option to join pipes at T-intersections within the specified tolerance, and click **Next**.

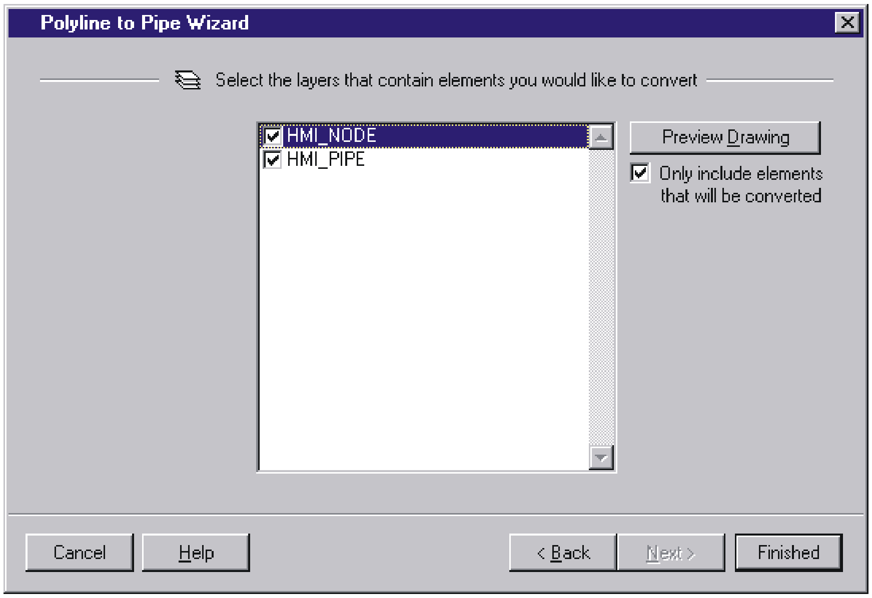


3. Select **Yes** when asked if you have blocks that you would like to convert to nodes, and fill in the table by matching the AutoCAD Blocks JUNCTION, PRV, PUMP, RESERVOIR, and TANK with the corresponding WaterCAD elements (Pressure Junction, PRV, Pump, Reservoir, and Tank). Click **Next**.

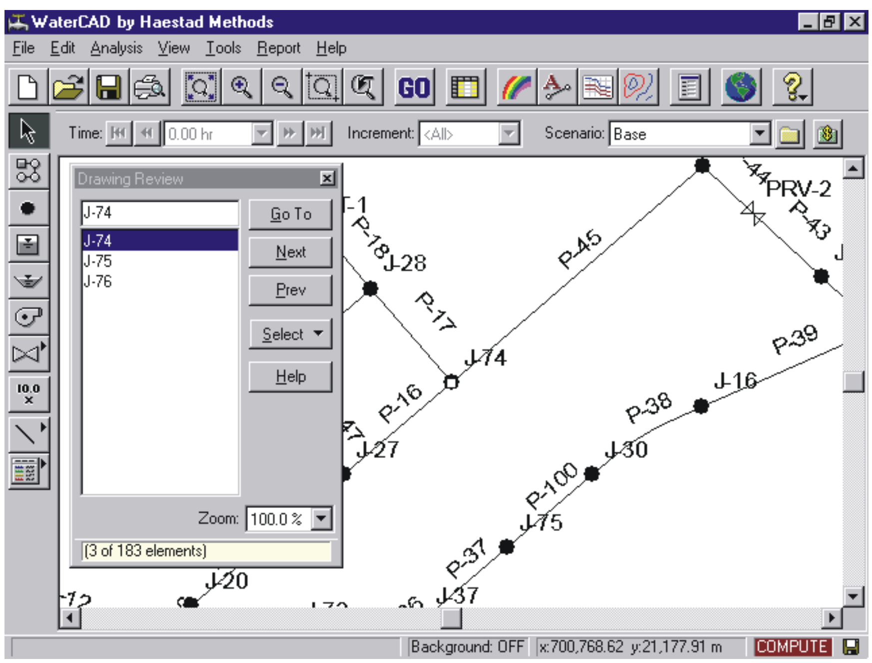


4. You will be given the option to alter the prototype settings. This option is useful if you want to import in multiple passes, grouping like data together to make the data entry process more automated. For instance, we could have chosen to import all of the 100 mm pipes, then the 150 mm pipes, etc., changing the prototype each time. For this example, we will leave the prototypes as we set them in the Project Setup Wizard. Click **Next**.

5. Make sure that the layers **HMI_NODE** and **HMI_PIPE** are both checked, and click **Finished** to perform the conversion. When it is completed, **Close** the statistics window.



6. A Drawing Review dialog element opens, with junctions listed. The purpose of the Drawing Review is to alert you to problems or assumptions made during the import. Find any one of these junctions by highlighting it in the list and clicking **Go To**. The drawing pane will center on the junction and select it. If you have difficulty seeing the selected element, increase the zoom factor in the **Drawing Review** dialog.



7. Open the element, and click the **Messages** tab. There will be a message telling you that the node was added during the Polyline to Pipe conversion. The junction had to be added because there was no node at that location in your DXF drawing, but there was a polyline endpoint. In the Polyline to Pipe Wizard, you instructed WaterCAD to add junctions to endpoints.

   Even though you now have your drawing converted to a pipe network, it is still not ready to be run, because you can only bring in element types and network connectivity using this type of import. Before you could run this model, you would have to input data for elevations, demands, pipe sizes, etc., either directly into WaterCAD or through database connections.

For users in AutoCAD mode only:

1. The WaterCAD elements are now on layer 0, since that layer was current when you performed the conversion. If you turn off layers HMI_PIPE and HMI_NODE, only the actual WaterCAD elements will be visible.

Chapter 4

Starting a WaterCAD Project

This chapter describes how to start a new project as well as the files that WaterCAD creates to save your project's data. At the beginning of a project, you need to set some global settings. In the **Global Options** and **Project Options** dialogs, you can specify the unit system, the friction formula(s) used, and whether you want to use Auto Prompting. In the **Drawing Options**, you can specify settings such as the drawing scale and the size of the symbols and annotations.

> **These options can also be viewed or edited by selecting the Options menu item from the Tools pull-down menu.**

You can also access the **FlexUnit** dialog in order to globally specify the units and number of decimals to be displayed for each type of data.

4.1 File Management

WaterCAD uses the .WCD extension to store all model input data (including element inputs, alternatives and scenarios, etc.) both for Stand-Alone and AutoCAD modes.

When WaterCAD runs within AutoCAD, two important files are used. The .WCD file is still used to hold all model data, and a .DWG file contains all of the AutoCAD entities. This means that even a complete AutoCAD drawing corruption (or loss) will not endanger your hydraulic model data – in fact, you can even regenerate the AutoCAD modeling elements from the .WCD file!

WaterCAD Backup Files

When a .WCD file is overwritten by a save action, a backup file of the .WCD file is created with a .WCK extension. Note that AutoCAD also generates a backup drawing file, with a .BAK extension.

Shapefile/Database Connection Files

If you use the Shapefile or Database Connections Managers to exchange data with external data sources, the connection files you create/use have the .HSC and .HDC extension respectively. By default they are named WTRC.HSC and WTRC.HDC and are located in your **haestad\wtrc** directory. In this case, these connections are shared between all your projects. You are also given the option to keep these connections local to your project. In this case, these files are placed in your current project directory, with the same name as your .WCD file but with the .HSC and .HDC file extensions, respectively. See the "Sharing Shapefile Connections between Projects" and "Sharing Database Connections between Projects" topics in the "GIS and Database Connections" chapter for more information.

Libraries

Libraries are saved in separate files so they can be shared between projects. The paths are specified in the Engineering Library Manager. They have a .HLB extension, and are located by default in your **haestad\wtrc** directory.

Additional Files

WaterCAD creates additional files in the same directory as your .WCD file to save the calculation results. Since recalculating the scenarios can regenerate these results, these files do not necessarily need to be included when backing up your important files. However, if you are unsure, simply copy all files present in your project directory.

It is recommended to use a separate directory for every WaterCAD project on your computer to facilitate project management and backup.

4.1.1 Import Command

The **Import** command allows you to import data from KYpipe v1, v2 or v3; EPANet v1 or v2; and Cybernet v1 or v2. You will then be able to save this project as a WaterCAD v4 project. The data is imported into an empty project, therefore, before the data is imported, you will be prompted to save your current project if it contains WaterCAD elements.

The current project remains in schematic mode when importing data.

We've made every effort to prevent the loss of data during these imports. However, all imported data should be checked for accuracy.

WaterCAD v3 projects should simply be opened in WaterCAD v4 (as any WaterCAD v4 project), without using the Import command. However, once you save the project in WaterCAD v4, the project files cannot be opened in WaterCAD v3 any longer.

To access the **Import** dialog, select **Import\Network** from the **File** pull-down menu.

4.1.2 Multiple Sessions

WaterCAD does not support multiple sessions. WaterCAD uses a single document model and support for multiple views has not been implemented. Therefore, do not try to open more than one session of WaterCAD at the same time or data loss and data corruption could occur.

4.2 Project Management

4.2.1 Project Setup Wizard

The **Project Setup Wizard** dialog can only be accessed at the start of a new project (**File\New** from the pull-down menus, or from the **Beginner's Welcome** dialog). All of the options that are edited from the wizard, however, can be changed individually from other pull-down menus.

The **Project Setup Wizard** assists you in the creation of a new project by stepping you through many of the project-wide options allowing you to set most of your notes and defaults before you even create the first pipe.

The areas covered by this wizard include:

- **Project Summary** – Includes information about the project, such as the project title, the project engineer, and general comments.
- **Project Options** – Include information regarding global options, such as the desired friction method and coordinate system.
- **Drawing Options** – Include information regarding the drawing pane, such as the drawing scale, annotation multipliers, and background drawing data (for Stand-Alone mode only).
- **Prototypes** – Enable you to set default values for elements, which are used to initialize values for any new elements that are added to the project.

4.2.2 Creating a New Project

To create a new project in Stand-Alone:

1. At the **Welcome** dialog, click **Create New Project** and click **OK**. Or, select **File\New** from the pull-down menu. The **Create Project File As** window appears so that you can enter a new file name for your project.
2. Select the appropriate drive and directory where you want to save the project.
3. Enter the file name. Then click **OK**.
4. Follow the **Project Setup Wizard**.

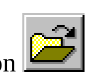
To create a new project in AutoCAD:

1. Select **File\New** from the pull-down menu.
2. The **Create Project File As** window appears so that you can enter a new file name for your project.
3. Select the drive and directory where you want to save the project.
4. Enter the file name. Then click **OK**.
5. Follow the Project Setup Wizard.

4.2.3 Opening an Existing Project

To open an existing project:

1. Do one of the following:

   From the toolbar, click the **Open** button 

   - or -

   At the **Welcome** dialog, click **Open Existing Project**, then click **OK**.

   - or -

   Choose **File\Open** from the pull-down menus. The **Open File** window appears.
2. Select the appropriate drive and directory where the project is located. The program displays all the projects stored in that directory.
3. Select the desired file name and click **Open** or **OK**.

4.2.4 Saving a Project

To save an existing project to disk:

1. Select **File\Save** from the pull-down menus. The **Save Project File As** dialog will appear. If the project has been previously saved, the command will execute. Otherwise, the next two steps must be completed.
2. Select the drive and directory where you want the project located. The program displays all the projects stored in that directory.
3. Enter the file name under which you wish to save the file.

4.2.5 Project Summary

The **Summary** dialog provides a way to enter a Project Title, the name of the Project Engineer, and any significant Comments (for example, the project revision history). The **Date** field defaults to the current day. To change any portion of the date, click the item to change (i.e: month field), then use the up and down arrows to set the date.

The Project Title and Project Engineer will print in the footer of the Reports.

To access the **Project Summary** dialog select **File\Project Summary** from the pull-down menu.

4.3 Options

4.3.1 Global Options

The **Global Options** dialog allows you to customize the following options for this application:

- **Welcome** dialog (Stand-Alone mode only)
- Unit System
- Enter Key Behavior
- Background/Foreground Color (Stand-Alone mode only)
- Sticky Tool Palette (Stand-Alone mode only)
- Auto Prompting
- Right-Click Context Menu (This toggle exists only in AutoCAD R14; that option is automatically On in AutoCAD 2000)

To access the **Global Options** tab, select **Tools\Options** from the pull-down menus.

Welcome Dialog

The **Welcome** dialog appears when the program is started, and provides easy access to common tasks you may want to perform when you first start using the program. The following options are available:

- Tutorials
- Create New Project
- Open Existing Project
- Exit Program

Unit System

Although individual units can be controlled throughout the program, you may find it useful to change your entire unit system at once to either the **System International** (Metric) unit system or the **US Customary** (English) system.

When you switch to a different unit system, you will be asked to confirm this action. If you choose **YES**, all data will be displayed in the default unit for the selected system.

If the file that you are editing in Stand-Alone mode is already associated with an AutoCAD drawing, be careful not to change the unit systems. Otherwise, the .DWG and the .WCD files may become irreversibly out of sync.

Enter Key Behavior

This controls which standard the **Enter** key follows during editing:

- **CUA Enter Key** – With this setting, the **Enter** key acts as it normally does for Windows applications. It is conforming to Common User Access (CUA) standards. This means that when you press the **Enter** key, it is as though you pressed the default button on the dialog. CUA Enter Key is the recommended setting.
- **Tabbing Enter Key** – With this setting, the **Enter** key behaves the same as the **Tab** key for editable fields (not buttons). This means that when you press the **Enter** key, the cursor will move to the next field in the dialog.

Window Color

You can specify the background and foreground colors of the main graphical window in Stand-Alone mode. The foreground color is the default color that is applied to all elements symbols, pipes, labels, and annotations when no color coding is defined. These color settings also apply to the **Scenario Comparison** window, but do not apply to the **Profile** or **Graph Plot** windows.

Sticky Tools

Available in Stand-Alone mode. With **Sticky Tools** disabled, the drawing pane cursor will return to the Select tool after creating a node or finishing a pipe run. With **Sticky Tools** enabled, the tool does not reset to the Select tool, allowing you to continue dropping new elements into the drawing without reselecting the tool.

The Sticky Tool Palette can be turned on or off to meet your needs and preferences.

Auto Prompting

Auto Prompting allows you to immediately enter data as elements are added to the drawing, without interrupting the layout process.

When Auto Prompting is active, the **Auto Prompting** dialog will immediately appear when you add an element to the drawing. From the **Auto Prompting** dialog, you can modify the element's default label, and you can access the remaining input data by clicking the associated **Edit** button. Auto Prompting can also be toggled off in this dialog.

Right-Click Context Menu Option in AutoCAD Mode

If the Right-Click Context Menu option is enabled, a right mouse click on a WaterCAD entity in AutoCAD R14 will pop-up a context menu for editing or modifying the element (this functionality emulates the ability that is available in WaterCAD Stand-Alone mode). Right-clicking over any other entity in the drawing will invoke standard AutoCAD right-click behavior.

In AutoCAD 2000, this option is always available. Simply select the element in the AutoCAD drawing and right-click to obtain a pop-up menu, from which you can select Edit.

4.3.2 Project Options

The **Project Options** dialog allows you to set essential information about your project, grouped into the following:

- Friction Method
- Liquid
- Input Modes
- Pipe Length Rounding

To access the **Project Options** tab, select **Tools\Options** from the pull-down menus.

Friction Method

The friction method option enables you to select the methodology for determining flow resistance and friction losses during calculations.

Available methodologies include:

- Darcy-Weisbach: Colebrook-White Equation
- Hazen -Williams Formula
- Manning's Formula

If you change the friction method after pipes have been entered into the network, the program will ask if you want to update the roughness values of those pipes. If you select **Yes**, the program will assign all pipes a new roughness that corresponds to the default roughness of the pipe material.

Liquid

You can specify the type of liquid transported by the network, the characteristics of which (kinematic viscosity and specific gravity) are defined in the Liquid Library.

The kinematic viscosity is used in determining the friction coefficient in the Darcy-Weisbach Friction Method.

Input Modes

WaterCAD supports several input modes to adjust data entry to your style or the needs of a particular project.

- **Coordinates** – Coordinates can be displayed either in X and Y format or as Northing and Easting. Whichever coordinate input mode is chosen will be active everywhere within the program (element editors, FlexTables, etc.).

- **Settings –** This drop-down list allows you to set whether values on control conditions will be input in terms of hydraulic grade or pressure. Note that regardless of the mode you choose, the program will always display values in both hydraulic grade and pressure.
- **Tank Levels –** This drop-down list allows you to set whether tank operating ranges will be input in terms of elevations (height above a datum elevation of 0) or levels (height above the wet well's base elevation).

Pipe Length Rounding

Pipe length rounding is used to determine the level of precision desired for scaled pipe lengths. Pipe lengths will automatically be rounded according to the pipe length rounding value.

For example, consider a pipe with an actual scaled length of 35.8 meters. If the pipe length rounding value is 1.0 meters, the program will assume the pipe length to be 36.0 meters.

This only affects the value as it appears in elemental editors, FlexTables, and so on. The actual length of the pipe figure in the drawing pane is not physically adjusted to force the pipe to a rounded length.

A change to the pipe rounding length is not retroactive. Therefore, it will not affect existing pipes unless the user-defined length is toggled on and then off again.

4.3.3 Drawing Options

The **Drawing Options** dialog allows you to specify information regarding the graphical display of elements in the drawing pane, including:

- Drawing Scale
- Annotation Multipliers
- Pipe Text
- Background Drawing
- Symbol Visibility

To access the **Drawing Options** tab, select **Tools\Options** from the pull-down menus.

Drawing Scale

You can set the scale that you want to use as the finished drawing scale for the plan view output. Drawing scale is determined based upon engineering judgement and the destination sheet sizes to be used in the final presentations.

You may choose either Schematic mode or Scaled mode to define the horizontal and vertical distance scales.

- **Schematic** – Pipe lengths are not automatically initialized from their lengths in the drawing pane, but must be manually entered for each pipe.
- **Scaled** – Pipe lengths are determined from the lengths of the pipe elements in the drawing pane.

– **HOR** – Horizontal scale controls the scale of the plan view.

– **VER** – Vertical scale controls the default elevation scale (for use in profiles, for example).

Scaled and schematic mode can be set on a pipe-by-pipe basis. This is useful when scaled mode is preferred, but an exaggerated scale is needed for layout of detailed piping arrangements.

Whether the drawing is set in scaled or schematic mode automatically reflects the setting of the pipe prototype. While in schematic mode, Gravity Pipe Prototypes and Pressure Pipe Prototypes can be assigned a default length. When the drawing mode is scaled, pipe lengths do not need to be initialized from the prototype. Note that switching between scaled and schematic in either the **Project Options** or **Pipe Prototype** dialogs has no effect on existing pipes.

Annotation Multipliers

Annotation multipliers allow you to change the size of symbols, labels, and annotation text relative to the drawing scale. There is not a single annotation size that is going to work well with all projects and scales, so these values should be adjusted based on your judgment and the desired look of the finished drawings.

- **Symbol Size –** The number entered in this field will either increase or decrease the size of your symbols by the factor indicated. For example, a multiplier of 2 would result in the symbols being doubled in size. The program selects a default symbol height that corresponds to 4.0 ft (approximately 1.2 m) in actual-world units, regardless of scale. Unlike text, this symbol height will not automatically change as you modify the drawing scale. So, if you are generating drawings in smaller dimensions you may want to increase the graphical display size of your symbols.

- **Text Height –** The text height multiplier increases or decreases the default size of the labeling 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, would result 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.

- **Annotation Height** – The annotation height multiplier increases or decreases the default size of the element annotation 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, would result in a text height (to scale) of approximately 1.25 m. Likewise, a 1 in = 40 ft scale equates to a text height of around 4.0 ft.

Pipe Text

The **Align Text with Pipes** toggle lets you specify whether you want the pipe labeling and annotations to be parallel to the pipe or horizontal.

Background Drawing

In Stand-Alone mode, a DXF file may be used as a background image for the drawing pane.

- **Show Background –** If the background DXF file is turned off, it will not be read from a disk or displayed in the drawing pane. If the background is not turned off, it will be read from a disk and displayed.

- **DXF Unit –** The DXF drawing unit conversion is used when importing DXF background files, and also when exporting a DXF file from the project. Note that the value in this field governs the DXF file import behavior for DXF files saved in scientific, decimal, or fractional units, but not for DXF files saved in architectural or engineering units.

- **DXF Background Filename** – This field enables you to specify a DXF file to be used as the background for your project. Enter the drive, directory, and file name, or click the **Browse** button to select a file interactively.

DXF file import behavior is governed by specific factors within the DXF file. If a file does not import as you expect, check the options used to generate it carefully. For example, try importing the DXF back into the original program, or into another program that supports the DXF format (such as AutoCAD, MicroStation, etc.). If the DXF does not import into other applications, there may be an invalid or missing header, invalid elements, or other errors.

Symbol Visibility

Symbol visibility allows you to customize the drawing by turning specific layers on or off. Each drawing layer holds a particular type of graphical element, such as labels and annotation. To remove the graphical elements of a particular layer from the drawing view, simply uncheck the associated box.

- **Show Labels** – The label layer holds the labels for all network elements.
- **Show Graphic Annotations –** Graphic annotation includes lines, borders, and text (in Stand-Alone mode only).
- **Show Element Annotations –** Element annotation includes any dynamic annotation that is added to the project (by the Annotation Wizard).
- **Show Source Symbols** – For a water quality analysis, a symbol may be displayed next to nodes that are defined as a Constituent Source.
- **Show Control Symbols** – A symbol may be displayed next to pump, valve, and pipe elements with one or more controls, as defined in the **Controls** tab of the element editors.
- **Show Flow Arrows** – Arrows indicating the flow direction may be displayed after calculations have been run.

4.4 FlexUnits

Unit flexibility is available from almost anywhere within Haestad Methods' software, including elemental dialogs, FlexTables, and the FlexUnits Manager.

4.4.1 Set Field Options

Most dialogs allow you access to FlexUnits, to set such options as the units, rounding, and scientific notation for any field in the dialog.

To set the display options for a unitized attribute:

1. Right-click the field and select **Properties** from the pop-up context menu. The **Set Field Options** dialog will appear.
2. Set the options you want for your units.
3. Click **OK** to set the options for the field, or **Cancel** to leave without making changes.

You will be able to change the following characteristics:

- Units
- Display Precision
- Scientific Notation
- Minimum and Maximum Allowable Values

Some attributes do not have theoretical minimum or maximum values, and others may have an acceptable range governed by calculation restrictions or physical impossibilities. For these attributes, minimum and maximum allowable values may not be applicable.

You can see the results of your changes in the preview at the top of the dialog.

4.4.2 Units

Units are the method of measurement displayed for the attribute. To change units, click the drop-down list, then click the desired unit. The list is not limited to SI or US customary units, allowing you to mix unit system within the same project.

Note that FlexUnits are intelligent – the units actually have meaning. When you change units, the displayed value is converted to the new unit, so the underlying magnitude of the attribute remains the same.

For example, a length of 100.0 feet is not converted to a length of 100.0 m or 100.0 in. It is correctly converted to 30.49 m or 1200.0".

4.4.3 Display Precision

The precision setting can be used to control the number of digits displayed after the decimal point, or the rounding of numbers.

Number of Digits Displayed After Decimal Point

Enter 0 or a positive number to specify the number of digits after the decimal point.

For example, if the display precision is set to 3 then a value of 123.456789 would display as 123.457. This works the same regardless of whether or not scientific notation is active.

Rounding

Enter a negative number to specify rounding to the nearest power of 10; -1 rounds to the nearest 10, -2 rounds to the nearest 100, and so on.

For example, if the display precision is set to -3 then a value of 1,234,567.89 would display as 1,235,000.

Display precision is for numeric formatting only and will not affect calculation accuracy.

4.4.4 Scientific Notation

Scientific notation displays the number as a real number beginning with an integer or real value, followed by the letter "**e**" and an integer (possibly preceded by a sign). Click the field to turn scientific notation on or off. A check will appear in the box to indicate that this setting is turned on.

Scientific Notation is for numeric formatting only and will not affect calculation accuracy.

4.4.5 Minimum and Maximum Allowed Value

Minimum and maximum values are used to control the allowable range for an attribute, and are used for validation of user input. For example, some coefficient values might typically range between 0.09 and 0.20. A frequent user input error is to misplace the decimal point when entering a value. If you enter a number that is less than the minimum allowed value, a warning message will be displayed. This helps reduce the number of input errors.

You may change this number in cases where you find the default limits too restrictive.

These allowable minimums and maximums are only available for certain parameters.

4.4.6 Flex Unit Table

The flexUnits table, which can be accessed by selecting **Tools\FlexUnits**, allows you to set the parameters for all the units used. The dialog consists of the following five columns:

- **Attribute Type** – Attribute measured by the unit.
- **Unit** – Type of measurement displayed. To change the unit of an attribute type, click the drop-down menu, and click the unit you want. This option also allows you to use both US customary and SI units in the same worksheet.
- **System** – Set the system of units. Click the system column for the desired unit, and a button should appear. Click the button, and set the unit system to the US or SI system.
- **Display Precision –** 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 8 to indicate the number of digits after the decimal point. This feature works the same whether scientific notation is on or off.
- **Scientific Notation** – Display numbers in scientific notation. Click the field to turn scientific notation on or off. If it is turned on, a checkmark appears in the box.

Use Defaults – Clicking this button resets all the units to default US customary or metric units, based on the project global unit system. This is specified on the **Global Options** tab, accessed from the **Tools\Options** menu from the main menu.

The display units can also be changed from several other areas in the program, with any changes being project-wide. For example, if length is changed from units of feet to meters, all pipe dialogs will display length in meters. If you change the units in the pipe dialog from meters to yards, the FlexUnits Manager will indicate that length is in yards.

FlexTables have the ability to use localized units, which are maintained separately from the current project settings. This allows you to create reports using units that differ from the currently active ones.

Notes

Chapter 5

Layout and Editing Tools

This chapter describes the various tools that are available to simplify the process of graphically or manually entering network data. These tools allow you to select elements to perform various graphical or editing operations, locate particular elements, review the network for potential connection problems, label or relabel elements, review your data, or define any new type of data.

5.1 Graphical Editor

One of the most powerful features of the graphical editor, both in Stand-Alone and AutoCAD mode, is the ability to create, move, edit, and delete network elements graphically. With these capabilities, modeling becomes a simple point-and-click exercise. The on-line tutorials have step-by-step instructions for performing common tasks in the graphical editor, and Lesson 1 also offers assistance.

5.1.1 Working with Network Elements Within the Graphical Editor

Most network editing tasks can be performed using only your mouse. The pull-down menus and AutoCAD command line also offer the ability to perform many of these tasks, but by simply pointing-and-clicking with the mouse you will be able to:

- Create New Elements
- Select Elements
- Edit Elements
- Move Elements
- Delete Elements
- Annotate the Drawing

As you move your mouse over each element, a tool-tip is displayed informing you of the element's label and annotations.

5.1.2 Creating New Elements

The tool palette contains all of the tools necessary for adding network elements to the drawing. These element tools include:

Pipe Layout Tool – Pipes are link elements that connect junction nodes, pumps, valves, tanks, and reservoirs.

Pressure Junction Tool – Junctions are non-storage nodes where water can leave the network to satisfy consumer demands or enter the network as an inflow. Junctions are also where chemical constituents can enter the network.

Tank Tool – Tanks are a type of storage node. The water surface elevation of a tank will change as water flows into or out of it during an extended period simulation.

Reservoir Tool – Reservoirs are a type of storage node. The water surface elevation of a reservoir does **not** change as water flows into or out of it during an extended period simulation.

Pump Tool – Pumps are elements that add head to the system as water passes through. A Pump is represented as a node.

Valve Tool – Valves are elements that open, throttle, or close to satisfy a condition you specify. A valve is represented as a node.

Spot Elevation Tool – In addition to the elevations at junction nodes and other network elements, supplemental spot elevations can be entered throughout the model without adding unnecessary model nodes.

Although elements can be inserted individually, the most rapid method of network creation is through the **Pipe Layout** tool. The **Pipe Layout** tool enables you to connect existing nodes with new pipes, but also allows you to create new nodes as you lay out the pipes.

For example, when the **Pipe Layout** tool is active, clicking within the drawing pane will insert a node. Clicking again at another location will insert another node and connect the two with a pipe. Use the on-line tutorials to experience it interactively.

5.1.3 Changing the Pipe Layout Tool to Insert a Different Type of Node

While laying out the network, you may need to change the type of node that the pipe layout tool inserts. This can be done very easily and quickly by following the steps outlined below.

1. Right-click in a blank area of the graphical editor.
2. A context menu will pop up with a list of available element types.
3. Select an element type from the context menu.

The cursor appearance will change to reflect the type of node to be inserted.

5.1.4 Morphing Elements

Occasionally, you may find that you need to replace a node with a different type of node. You can change the element's type without deleting it through a process called morphing.

Morphing enables you to change the type of an existing network node, without having to delete and re-create the node and all of its connecting links. Information that is common between the existing and new element will be copied into the new element. To morph an existing element into a different type of element:

1. From the Tool Palette, select the new element type.

2. In the drawing pane, place the cursor over the old element and click.

3. You will be prompted to verify that you want to morph the old element simply as a security measure to ensure that an element is not accidentally morphed. In the event that you do accidentally morph an element, this action can be undone.

5.1.5 Splitting Pipes

You may encounter a situation where you need to add a new node in the middle of an existing pipe. For example, you may need to insert a new inlet to capture excessive surface flow in StormCAD or a new manhole in SewerCAD.

You can split existing pipes simply by inserting a node along the pipe.

1. From the Tool Palette, select the node type.

2. In the Drawing Pane, place the cursor over the pipe and click.

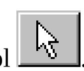
3. You will be prompted to confirm that you wish to split the pipe. If you choose to split the pipe, the node will be inserted and two new pipes will be created with the same characteristics as the original pipe (lengths are split proportionally).

4. If you choose not to split the pipe, the new element will be placed on top of the pipe without connecting to anything.

5.1.6 Selecting Elements

You can select one element or a group of elements in the graphical editor on which you can then perform various operations such as moving, deleting, and editing.

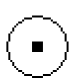
Selecting Elements (Stand-Alone Mode)

In Stand-Alone mode, first activate the Select tool .

To select a single element, simply click it. To select a group of elements, click in the drawing pane and drag the mouse to form a rectangle around the elements you want to select; then click again to choose the other corner of the rectangle. All elements that are fully enclosed within the selection rectangle will be selected.

To select or deselect other elements or groups of elements, you can follow the same instructions as above, while holding down the **Shift** key. Note that there are also many other ways to select elements through the **Edit** menu.

When an element is selected in the Stand-Alone drawing pane, it will be displayed with at least one grip. A grip is a black box, as shown below, that indicates the figure's insertion point. The label of a selected item, or the number of selected items, will be displayed in the status bar.

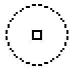


Selecting Elements (AutoCAD Mode)

Within AutoCAD, the Select tool does not need to be active when making a selection. In fact, no tools need to be active. Many of the standard Windows selection techniques can be used in AutoCAD, similar to the selection method in Stand-Alone mode.

AutoCAD also offers a variety of other selection methods, which are outlined in AutoDesk's documentation.

When an element is selected in AutoCAD, it may be displayed in a dashed linetype and the grips may become visible, as shown below. The exact display depends on how the element was selected, and the value of the AutoCAD variable – GRIPS.



5.1.7 Editing Elements

Not all network elements have the same input data. There are several methods for editing network elements' data, including Database Connections, FlexTables, and the Alternative Manager.

Perhaps the most common method of changing element data, however, is from an individual element's editor. To edit a single element, use the **Selection** tool, in Stand-Alone mode or in AutoCAD mode.

In Stand-Alone mode, editing a single element is very easy. Simply double-click the element, and the editor dialog will open. Alternatively, you can right-click the element and select **Edit…** from the pop-up context menu.

In AutoCAD mode, the process is slightly different but still very easy. With the Select tool active, simply click the element. If you have **Right Mouse Context Menu** checked in the **Global Options** dialog, you can also right-click to bring up the pop-up context menu.

Right-click context menus can provide easy access to common functions and actions.

5.1.8 Moving Elements

You can change the location of elements easily, whether you are in Stand-Alone or AutoCAD mode.

The first step is to select the element(s) to be moved. With grips displayed, simply drag a grip and drop it at the new location. When a node is moved to a new location, pipe lengths are automatically updated according to the set scale.

In AutoCAD this is the equivalent of the STRETCH command, not the MOVE command. There are also several other methods of moving items within AutoCAD. For more information regarding moving elements within AutoCAD, please refer to your AutoDesk documentation.

In the same fashion, you can also graphically change the location of element labels and annotation relative to the element.

Additionally, a node element can be moved by editing its coordinates in the element's editor, in FlexTables, or through database connections.

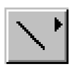
5.1.9 Deleting Elements

Deleting elements is quite easy. Simply select the element(s) to be deleted, and press the **Delete** key on the keyboard. Note that the integrity of the network is automatically maintained when deletions are performed. This means that when a node is deleted, any connecting pipes are also deleted to prevent "dangling" pipes that would otherwise cause the network to be invalid.

There are also several other methods of deleting elements, including selecting **Edit\Delete** from the pull-down menu, or typing **ERASE** at AutoCAD's command line.

5.1.10 Other Tools

Although this product 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.



In Stand-Alone mode, drafting and annotation tools allow you to add polylines (multi-segmented lines), rectangles, and text to the drawing pane.

Line Enclosed Area

In the Stand-Alone drawing pane, you can calculate the enclosed area of any closed polyline. This feature can be especially helpful for determining catchment areas or residential areas.

Simply right-click the closed polyline, and select **Enclosed Area** from the pop-up context menu. The **Area** dialog will open, displaying the calculated area of the polyline enclosure.

This tool is only available for closed polylines. To close an open-ended polyline, right-click it and select Close from the context menu.

Although this feature is not provided in AutoCAD mode, you can determine the area of any AutoCAD polyline by performing a **LIST** command on it.

5.2 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. Selection sets are defined through the Selection Set Manager in the **Tools** menu.

5.2.1 Selection Set Manager

The Selection Set Manager is used to create, edit, and duplicate selection sets.

- **Add –** Add a new selection set.
- **Edit** – Edit an existing selection set.
- **Duplicate** – Copy an existing selection set.
- **Delete** – Delete an existing selection set.
- **Rename** – Rename an existing selection set.
- **Notes** – Add a note regarding the selection set.

New Selection Set

After clicking **Add** in the **Selection Set Manager**, a dialog appears. Simply enter the name of your new selection set in the dialog. Click **OK** to name the selection set or **Cancel** to exit the dialog without creating a selection set.

Selection Set Dialog

In this dialog, you will notice two panes. A listing of all the elements in the network is displayed in the **Available Items** pane. To add items to the **Selected Items** pane, select elements using the mouse and click the **[>]** button under **Add**. To add all the elements to your selection set, click the **[>>]** button.

Additionally, you can click the **Select** button to highlight those items in the **Available Items** pane using sorts, filters, or the actual drawing itself. It will also allow you to invert the selection set, thereby unselecting the ones already selected and selecting the ones not already selected. You can also clear the selected items using the **Select** button.

The features mentioned above are also available to remove items from the **Selected Items** pane.

Duplicate Selection Set

Allows you to make a copy of the highlighted selection set.

Rename Selection Set

This dialog allows you to change the name of the highlighted selection set.

Selection Set Notes

The memo field on the **Notes** dialog allows you to input free-form paragraph text that will be associated with the highlighted selection set. Use it to write detailed notes.

5.3 Find Element

This is a powerful feature that allows you to quickly locate any element in the drawing by its label alone. It performs a case *insensitive* search. The **Find Element** feature is available from the **Edit** menu on the main window.

To find an element:

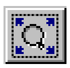
1. Choose **Edit\Find Element**.
2. Type the label of the element you wish to find, or you can click in the list box to choose from a sorted list of elements in the system.
3. You may wish to choose a *Zoom Factor* from the list provided. 100% is the default zoom factor. If you wish to magnify the view of the drawing, then choose a zoom factor greater than 100%. To decrease the view of the drawing, choose a zoom factor less than 100%.
4. Click the **OK** button.

5.4 Zooming

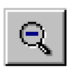
Zooming controls how large or small a drawing appears on the screen. Zooming is helpful when you want to enlarge the display to see the drawing's details, or to reduce the display to see an entire drawing. Zooming does not change the actual size of the drawing; only the size of the current view.

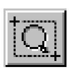
You can zoom by doing one of the following:

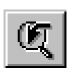
1. From the **View** pull-down menu choose:
   - **Zoom In** – Enlarge the view of the drawing.
   - **Zoom Out** – Decrease the view of the drawing.
   - **Zoom Window** – Choose the portion of the drawing to fit in the window by drawing a box around it.
   - **Zoom Extents** – Bring all elements in the drawing into view.
   - **Zoom Previous –** Return to the most recent view of the drawing.
   - **Zoom Center –** Center the location of specific coordinates within the drawing pane.
2. Use the toolbar buttons:

 Zoom Extents

 Zoom In

 Zoom Out

 Zoom Window

 Zoom Previous

You can use the *Plus* key (+) and the *Minus* key (-) on the numeric keyboard as a shortcut for zooming in and out respectively (only in Stand-Alone mode).

The previous zooms are maintained in a stack, so it is possible to zoom and pan to different portions of the drawing in sequence, then recall those zooms in reverse sequence.

5.4.1 Zoom Center

The **Zoom Center** dialog provides you with a quick way to zoom to any area of your drawing. This feature is useful if you want to start laying out a network around a certain coordinate, or if you know the coordinate of an existing element that you would like to locate.

To use Zoom Center:

1. Select the **View\Zoom Center** menu item.
2. In the **Zoom Center** dialog, enter the coordinate to which you would like to zoom.
3. Select a zoom factor if you would like to increase or decrease the magnification.
4. Click **OK,** and the specified coordinate will be located at the center of the drawing.

5.4.2 Aerial View

This feature displays a second view of the drawing at a larger scale in a separate window, in order to facilitate zooming, panning and locating a small viewing area in the main drawing window. It is enabled and disabled from the **View** menu.

In Stand-Alone mode:

With **Aerial View** window enabled (by selecting the **View\Aerial View** menu item), 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 Extent –** 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.

To resize the view box directly in the **Aerial View** window, simply draw a new rectangular view box. To change the location of the view box directly in the **Aerial View** window, you can either drag the view box frame, or create a new view box.

In AutoCAD mode:

Refer to the AutoCAD on-line Help for a detailed explanation.

5.5 Drawing Review

The **Drawing Review** window allows you to quickly review and navigate to any group of elements. This tool is particularly useful for finding potential problems in a network. These problems may result from data entry errors or data discrepancies in the source (database, Shapefile, or CAD drawing) from which a model was imported.

By default, when the **Drawing Review** window opens, all elements will appear in the list. You can work with any subset of elements choosing one of the following items:

- **Select\Custom –** This option allows you to choose any set of elements to review using the **Selection Set** dialog.
- **Select\All Elements** – This option automatically selects all available elements.
- **Select\Nodes in Close Proximity –** This option allows you to select all nodes that are within a user-defined tolerance of a pipe, but are not connected to that pipe. The tolerance is defined in the **Nodes in Close Proximity** dialog, which opens when this option is selected. This tool is useful for finding and correcting connectivity problems.
- **Select\Pipe-Split Candidates –** This option allows you to find nodes that are closer to a pipe than a user-defined tolerance but are not connected. The tolerance is defined in the **Pipe-Split Candidates** dialog, which opens when this option is selected. This option is useful for finding and correcting connectivity problems.
- **Select\Orphaned Nodes** – This option allows you to select all orphaned nodes in your network. A node is an orphan when it is not connected to any pipe.
- **Select\Elements with Messages –** This option allows you to select all the elements that have warnings or error messages which appear in the **Messages** tab of an element editor dialog. This is useful for correcting data entry errors.

- **Select\Clear Drawing Review Messages** – This option allows you to reset *Drawing Review* messages for all elements in the list. Drawing Review messages are automatically added during various *Import* operations such as Polyline to Pipe Import, Land Development Desktop Import (in SewerCAD or StormCAD), etc. After you review and fix these problems, you may want to clear the review messages. If you want to retain some of the drawing review messages, simply remove those elements from the list prior to performing this operation.

The elements you select will appear in the primary list located along the left side of the **Drawing Review** window.

- **Go To** – To navigate to an element, select the desired element in the list and press the **Go To** button.
- **Next/Prev –** To navigate to the elements sequentially, use the **Next** or **Prev** buttons.
- **Zoom** – You can control the degree to which the drawing review zooms into the selected element by choosing a zoom factor from the field labeled **Zoom**, located in the lower right corner of the dialog.

> **You can double-click an element in the list to quickly navigate to that element.**
>
> **If you know the name of the element to which you wish to navigate, type the label in the field located above the element list and click the Go To button.**
>
> **The Drawing Review window is modeless, so all menus and toolbars will remain available even when the window is open. This will allow you to navigate to and fix any problems that you find.**

Use the **Drawing Review** window in conjunction with the **Quick View** window to review the data for the selected elements.

To access the **Drawing Review** dialog select **Edit\Review Drawing** from the main menu.

5.5.1 Selection Tolerance

Some *select* operations require you to specify a tolerance for defining which nodes will be selected for the **Drawing Review** window.

- **Elements in Close proximity** – If the distance between the elements in the drawing is within the specified tolerance, those elements will be selected for display in the Drawing Review window.
- **Pipe Split Candidates –** If the distance between a node and a pipe is within the specified tolerance, it will be selected for display in the Drawing Review window.

5.6 Relabel Elements

5.6.1 Relabel Elements Dialog

Element relabeling allows you to modify the labels of a selected set of elements. An instance where this is especially useful is with a model built from a database that uses numeric ID's to identify elements. The element labels will be numeric identifiers when they are imported through a database connection, making it difficult to distinguish between the different types of elements in the system. With the element relabeling tool you can quickly append a prefix such as 'P-' to all the pipes in your system so that it is obvious which labels belong to elements representing pipes.

The **Relabel Elements** dialog contains two sections:

- **Relabel Operations** – Allows you to select and define the operations you want to perform.
- **Elements Selected –** Allows you to select which elements in your project you want to relabel.

To access the **Relabel Element** dialog select **Tools\Relabel Elements**… from the main menu.

5.6.2 Relabel Operations

The element relabeling tool allows you to perform three types of operations on a set of element labels: Append, Replace, and Renumber. The active relabel operation is chosen from the list box in the **Relabel Operations** section of the **Relabel Elements** dialog. 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 P-1, P-2, P-12, and J-5, you could to replace all the P's with the word Pipe by entering 'P' in the **Find** field, 'Pipe' in the **Replace With** field and clicking the **Apply** button. The resulting labels would be Pipe-1, Pipe-2, Pipe-12, and J-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 'ipe' in the **Find** field and leave the **Replace With** field blank to reproduce the labels P-1, P-2, P-12, and J-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 following labels P-1, P-4, P-10, and Pipe-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 'P-' 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 ID number for the first element relabeled, five, is entered in the **Next** field. The value by which the numeric base of each consecutive element is incremented, five, is entered in the **Increment** field. The minimum number of digits in the ID number, two, is entered in the **Digits** field. If the number of digits in the ID number is less then this value it is padded with zeros. Click the **Apply** button to produce the following labels P-05-Z1, P-10-Z1, P-15-Z1, and P-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 pipes in zone 1 of your system. You can use the append operation to add an appropriate prefix and suffix, such as 'P-' and '-Z1', by specifying these values in the **Prefix** and **Suffix** fields and clicking the **Apply** button. Performing this operation would yield the labels P-5-Z1, P-10-Z1, P-15-Z1, and P-20-Z1. You can append only a prefix or suffix by leaving one of the entry fields empty. However, in order for the operation to be valid, one of the entry fields must be filled in.

The selection of elements on which the relabel operation is to be performed can be selected in the **Elements** section of the **Relabel Elements** dialog.

To access the **Relabel Elements** dialog, select **Tools\Relabel Elements**… from the main menu.

5.6.3 Elements Selected

The **Elements** section contains a text pane that lists the elements to be relabeled. You can select the set of elements that appears in this text pane by clicking the **Select** button. This accesses the

Selection Set dialog, where you can pick a set of elements from all the elements currently in the project.

For the Append and Replace operations, the order that the elements appear in the text pane does not affect the results of the operation. However, for the Renumber operation, the order in which the elements appear in the text pane determines the order in which they will be renumbered. The default order in which the elements appear in the text pane is in the alphanumeric order of the element labels called ascending order. If you wish to change this order you can click the **Sort** button and select **Network Order** to put the elements in the order they appear in the network, **Descending Order** to put them in reverse alphanumeric order, or **Ascending Order** to put them back in alphanumeric order.

5.7 Element Labeling

The **Element Labeling** dialog is used to specify the automatic numbering format of new elements as they are added to the network.

- **Element** – A non-editable field, indicating the type of element the label applies to.
- **Next** – Enter the integer you want to use as the starting value for the ID number portion of the label. The program will generate labels beginning with this number, and will choose the first available unique label.
- **Increment** – Enter the integer that will be added to the ID number after each element is created to yield the number for the next element.
- **Prefix** – Enter the letters or numbers that will appear in front of the ID number for the elements in your network.
- **Digits** – Enter the total number of digits the ID number will have.
- **Suffix** – Enter the letters or numbers that will appear after the ID number for the elements in your network.
- **Preview** – An example of what the label will look like based on the information you have entered in the previous fields.

Changes to the element labeling specifications will only affect the numbering of **new** elements. Existing elements will not be affected. In order to adjust the numbering of existing elements, utilize the **Relabel Elements** option accessible from the **Tools** menu.

Pipe labeling can be aligned with the pipes or be displayed horizontally, depending on the Pipe Text setting specified in the Drawing Options dialog.

You can control the angle at which the text flips from one side of the pipe to the other (reading in the opposite direction) to maintain readability, when the pipe direction on a plot is nearly vertical. By default, the text flips direction when the pipe direction is 1.5 degrees, measured counter-clockwise from the vertical. You may modify this value by inserting a **TextFlipAngle** variable in the Haestad.ini file located in your \Haestad directory, under the [WTRC] section and specifying the angle at which the text should flip. That angle is measured in degrees counter-clockwise from the vertical. For instance, if you want the text to flip when the pipe direction is vertical, you should add the following line to the Haestad.ini file:

TextFlipAngle=0

Reasonable values would fall in the range 15.0 deg to -15.0 deg.

5.8 Quick View

The **Quick View** window provides you with a fast way to view the data associated with any element in the network without having to open the element dialog. It is a floating window that includes input and output information for any element that you have selected. It also includes a convenient color coding legend. Three tabs are provided on the window:

- **Input –** Input data for the selected element.
- **Output –** Output data for the selected element.
- **Legend –** Ranges of the active color coding.

When the **Quick View** window is active, the data for a selected entity will immediately be displayed.

You can change the size of the Label, Value, and Unit columns on the Input/Output tabs by using the resizing bar at the top of the Quick View window.

You can highlight an Input or Output attribute (e.g. Demand), by clicking the label of that attribute in the Quick View window. This provides for better visual feedback, for example, when monitoring the pressures at several nodes.

Use the Quick View button  in the toolbar to toggle the **Quick View** window on and off.

Chapter 6

Hydraulic Element Editors

This chapter presents a detailed look at the input and output data for each type of element used in a WaterCAD project and the way it is organized in the graphical user interface. First, a description of the elements used to model the water distribution network is provided, including prototypes as a way to initialize new model elements with default values. Then the chapter addresses the user data extension, which allows you to add your own attributes to any element and the zone manager that allows you to group modeling elements into zones, used for fire flow analysis.

The primary component of a WaterCAD project is the network model. The element types that are used to form a network are:

- **Pressure Pipes –** Pipes are link elements that connect junction nodes, pumps, valves, tanks, and reservoirs to each other. The only way for water to travel from one node to another is by following a path through one or more pipes.
- **Pressure Junctions** – Junctions are non-storage nodes where water can leave the network to satisfy consumer demands, water can enter the network as an inflow, or chemical constituents can enter the network.
- **Tanks –** Tanks are a type of storage node. The water surface elevation of a tank will change as water flows into or out of it during an extended period simulation. Tanks can have either a circular or non-circular cross section.
- **Reservoirs** – A reservoir is a type of storage node. The water surface elevation of a reservoir does **not** change as water flows into or out of it during an extended period simulation. Reservoirs can be used to model external water sources such as lakes, streams, and wells.
- **Pumps** – A pump is an element which adds head to the system as water passes through. It is typically defined by a pump curve and control elevations, which turn the pump on or off. It is represented as a node.
- **Valves** – A valve is an element that opens, throttles, or closes to satisfy a condition you specify. It is represented as a node.

A water distribution model will not be considered valid for calculation if the number of pipes exceeds the licensed size. To determine how many pipes you are licensed for, choose the Help\About WaterCAD menu item. Click the Registration button to view the size that has been licensed. If the total number of pipes exceeds the licensed size, the project will not calculate.

6.1 Element Editors

The element editors allow you to edit all input data and view all output data defining a single network element.

> **Element data may also be viewed/edited more efficiently through FlexTables, which display all the data in customizable tabular format, allowing you to perform sorting, filtering, global editing, etc. The data may also be quickly reviewed through the Quick View window.**

To access an element editor, follow the directions below for the mode you are using:

Stand-Alone:	Double-click the element you wish to edit, or right-click the element and select **Edit** from the drop-down menu.
AutoCAD R14:	Pick the Select tool and click the element you wish to edit. If the **Right-Click Context Menu** is selected, you can also right-click the element and select **Edit** from the drop-down menu.
AutoCAD 2000:	Pick the Select tool and click the element you wish to edit, or select the element and choose **Edit** from the drop-down menu.

6.1.1 Pressure Pipe Editor

Pipes are link elements that connect junction nodes, pumps, valves, tanks, and reservoirs to each other. The only way for water to travel from one node to another is by following a path through one or more pipes. The pressure pipe editor organizes the related input data and calculated results into the following tabs:

- **General** – General pipe information including dimension and physical characteristics data, as well as hydraulic results.
- **Controls –** Control data used to specify whether the pipe is open or closed at a specified time or based on the HGL or pressure at any node in the system.
- **Quality –** Input parameters used when performing a Water Quality Analysis as specified in the **Scenario Calculation** dialog.
- **Cost** – Cost Analysis input/output data used when performing Cost Analysis calculations.
- **User Data** – Additional data as defined by the user. New fields can be added, such as the pipe installation date or the pipe condition.
- **Messages –** Calculation messages, such as warnings or error messages, and user-entered notes and descriptions.

For more information on the data, refer to the topics describing each tab.

6.1.2 Pressure Junction Editor

Junctions are non-storage nodes where water can leave the network to satisfy consumer demands, water can enter the network as an inflow, or chemical constituents can enter the network. The pressure junction editor organizes the related input data and calculated results into the following tabs:

- **General –** General junction information including geographical data and hydraulic results.

- **Demand** – Assignment of demands or inflows to junction elements in order to simulate water leaving or entering the network. Inflows and demands consist of a baseline flow rate and an associated Fixed or Extended Period Simulation (EPS) Pattern.
- **Quality** – Input parameters used when performing a Water Quality Analysis, as specified in the **Scenario Calculation** dialog.
- **Fire Flow** – Contains fire flow input and output data.
- **Cost** – Cost Analysis input/output data used when performing Cost Analysis calculations.
- **User Data** – Additional data as defined by the user. New fields can be added
- **Messages** – Calculation messages, such as warnings or error messages, and user-defined notes and descriptions.

For more information on the data, refer to the topics describing each tab.

6.1.3 Tank Editor

Tanks are a type of storage node. The water surface elevation of a tank will change as water flows into or out of it during an extended period simulation. Tanks can have either a circular or non-circular cross section. WaterCAD allows you to define tanks with either fixed or variable sections.

For steady-state simulations, a tank is considered to have a constant water surface elevation, similar to a reservoir.

The tank editor organizes the related input data and calculated results into the following tabs:

- **General** – General tank information including geographical data and hydraulic results.
- **Section** – Data defining the geometric characteristics of the tank and its operating level range.
- **Quality** – Input parameters used when performing a Water Quality Analysis, as specified in the **Scenario Calculation** dialog.
- **Cost** – Cost Analysis input/output data used when performing Cost Analysis calculations.
- **User Data** – Additional data as defined by the user. New fields can be added, such as the tank installation date or the tank condition.
- **Messages** – Calculation messages, such as warnings or error messages, and user-defined notes and descriptions.

For more information on the data, refer to the topics describing each tab.

6.1.4 Reservoir Editor

A reservoir is a type of storage node. The water surface elevation of a reservoir does **not** change as water flows into or out of it during an extended period simulation. Reservoirs can be used to model external water sources such as lakes, streams, and wells. The reservoir editor organizes the related input data and calculated results into the following tabs:

- **General** – General reservoir information including geographical data and hydraulic results.
- **Quality** – Input parameters used when performing a Water Quality Analysis, as specified in the **Scenario Calculation** dialog.
- **Cost** – Cost Analysis input/output data used when performing Cost Analysis calculations.

- **User Data –** Additional data as defined by the user. New fields can be added.
- **Messages –** Calculation messages, such as warnings or error messages, and user-defined notes and descriptions.

For more information on the data, refer to the topics describing each tab.

6.1.5 Pump Editor

A pump is an element that adds head to the system as water passes through it. This software can currently be used to model six different pump types:

- Constant Power
- Design Point (One-Point)
- Standard (Three-Point)
- Standard Extended
- Custom Extended
- Multiple Point

Avoid using constant power or design point pumps. They are often enticing because they require less work on behalf of the engineer, but they are much less accurate than a pump curve based on several representative points.

It is not necessary to place a check valve on the pipe immediately downstream of a pump, because pumps have built in check valves that prevent reverse flow.

The pump editor organizes the related input data and calculated results into the following tabs:

- **General** – General pump information including geographical data, pump curve data, initial settings, and hydraulic results.
- **Controls** – Data specifying the on/off elevation settings of the pump, as well as relative speed factor settings in the case of a variable speed pump.
- **Quality –** Input parameters used when performing a Water Quality Analysis as specified in the **Scenario Calculation** dialog.
- **Cost –** Cost Analysis input/output data used when performing Cost Analysis calculations.
- **User Data** – Additional data as defined by the user. New fields can be added.
- **Messages** – Calculation messages, such as warnings or error messages, and user-defined notes and descriptions.

For more information on the data, refer to the topics describing each tab.

6.1.6 Valve Editor

A valve is an element that opens, throttles, or closes to satisfy a condition you specify. This software can model several different types of valves. The behavior of a valve is determined by the upstream (*From Pipe*) and downstream (*To Pipe*) conditions. The valve types include:

- **Pressure Reducer Valve (PRV) –** PRVs throttle to prevent the downstream hydraulic grade from exceeding a set value. If the downstream grade rises above the set value, the PRV will close. If the head upstream is lower than the valve setting, the valve will open fully.
- **Pressure Sustaining Valve (PSV) –** PSVs throttle to prevent the upstream hydraulic grade from dropping below a set value. If the upstream grade is lower than the set grade, the valve will close completely.
- **Pressure Breaker Valve (PBV)** – PBVs are used to force a specified pressure (head) drop across the valve. These valves do not automatically check flow, and will actually boost the pressure in the direction of reverse flow to achieve a downstream grade that is lower than the upstream grade by a set amount.
- **Flow Control Valve (FCV) –** FCVs are used to limit the maximum flow rate through the valve from upstream to downstream. FCVs do not limit the minimum flow rate or negative flow rate (flow from the *To Pipe* to the *From Pipe*).
- **Throttle Control Valve (TCV) –** TCVs are used as controlled minor losses. A TCV is simply a valve that has a minor loss associated with it, where the minor loss can change in magnitude according to the controls that are implemented for the valve.

You can change a valve from one type to another by a process called "morphing". Just click the new valve type on the toolbar, and drag the new valve on top of the old one.

If you are using a valve that does not normally check flow, but you would like it to, simply set one of the pipes connecting to the valve with a check valve.

The valve editor organizes the related input data and calculated results into the following tabs:

- **General** – General valve information including geographical data and hydraulic results.
- **Controls** – Data specifying how the valve is controlled, as a function of the time or the hydraulic condition at any node in the system.
- **Quality** – Input parameters used when performing a Water Quality Analysis, as specified in the **Scenario Calculation** dialog.
- **Cost** – Cost Analysis input/output data used when performing Cost Analysis calculations.
- **User Data –** Additional data as defined by the user. New fields can be added.
- **Messages –** Calculation messages, such as warnings or error messages, and user-defined notes and descriptions.

For more information on the data, refer to the topics describing each tab.

6.2 Element Editors' Tabs

6.2.1 General Tab

Pressure Pipes General Tab

The **General** tab for pressure pipes is organized into the following groups:

- **Pipe** – General pipe data.
- **Initial Status** – Specify whether the pipe is initially open or closed.

- **User-Defined Length** – Specify whether the pipe length is calculated automatically or user-defined.
- **Nodes** – Define a positive direction for the flow in the pipe. This is used for check valves or flow results. A reported negative flow indicates that the water is flowing from the *To Node* to the *From Node*.
- **Hydraulic Results** – Calculated hydraulic data.
- **Water Quality** – Results of the water quality computations in the pipe, reported when a Water Quality Analysis has been performed.

For more information on the data, see the topic on each data section below.

Pipe Section

In this section you enter all of the pipe general characteristics:

- **Label** – Unique "name" referencing the pipe in reports, error messages, and tables.
- **Material** – Pipe material, with its associated roughness value, selected from the material library.
- **Diameter** – Diameter of the pipe.
- **Roughness Coefficient** – Pipe roughness coefficient or value associated with the roughness method selected during the project setup (Manning's n, Hazen-Williams C, or Darcy-Weisbach roughness height) for the selected material. You can keep the roughness value associated with the selected material, as defined in the material library, or override the roughness value for that specific pipe.
- **Minor Loss Coefficient** – Coefficient K used in the minor loss equation, as defined in the Minor Loss section in the "WaterCAD Theory" Appendix. This is the equation most commonly used for determining the headloss in a fitting, valve, meter, or other localized component.
- **Check Valve** – When this box is checked, flow can only travel from the *From Node* to the *To Node* in a pressure pipe.

By clicking the ellipsis (...) button located next to the Material or the Minor Loss Coefficient field, you can access the engineering library to create and customize materials and composite minor loss coefficients, respectively.

Set the minor loss coefficient value to 0.0 if there is no minor loss in the pipe.

Composite Minor Loss

Pressure pipes can have an unlimited number of minor loss elements associated with them. This program provides an easy-to-use table for editing these minor losses. The composite minor loss table consists of four columns: **Quantity, Minor Loss**, **K Each**, and **K Total**. Each row represents a group of similar minor losses:

- **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.
- **K Each** – The headloss coefficient for a single minor loss element of the specified type.

- **K Total** – The total minor loss coefficient for the row. It is the **Quantity** multiplied by **K Each**.

The composite minor loss dialog also has three command buttons:

- **Insert** – Insert a row in the table.
- **Duplicate** – Create a new row in the table with the same values as the selected row.
- **Delete** – Delete the selected row of the table.

Initial Status Section

The Initial Status of the pipe can be either Open or Closed. The status can possibly change when calculations are performed based on the presence of controls for that pipe.

> **In Steady-State Analysis mode (in WaterCAD), the Initial Status is used as the permanent status. However, it can be overruled by the presence of controls, if the Use Controls in Steady-State Analysis check box in the Calculation Options dialog is checked. The Calculation Options dialog is accessed by clicking the GO button in the main view to display the Calculation tab of the Scenario Editor, and then clicking the Options button.**

User-Defined Length Section

If the **User-Defined Length** box is checked, you can enter a pipe length. Otherwise, the program will compute a pipe length from node center to node center, accounting for bends if there are any. Creating user-defined lengths is useful for drawing quick schematics to accelerate your design process.

Nodes Section

This section allows you to identify the calculated flow direction. A reported positive flow value indicates that the flow is in the direction of the *From Node* to the *To Node*. It is also useful for check valves, which allow flow only in the *From Node* to *To Node* direction.

The **Reverse** button allows you to change the direction of a pipe, switching the *From Node* and the *To Node*.

Hydraulic Results Section

This section reports the following hydraulic results:

- **Discharge** – Calculated total flow in the pipe.
- **Velocity** – Calculated velocity in the pipe.
- **Headloss Gradient** – Headloss in the pipe represented as a slope, or gradient.
- **Pressure Pipe Headloss** – Loss of energy in the pipe due to friction and minor losses.
- **Control Status** – *Open* or *Closed* status of the pipe. *Open* means that flow occurs in the pipe and *Closed* means that there is no flow.

Water Quality Section

This section reports the results of the water quality computations at this location, assuming that a Water Quality Analysis was performed. The water quality parameter displayed depends on the type of water quality analysis being performed. This parameter is one of the three types:

- **Age** – Report how long the water has been in the system at this node or link.

- **Trace –** Report the percentage of water at this node or link that originated at another chosen node (tank, reservoir, or junction).
- **Constituent –** Report the concentration of a given constituent at this node or link.

Pressure Junctions General Tab

The **General** tab for junctions is organized into the following sections:

- **General** – General information about the junction.
- **Calculated Hydraulics** – Calculated demand, hydraulic grade and pressure at the junction.
- **Water Quality –** Result of the water quality computations at this node reported when a Water Quality Analysis has been performed.

For more information on the data, see the topic on each section. The **Water Quality** section is identical for all elements and is described in the "Pressure Pipe General Tab" section.

Pressure Junction General Section

This section allows you to enter general information about the junction, such as:

- **Label** – Unique "name" referencing the junction in reports, error messages, and tables.
- **X** (Easting) – The location of the junction may be represented by an X-value or an Easting value, depending on individual preferences.
- **Y** (Northing) **–** The location of the junction may be represented by a Y-value or an Northing value, depending on individual preferences.
- **Elevation** – Elevation of the junction.
- **Zone –** Specify the zone the junction belongs to. You may click the **ellipsis (…)** button to access the Zone Manager, which allows you to edit or add zones.

Junction Calculated Hydraulics Section

This section reports the following results:

- **Demand (Calculated) –** Total demand leaving (or entering) the pipe network at the junction at the current time.
- **Calculated Hydraulic Grade –** Hydraulic grade at the junction.
- **Pressure** – Pressure at the junction.

Tanks General Tab

The **General** tab for tanks is organized into the following sections:

- **General** – General geographic information about the tank.
- **Hydraulics** – Calculated flow entering/leaving the tank and the calculated hydraulic grade in the tank.
- **Water Quality –** Result of the water quality computations at this node reported when a Water Quality Analysis has been performed.

For more information on the data, see the topic on each section. The **Water Quality** section is identical for all elements and is described in the "Pressure Pipe General Tab" section.

General Section

This section allows you to enter general information about the tank such as:

- **Label** – Unique "name" referencing the tank in reports, error messages, and tables.
- **X** (Easting) – The location of the tank may be represented by an X-value or an Easting value, depending on individual preferences.
- **Y** (Northing) – The location of the tank may be represented by a Y-value or an Northing value, depending on individual preferences.
- **Elevation** – Ground elevation of the tank.
- **Zone –** Specify the zone the tank belongs to. You may click the **ellipsis (…)** button to access the Zone Manager, which allows you to edit or add zones.

Hydraulics Section

This section reports the hydraulic data of the tank:

- **Hydraulic Grade** – Calculated hydraulic grade in the tank.
- **Inflow/Outflow** – Flow entering/leaving the tank (the field label changes accordingly).

Reservoirs General Tab

The **General** tab for reservoirs is organized into the following sections:

- **General –** General geographic information about the reservoir.
- **Reservoir Calculated Hydraulics –** Calculated flow entering or leaving the reservoir.
- **Water Quality –** Result of the water quality computations at this node reported when a Water Quality Analysis has been performed.

For more information on the data, see the topic on each section. The **Water Quality** section is identical for all elements and is described in the "Pressure Pipe General Tab" section.

General Section

This section allows you to enter general information about the reservoir, such as:

- **Label** – Unique "name" referencing the reservoir in reports, error messages, and tables.
- **X** (Easting) **–** The location of the reservoir may be represented by an X-value or an Easting value, depending on individual preferences.
- **Y** (Northing) **–** The location of the reservoir may be represented by a Y-value or a Northing value, depending on individual preferences.
- **Elevation** – Elevation of the water surface in the reservoir, which is assumed to remain constant through time.
- **Zone –** Specify the zone in which the reservoir belongs. Click the **ellipsis (…)** button to access the Zone Manager, which allows you to edit or add zones.

Reservoir Calculated Hydraulics Section

This section reports the hydraulic data of the reservoir:

- **Inflow/Outflow –** Flow entering/leaving the reservoir (the field label changes accordingly).

Pumps General Tab

The **General** tab for pumps is organized into the following groups:

- **General –** General data about the pump.
- **Pump –** Type of pump curve and related data.
- **Initial Setting –** Initial conditions for a pump describing the pump's behavior at the start of the analysis in EPS mode, or its permanent setting in Steady-State mode.
- **Pipes** – Direction the pump is operating (i.e. from upstream to downstream node). The direction of pumping can be reversed by clicking the **Reverse** button.
- **Operating Point** – Values of pump head and discharge, which are computed by the program to balance with the remaining system heads and flow rates.
- **Water Quality** – Result of the water quality computations at this pump reported when a Water Quality Analysis has been performed.

For more information on the data, see the topic on each section. The **Water Quality** section is identical for all elements and is described in the "Pressure Pipe General Tab" section.

General Section

This section allows you to enter general information about the pump such as:

- **Label –** Unique "name" referencing the pump in reports, error messages, and tables.
- **X** (Easting) – The location of the pump may be represented by an X-value or an Easting value, depending on individual preferences.
- **Y** (Northing) – The location of the pump may be represented by a Y-value or an Northing value, depending on individual preferences.
- **Elevation** – Elevation of the pump.

Pump Section

The information required for the pump varies depending on the type of pump which was selected. The possible information presented is as follows:

- **Pump Type –** Select one of the six available types of pump curve definitions.
- **Pump Power –** Represents the horsepower that is actually transferred from the pump into the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.
- **Shutoff Point** – Point at which the pump will have zero discharge. It is typically the maximum head point on a pump curve.
- **Design Point –** Point at which the pump was originally intended to operate. It is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
- **Maximum Operating (Point) –** Highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.
- **Maximum Extended (Point) –** Absolute maximum discharge at which the pump can operate, adding zero head to the system. This value may be computed by the program, or entered as a custom extended point.

All defined pump curve points have an associated head and discharge.

Initial Setting Section

The initial conditions for a pump describe the pump's behavior at the start of the analysis. These conditions include:

- **Status** – One of two available status conditions: On (normal operation), Off (no flow under any condition).
- **Relative Speed Factor** – Characteristics of the pump relative to the speed for which the pump curve was entered, in accordance with the affinity laws. A speed factor of 1.00 would indicate pump characteristics identical to those of the original pump curve.

In Steady-State Analysis mode (in WaterCAD), the Initial Status is used as the permanent status. However, it can be overruled by the presence of controls, if the Use Controls in Steady-State Analysis check box in the Calculation Options dialog is checked. The Calculation Options dialog is accessed by clicking the GO button in the main view to display the Calculation tab of the Scenario Editor, and then clicking the Options button.

Pipes Section

This indicates the direction in which the pump is operating (from upstream node to downstream node).

You can switch the Upstream and Downstream Pipes by clicking the Reverse button.

Operating Point Section

The pump's operating point represents the values for pump head and discharge which are computed by the program to balance with the remaining system heads and flow rates.

The calculated parameters are:

- **Pump Head** – Head generated by the pump at the operating point.
- **Discharge** – Discharge produced by the pump at the operating point.
- **Pump Intake Grade** – Calculated hydraulic grade line at the intake of the pump.
- **Pump Discharge Grade** – Calculated hydraulic grade line at the downstream end of the pump.

For a constant power pump, the calculated operating point may be outside of the range for which the pump is representative of a real pump. Be very cautious and check all results carefully.

For more information about the theory behind the pump operating point, see the Pump Theory chapter.

Valves General Tab

The **General** tab for valves is organized into the following sections:

- **General –** General information about the valve.
- **Valve Characteristics –** Diameter and minor loss coefficient of the valve.
- **Initial Setting –** Behavior of the valve at the start of the analysis.
- **Pipes** – Direction in which the valve is controlling the flow. You can reverse that direction by clicking the **Reverse** button.
- **Calculated Hydraulics** – Calculated hydraulic data upstream, downstream, and through the valve.
- **Water Quality** – Result of the water quality computations at the valve when a Water Quality Analysis has been performed.

For more information on the data, see the topic on each section. The **Water Quality** section is identical for all elements and is described in the "Pressure Pipe General Tab" section.

General Section

This section allows you to enter general information about the valve such as:

- **Label –** Unique "name" referencing the valve in reports, error messages,and tables.
- **X-Coordinate** (Easting) – The location of the valve may be represented by an X-value or an Easting value, depending on individual preferences.
- **Y-Coordinate** (Northing) – The location of the valve may be represented by a Y-value or an Northing value, depending on individual preferences.
- **Elevation** – Elevation of the valve.

Valve Characteristics Section

The **Valve Characteristics** section defines the following parameters:

- **Element Type –** Indicates whether the valve type is PRV, PSV, PBV, FCV or TCV.
- **Diameter –** Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss, when a minor loss coefficient is entered.
- **Minor Loss Coefficient** – Coefficient used to model any minor loss associated with the valve for the specified valve diameter, when the valve is fully open. Click the **ellipsis (…)** button to define composite minor losses. The valve is fully open in the following two cases:
  - The valve status is set to Inactive.
  - The valve status is set to Active and the hydraulic conditions are such that the valve is fully open.

Minor loss data is not required for Throttle Control Valves (TCVs) because the minor losses are already accounted for by the valve's primary purpose.

To change the type of a valve, use the element morphing feature of WaterCAD.

Initial Setting Section

The initial conditions describe the valve's behavior at the start of the analysis. These conditions include:

- **Valve Status** – A valve can have several different status conditions:
  - Active (throttling, opening, or closing depending on system pressures and flows)
  - Closed (no flow under any conditions)
  - Inactive (wide open, with no regulation)
- **Settings/Hydraulic Grade/Pressure** – For Pressure Reducing, Pressure Sustaining and Pressure Breaker Valves, specify either the initial hydraulic grade or the pressure setting associated with the valve.

> **You only need to specify either the pressure setting or the hydraulic grade setting. The other will be automatically calculated based on the valve's elevation.**

- **Discharge** – For Flow Control Valves, specify the initial discharge to maintain through the valve.
- **Headloss Coefficient –** For Throttle Control Valves, specify the initial minor losses associated with the valve.

> **In Steady-State Analysis mode (in WaterCAD), the Initial Status is used as the permanent status. However, it can be overruled by the presence of controls, if the Use Controls in Steady-State Analysis check box in the Calculation Options dialog is checked. The Calculation Options dialog is accessed by clicking the GO button in the main view to display the Calculation tab of the Scenario Editor, and then clicking the Options button.**

Pipes Section

This section allows you to specify the Upstream Pipe and Downstream Pipe.

> **The valve direction, along with the flow direction, affects the behavior of the valve, as explained in the Valve Editor topic at the beginning of this Chapter.**

> **You can switch the Upstream Pipe and Downstream Pipe by clicking the Reverse button.**

Calculated Hydraulics Section

This section reports the following calculated hydraulic parameters for a valve:

- **Discharge** – Calculated flow rate passing through the valve.
- **Velocity** – Calculated velocity inside the valve, based on the valve diameter.
- **Headloss** – Calculated headloss through the valve.
- **From HGL –** Calculated hydraulic grade immediately upstream of the valve.
- **To HGL** – Calculated hydraulic grade immediately downstream of the valve.

6.2.2 Demand Tab

This program provides the ability to define a hydraulic load connsisting of multiple demands and inflows for each junction node in the network. Each individual hydraulic demand or inflow consists of a baseline flow rate and a pattern that is applied when performing an Extended Period Simulation (EPS). This software provides a table for editing hydraulic loads. Each row represents an individual hydraulic demand or inflow. The table has three columns: Type, Demand, and Pattern:

- **Type** – Choose the type of load. Demand represents a withdrawal of liquid from the network system (if the value entered is negative, then the liquid is entering network). Inflow represents the addition of liquid to the system (negative inflow represents flow leaving the system).
- **Demand –** Enter the baseline flowrate for the load. This number will always be positive. If you need to define an inflow, change the load type. The units are volume per unit time (typically l/s or gpm).
- **Pattern –** Choose the EPS pattern that will apply to this load. Each load in the table can have a different EPS pattern. The multipliers defined in the pattern will be applied against the baseline load.

The **Demand** dialog has the following command buttons:

- **Insert –** Insert a row in the table.
- **Duplicate –** Create a new row in the table with the same values as the selected row.
- **Delete –** Delete the selected row of the table. The selected demand or inflow is removed from the list.
- **Graph –** Generate a graph of the total demand over time at this junction.

6.2.3 Section Tab

Tank section data includes the information necessary to describe the storage characteristics of the tank. They have been factored into the following logical groups:

- **Section –** The type of cross-section and the basic storage parameters.
- **Operating Range –** The minimum, initial, and maximum operating elevations.
- **Cross Section** – Parameters describing the cross-sectional geometry.

Tank Section

The general information for tank section consists of the following:

- **Section** – Choose the type of cross section for this storage tank. There are two types of cross sections to choose from: Constant Area and Variable Area.
- **Inactive Volume –** Enter the inactive volume for this storage tank. This data is used when performing water quality analysis.
- **Total Active Volume** – If this storage tank is a Constant Area Tank, the total active volume will be computed from the other tank data and this field will not be editable. If this is a Variable Area Tank, then enter the total storage volume for the tank.

Operating Range Section

This section allows you to set the absolute limits for the water levels in the tank. Elevations are relative to the same datum as the rest of your system, while levels refer to heights of water above the tank's base elevation. The operating range fields prompt you for the following values:

- **Elevations/Levels** – Select whether you want to enter the data in terms of absolute elevation (typically based on the sea level) or in terms of levels (relative to an arbitrary base elevation of the tank you specify).
- **Maximum** – Highest allowable water surface elevation or level. If the tank fills above this point, it will automatically shut off from the system.
- **Initial** – Value used as the water surface elevation or level when performing steady-state calculations, or as the beginning condition when performing an extended period simulation.
- **Minimum** – Lowest allowable water surface elevation or level. If the tank drains below this point, it will automatically shut off from the system.
- **Base Elevation** – Elevation of the storage tank base used as a reference when entering water surface elevations in the tank in terms of levels.

Cross Section Section

There are two basic types of storage tanks, as described below:

Constant Area Section

The cross sectional geometry of the tank is constant between the minimum and maximum operating elevations. Two parameters are needed to fully describe a constant area tank section:

- **Cross Section** – Choose whether the cross section is circular or non-circular.
- **Average Area/Diameter** – Enter the average area of the non-circular cross-section, or the diameter of the circular cross-section.

Variable Area Section

The cross-sectional geometry varies between the minimum and maximum operating elevations.

Depth/Volume Ratio Table – Enter a series of points describing the storage characteristics of the tank. For example, at 0.1 the total depth (depth ratio = 0.1) the tank stores 0.028 the total active volume (volume ratio = 0.028). At 0.2 the total depth that tank stores 0.104 the total active volume (0.2, 0.104), etc.

> **The storage characteristics of the tank can be plotted. Choose Tank Curve from the Report Button at the bottom of the Tank Dialog.**

6.2.4 Controls Tab

Controls allow you to configure the hydraulic model to change the status or settings of a pump, valve, or pipe at a specific time or when specific junction pressures or tank water levels occur in the network.

- **To add a control** – Click the **Add** button. This will open the **Control** dialog where the specifics of the control can be edited.
- **To edit an existing control** – Select the description of the control you wish to edit and click the **Edit** button.
- **To duplicate an existing control** – Select the description of the control you wish to duplicate and click the **Duplicate** button.
- **To delete an existing control** – Select the description of the control you wish to delete and click the **Delete** button.

Pipes with check valves cannot have controls.

Control Dialog

Several types of information are required to define a control for a pressure pipe, pump, or valve. This data is grouped into the following sections:

- **Preview** – Textual description of the control being edited.
- **Control –** Specify the type of control, either Status or Setting.
- **Control Condition** – Specify whether the control is based on a time condition or a nodal condition and then specify the control setting.

Control Preview

The control preview provides a textual description of the control being edited. The control preview is continuously updated while you edit a control, providing constant feedback as to the state of your control.

Control

This software supports two types of controls:

- **Status –** Controls the Open/Closed (pipes), inactive/closed (valves), or On/Off (pumps) status.
- **Setting** – Controls the relative speed factor of a pump and the parameters for a valve.

Only status controls are available for pipes. Setting controls are not appropriate. When pumps are turned on by a control, their relative speed factor is set to 1.00.

To activate a closed or inactive valve, use a setting control. Similarly, to turn a pump on at a relative speed setting other than 1.00, use a setting control.

Node Condition

A node condition dictates that the control will be triggered when the hydraulic condition of a specified tank or pressure junction is reached.

The comparison component allows the following:

- **Above** – Trigger the control when the junction or tank’s hydraulic parameter is above the node condition's hydraulic parameter.
- **Below –** Trigger the control when the junction or tank’s hydraulic parameter is below the node condition's hydraulic parameter.

You can express the conditions at the control node in terms of Pressure or Hydraulic Grade.

Example:

"Closed if node J-2 below 10 psi" means that the controlled pipe will close when the pressure at junction J-2 goes below 10 psi.

Time Condition

A time condition dictates that the control will be triggered when the specified amount of time has elapsed.

Examples

- "Closed at time 2.00 hr" – At 2.00 hours into the analysis, this link will be closed.
- "Set hydraulic grade to 440 ft at time 5.50 hr" – At 5.5 hours into the analysis, the hydraulic grade of this pressure regulating valve will be set to 440 feet.

6.2.5 Quality Tab

The **Quality** tab of an element allows you to edit the input data related to water quality. Three types of water quality analyses can be performed, as defined in the **Scenario Editor** dialog accessed by clicking the **GO** button in the main WaterCAD window. These are Water Age, Constituent Concentration and Source Tracing. There are two basic parts to an element's water quality input data:

- **Water Quality** – Display the active water quality alternative for the current scenario, as well as initial water quality conditions or component reaction rates, depending on the type of water quality analysis being performed.
- **Constituent Source** – For nodes only. Allows you to define this node as a source for a chemical constituent (if a constituent analysis is to be done) and to specify the corresponding constituent concentration at this node over time.

A constituent source may be a tank, reservoir, or junction (but not a pump or valve).

The Water Quality data is only used when performing a Water Quality analysis, which can only be done in Extended Period Simulation mode.

When performing a Constituent or Trace Analysis, the constituent and source trace node are defined in the Constituent and Age Alternative Editor respectively.

Water Quality Section

The general water quality information consists of several parameters, some of which vary slightly for different types of water quality analysis (Water Age, Constituent Concentration or Source Tracing):

- **Alternative –** Read-only field showing which water quality alternative is active for the current scenario.
- **Initial Age, Constituent, or Trace –** Specify the initial water age, constituent concentration or source trace at the current location, depending on which type of water quality analysis is currently selected in the **Scenario Editor** dialog. This does not apply to pipes.

When performing Constituent analysis, reaction coefficients are needed, as defined below:

- **Bulk Reaction Coefficient –** Coefficient defining how rapidly a constituent grows or decays over time. This applies to Tank and Pipes only.
- **Wall Reaction Coefficient –** Coefficient defining the rate at which a substance reacts with the wall of a pipe.

For age and trace analyses, pipe velocity and flow rate are the only related data needed for computations. Therefore, these reaction coefficient fields are greyed out or not displayed. For constituent analyses, however, the bulk and pipe reaction coefficients are needed to define the reactions that occur within the pipes (in the water and between the water and pipe wall) and in the tanks.

The bulk and wall reaction coefficient fields are initialized with the values defined in the constituent library, but may also be edited individually. In order to select the constituent being modeled, and its corresponding parameters, or revert to default values, use the Constituent Alternative Editor.

Constituent Source Section

Any node element (i.e. tank, reservoir, or junction) can serve as a source for a chemical constituent.

To turn a node into a constituent source:

- Click the check box in the **Constituent Source** group title.
- Enter values for:
  - **Constituent Baseline Load –** Concentration of the constituent to be modeled, used in conjunction with the constituent pattern to represent the concentration over time.
  - **Constituent Pattern –** EPS Pattern that will apply to this load. The multipliers defined in the pattern will be applied against the baseline constituent load.

The behavior of the source during the course of a water quality calculation varies depending on the type of element, as follows:

- **Junction –** The concentration is the source concentration and varies with time according to the constituent pattern.
- **Tank –** Concentration is always calculated. If the tank is tagged as a source, the discharge from the tank will have the source concentration. It will vary with time according to the source pattern.
- **Reservoir –** Constituent concentration does not vary from the initial value with time, unless the reservoir is tagged as a source for the constituent. The concentration at the reservoir will vary with time according to the selected pattern.

Any water leaving a constituent source has a concentration in accordance with the baseline concentration and the chosen pattern.

6.2.6 Fire Flow Tab

The **Fire Flow** tab of the junction editor offers the ability to adjust an individual junction's required fire flows and pressures. If these values are not specifically entered for a given junction, the values will be based on the default fire flow data as entered in the **Fire Flow Alternative Editor**, accessed by selecting the **Analysis\Alternatives** menu item, and clicking the **Edit** button on the **Fire Flow** tab.

- **Fire Flow Input** – Minimum required fire flow at this junction and minimum pressures to be maintained.

- **Fire Flow Calculation Results** – After performing a fire flow analysis, results are available for the junction node assuming it is part of the fire flow selection set.

> **Edit the following things exclusively in the Fire Flow Alternative Editor: whether a fire flow analysis is to be performed at a node, whether the needed fire flow is to replace or be added to current demands, whether a minimum pressure is required for the entire system, and default fire flow input values.**

> **Results of fire flow calculations, which are obtained from calculations performed separately for an automatic batch run are only reported in the Fire Flow tab and in the Fire Flow Tabular Report (accessed from Report\Tables\Fire Flow Report or the FlexTable icon). Results reported in the other Element Editor tabs do not take into account any fire flow, unless you explicitly entered this fire flow as a demand at a specific junction.**

Fire Flow Input

The fire flow input data for a junction are as follows:

- **Needed Fire Flow** – The flow rate required at the junction to meet fire flow demands. This value will be added to or replace the junction's baseline demand, depending on the default setting for applying fire flows as specified in the Fire Flow Alternative dialog.

- **Fire Flow Upper Limit** – Similar to the Target fire flow described above, this defines the maximum allowable fire flow that a junction can provide and the maximum allowable fire flow that can occur at any single withdrawal location. This is a user-specified practical limit that will prevent this program from computing unrealistically high fire flows at locations such as primary system mains, which have a large diameter and high service pressures. Remember that a system's ability to deliver fire flows is ultimately limited by the size of the hydrant opening and service line, as well as the number of hydrants available to combat a fire at a specific location.

- **Residual Pressure** – Minimum residual pressure to occur at the junction node. The program determines the amount of fire flow available such that the residual pressure at the junction node does not fall below this target pressure.

- **Minimum Zone Pressure** – Minimum pressure to occur at all junction nodes within the Zone you are testing. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone associated with it, which can be located in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another Zone.

- **Minimum System Pressure** – Minimum pressure allowed at any junction in the entire system as a result of the fire flow withdrawal. If a node's pressure anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied. A fire flow analysis may be configured to ignore this constraint.

Fire Flow Calculation Results

After performing a fire flow analysis, the following calculation results are available for each junction node in the fire flow selection set:

- **Satisfies Fire Flow Constraints** – Whether or not this junction node meets the fire flow constraints.

- **Available Fire Flow** – Amount of flow available for fire protection while maintaining all fire flow pressure constraints.
- **Calculated Residual Pressure** – Calculated pressure at the junction node during the fire flow withdrawal.
- **Calculated Minimum Zone Pressure –** Minimum calculated pressure of all junctions in the same zone as this junction.
- **Minimum Zone Junction** – Label of the junction corresponding to the minimum zone pressure.
- **Calculated Minimum System Pressure –** Minimum calculated pressure of all junctions in the system.
- **Minimum System Junction –** Label of the junction corresponding to the minimum system pressure.

6.2.7 Cost Tab

Pressure Pipe Cost Tab

This tab allows you to specify the costs associated with a pipe, which can be divided into unit costs per pipe length, and fixed costs. It is divided into the following sections:

- **Fixed Cost Table** – Specify the fixed cost of this pipe.
- **Unit Costs** – The cost of the pipe can be calculated as a cost per unit length. That cost per unit length can itself be a function of a pipe variable (such as pipe diameter or material), and be a value specific to a pipe.
- **Total Costs** – Display the calculated total cost of this pipe.

You can specify if the total cost of this pipe is to be included in the Total System Cost Tabular Report by checking the **Include Element in Cost Calculation**.

Pressure Pipe Fixed Cost Table

This table allows you to define any pipe cost that is not proportional to the pipe length, such as bends, valves, etc.

- **Label** – Description of a fixed cost item associated with the pipe.
- **Quantity** – Number of identical cost item associated with the pipe.
- **Unit Cost** – Cost of one such fixed cost item.

Pressure Pipe Unit Costs Section

This section allows you to define costs that are proportional to the length of a pipe:

- **Unit Cost Function** – Allows you to associate a unit cost function selected from the Unit Cost Function scroll-down list. Click the **ellipsis (…)** button to access the Unit Cost Functions Manager. This allows you to edit or add unit cost functions in a tabular or formula format based on a pipe attribute such as pipe diameter.
- **Unit Cost –** Display the Unit Cost for this pipe calculated by using the selected Unit Cost Function, assuming you selected one.
- **Additional Unit Cost –** Define or add a unit cost specific to this pipe.

Pressure Pipe Total Costs Section

This section reports the calculated costs associated with this pipe, as follows:

- **Total Fixed Cost** – Cost of this pipe resulting from any fixed cost, as defined in the **Fixed Cost** section of this tab.
- **Extended Cost** – Cost of this pipe resulting from the unit cost function, as defined in the **Unit Cost** section of this tab.
- **Additional Cost** – Cost of this pipe resulting from any additional unit cost, as defined in the Unit Cost section of this tab.
- **Total Cost –** Sum of the Extended Cost, Additional Cost, and Total Fixed Cost described above.

Nodal Element Cost Tab

This tab allows you to specify the costs associated with a nodal structure, expressed as fixed costs. It is divided into the following sections:

- **Fixed Cost** – Specify the fixed costs of the element.
- **Total Fixed Cost** – Displays the calculated total cost of the element.

You can specify if the total cost of the node is to be included in the Total System Cost Tabular Report by checking the **Include Element in Cost Calculation**.

Nodal Element Fixed Cost Table

This table allows you to define any cost associated with a nodal structure (junction, tank, reservoir, pump, valve), as follows:

- **Label** – Description of a fixed cost item associated with the structure.
- **Quantity** – Number of identical cost item associated with the structure.
- **Unit Cost** – Cost of one cost item.

Total Fixed Cost Section

- **Total Fixed Cost** – Cost of the nodal structure, as defined in the Cost Table on this tab.

6.2.8 User Data Tab

The **User Data** tab allows you to view and edit the customizable user data for each element. This tab is composed of two sections:

- **User Data –** Any Date/Time, Number, Text, and Yes/No data defined by the user.
- **User Memos** – Any memo data fields defined by the user.

For information on how to add new fields or edit an existing field format, see the Help on the **User Data Extension** dialog.

> **Default user-defined attributes are provided. These can easily be deleted or modified.**

User Data Extensions are a powerful way to add your own data to the project. This data will not affect the hydraulic calculations in any way, but can be used as any other data for things such as sorting, annotating, reporting, and importing/exporting.

User Data Section

This section contains a list of Date/Time, Number, Text, and Yes/No user data fields, displayed as single line fields. User data fields are defined in the **User Data Extension** dialog which is accessed from the **Tools** menu.

User Memos Section

This section contains a list of any memo fields, displayed as multiple line scrolling text panes. User Memos are defined in the **User Data Extension** dialog, which is accessed from the **Tools** menu.

6.2.9 Messages Tab

All element dialogs have a **Messages** tab which contains three parts:

- **Message List –** Contains information that is generated during the calculation of the model, such as warnings, errors, and status updates.
- **Description –** An informative statement that you may enter about the element.
- **Notes –** Contains notes that you enter, and may include a description of the element, a summary of your data sources, or any other information of interest.

Messages, descriptions, and notes will be printed in any element report.

6.3 Prototypes

Prototypes allow you to enter default values for the 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 PVC pipes, use the pipe prototype to set the **Material** field to PVC. When a new pipe is created, its material attribute will default to PVC.

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.

You can configure the element prototypes at the beginning of a new project during the Project Setup Wizard. You can also select **Tools\Prototypes** to edit the prototypes for the project at any time.

6.4 User Data Extension

The User Data Extension feature allows you to add your own data fields to the project. For instance, you could add a field for keeping track of the date of installation for an element, or the type of area serviced by a particular element. User Data Extensions exhibit the same characteristics as the 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/edited in FlexTables, included in tabular reports or element detailed reports, annotated in the drawing, color coded, reported in the detailed element reports, and accessed on the **User Data** tab of each Element Editor dialog.

None of the user data affects the hydraulic model calculations. However, their behavior concerning capabilities like editing, annotating, sorting and database connections is identical to any of the other standard attributes.

6.4.1 User Data Extension Dialog

The **User Data Extension** dialog holds a summary of the user data extensions currently defined in the project. In this dialog, there is a tab for each type of element. By clicking a particular tab you can access the user data extensions currently defined for that type of element. The software initially contains default user data extensions, but these can be deleted or edited. Each tab in the **User Data Extension** dialog is composed of a table listing characteristics of the user data extensions defined for that type of element. In addition, there are a series of buttons that can be used to add, edit, delete, and share individual user data extensions. The table listing the user data extensions consists of the following four columns:

- **Label –** This is the description that will appear next to the field for the user data extension, or as the column heading if the data extension is selected to appear in a FlexTable.
- **Type –** This column lists the type of data that is valid for the data extension. The available data types are Date/Time, Number, Text, Memo, and Yes/No.
- **Unit/Picture –** This column contains the unit of each numeric data extension, and a picture of the format of the date and time for Date/Time data extensions. Both the unit and the date and time picture are selected when you create the data extension. They can always be modified by editing the data extension.
- **Shared –** If an asterisk appears in this column, it indicates that the user data extension is shared among two or more types of elements. See explanations on the **Existing Fields to Share With** dialog for more details.

The following list describes the four buttons that appear on the right side of the table:

- **Add** – You can open the **User Field Specification** dialog by clicking this button. Here, you can define the properties of the user data extension that you are adding.
- **Edit –** You can edit an existing user data extension by highlighting the data extension you wish to edit and clicking this button. This will open the **User Field Specification** dialog where you can change the properties for that item.
- **Delete –** You can delete a data extension by highlighting it and clicking this button. If the data extension you are deleting is shared among multiple types of elements, it will only be removed from the element type that you are currently editing. If you remove a user data extension all the information contained in that field will be permanently removed.

- **Share** – You can open the **Existing Fields to Share With** dialog by clicking this button. Here, you can pick which of the attributes already defined for other types of elements you would like to share with the current type of element.

At the bottom of the **User Data Extension** dialog is a **File** button that allows you to import or save a set of user-defined data extensions. You can save the current configuration of user data extensions for later use by selecting **File\Save**, and specifying a file location and name. The file extension for the files holding the user data extension configurations is '.udx'. Select **File\Import** to merge the data extension configurations defined in these files into the current project. Importing a '*.udx' 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.

To access the **User Data Extension** dialog click **Tools\User Data Extension…** from the main menu.

User Field Specification Dialog

The properties defining a user data extension can be viewed and edited in the **User Field Specification** dialog. This dialog is composed of two tabs:

Type – Enter the user data specification.

Notes – Enter any notes.

Type Tab

The **Type** tab is composed of two sections:

Type – Contain fields for entering the label for the user data extension and the data type.

Format – Contain fields for entering information defining the specification for the type of user data extension selected in the **Type** section.

Type Section

The **Type** section contains fields for entering the label and data type for the user data extension. The name entered in the **Label** field corresponds with the user data extension field in the **User Data** tab of the element editor dialog. It is also used as the column heading when the user data extension is selected to be reported in a FlexTable.

If you want the label to be displayed on multiple rows when it is used as a column heading, you can use forward slashes to specify the location of line breaks. When the label is used as a field label in a dialog, the forward slashes will be converted to spaces. In the FlexTables, there is an option to use abbreviated labels for the column headings. If you want an alternative label to be displayed, you can specify an abbreviated label after the original label and separate them by the bar symbol, '|'. When the option to display abbreviated labels is enabled in the FlexTables, this is the text that will be used as the column heading. For instance, if you specified the label, 'Date/Installed | Date/Inst.' it would be displayed in one of the following three ways depending on the location and options selected.

Field Label	Column Heading	Column heading with the short label option selected
Date Installed:	Date Installed	Date Inst.

You can select from five different types of data for your user data extension from the drop-down list in the field labeled **Type**. An explanation of each is presented in the list below:

- **Date/Time** – Use this data type when you want the values you are entering to be in a standard date and time format. This format can be more useful than storing date information in a simple text field because it will allow the dates to be sorted correctly when they appear in a FlexTable.
- **Memo** – If a user data extension is defined to be a memo, it will appear as a scrolling text pane in the **User Memos** section of the **User Data** tab in the element editor dialog.
- **Number –** Use this data type for fields that contain numeric values. You can specify a unit for the information in this field. The values contained in this field will then be automatically converted if you change the unit for this field.
- **Text –** Use this data type to create a single-line text field.
- **Yes/No –** Use this data type to display the attribute as a check box to represent the affirmative or negative state of the information.

Format Section

This section is enabled only if you select **Date/Time** or **Number** in the **Type** section, letting you define the properties governing the type of data selected.

Number Format – If the type of data you selected was numeric, you can select a unit type (length, volume, intensity, etc.), a unit, a display precision, and whether or not to use scientific notation. There are not any format options for memo, text, and Yes/No data types.

Date/Time Type Format – If you selected the **Date/Time** type, you can specify whether you expect the date or time to be displayed first in the input field and the format of the date and time information. The format in which the date and time information will be displayed can either be selected from the drop-down lists, or you can type your own custom format directly into the **Date Picture** and **Time Picture** fields. If one of these fields is left blank, the corresponding information will not be displayed.

The Date/Time data type consists of an input and an output format. The input format is a fixed format that is determined by the regional settings on your computer. Whenever you enter information into a **Date/Time** field, the information must be entered in the input format. If the date or time information is not entered according to the input format, then the value will simply revert to the original value.

The output format is simply a mask that defines the manner in which the date and time information will be displayed. It does not affect the way that the date and time information can be entered into a **Date/Time** field. The output format can be edited as follows:

- To create numbers with no leading zeros for single-digit days, years, or months, use lowercase d, lowercase y, or uppercase M.
- To create numbers with leading zeros for single-digit days, years, or months, use lowercase dd, lowercase yy, or uppercase MM.
- To create abbreviations for the day, year, or month, use lowercase ddd, lowercase yyy or uppercase MMM.
- To create the full name of the day, year, or month, use lowercase dddd, lowercase yyyy or uppercase MMMM.

If there are characters in the output format that do not map to valid date or time information, then the actual value of the character will be displayed. For example, if you wanted the date to be displayed as June 15, 1998, then you would define the format as 'MMMM d, yyyy'. Since the spaces and comma do not map to any of the date information, their actual values are displayed. To include a piece of text that contains a character that maps to the date or time information, use single quotation marks (') around the text.

Notes Tab

This tab contains a text pane for entering notes about the current data extension. The text entered here is not displayed anywhere in the model, but allows you to keep records for a particular data extension.

Existing Fields to Share With Dialog

This dialog allows you to choose which of the attributes already defined for other types of elements you want to share with the current type of element (corresponding to the **User Data Extension** tab currently opened).

Available Items – Lists attributes defined for other element types that are not already shared. In order to add attributes to the current element type, highlight them and click the **Add** button to transfer them to the **Selected Items** list.

Selected Items – The attributes in the **Selected Items** list will be added to the current element after you click the **OK** button.

All the characteristics such as data type, format, unit, and display precision for a particular user data extension are the same for all the elements that share it. This is useful when the attribute you are adding needs to be the same for all the element types for which it is defined. For instance, if you have a "date installed" field for every element, sharing guarantees that the date format is the same for every element and will appear in a single FlexTable column. If at a later point you decide the date should be in a different format, you can change the format for one type of element. That change will filter through to all the elements that share that attribute.

6.5 Zone Manager

The zone manager allows you to manipulate zones quickly and easily. Zones listed in the Zone Manager can be associated with each nodal element using the Element Editors, Prototypes, or FlexTables. This manager includes a list of all of the available zones and standard manager features, such as:

- **Add –** Add a new zone to the zone list.
- **Edit –** Make changes to an existing zone.
- **Duplicate** – Create a copy of an existing zone.
- **Delete** – Delete an existing zone.

A Zone cannot be deleted if it is referenced by any element.

To open the **Zone Manager** dialog, choose **Zones** from the **Analysis** menu.

6.5.1 Zone Dialog

The zone dialog allows you to name the zone label. When a zone is named, the junctions are automatically assigned the new name. The zone dialog contains pertinent information, including:

- **Label –** Required name to identify the zone.
- **Notes** – Optional input describing the zone.

In addition to this information, there are also buttons that enable you to make changes to the collection of elements in the zone, such as adding elements to the zone.

Only one zone can reference an element. If you add an element to a zone, the element is automatically removed from the zone that it was previously in.

Notes

Chapter 7

FlexTables

7.1 Tables

FlexTables provide you with a powerful data management tool that can be used to edit input data and present output data in a quick, efficient manner. Haestad Methods provides you with some basic element tables. However, these tables can be customized to fit your particular needs. You can create your own tables combining various input and output data from different model elements. You can use FlexTables to view all elements in the network, all elements of a specific type (e.g. all pipes), or any subset of elements. Additionally, tables can be filtered, globally edited, and sorted to ease data input and present output data for specific elements.

FlexTables may also be used for creating results reports that can be sent to a printer, a file, or the Windows clipboard.

7.2 Table Manager

The Table Manager provides support for creating, opening, and managing tables. The **Table Manager** dialog provides a list of all available tables. Although the predefined tables provide access to most of the network element information, it is sometimes practical to present model results and input data through user-defined tables. The **Table Management** menu button provides the following tools for manipulating user-defined tables:

- **OK** – Open the selected table.
- **Close –** Exit the **Table Manager** dialog without opening a table.
- **Table Management\New –** Create a new table using the **Create New Table** and **Table Setup** dialogs.
- **Table Management\Edit** – Modify the layout of the selected table using the **Table Setup** dialog.
- **Table Management\Rename** – Rename the selected table.
- **Table Management\Duplicate** – Duplicate the selected table for additional customizing. This is a very useful feature when you need to change a predefined table.
- **Table Management\Delete** – Delete the selected table.
- **Table Management\Reset** – Reset a table's units to the current unit system or reset a predefined table to factory defaults.

You cannot rename or delete the predefined Tables that come with this software.

When you choose to print a table, the table name will be used as the title for the printed report. You can change the report title by renaming the table.

To access the Table Manager, click the **Tabular Reports** button on the main toolbar, or choose **Report\Tables** from the main menu.

7.2.1 Creating New Tables

To create a new table, open the Table Manager by clicking the **Tabular Reports** button on the main toolbar, or by choosing **Report\Tables** from the main menu. In the **Table Manager** dialog, click the **Table Management** button and select **New**.

1. Specify the Table Type to indicate the type of network elements you want to display in your table.
2. Specify either a one or two row display for your table (in SewerCAD or StormCAD).
3. Enter the name of your new table in the field labeled "**Enter the description for this table:**". This name will also be used as the report title when this table is printed.
4. Click **OK** to accept these settings and proceed to the **Table Setup** dialog where you can define your table.

7.2.2 Editing Tables

The **Edit** option allows you to specify the columns that will appear in your table.

7.2.3 Duplicating Tables

The **Duplicate** option allows you to create a new table based on an existing table.

7.2.4 Deleting Tables

The **Delete** option allows you to delete any table that you have defined. You cannot delete any of the predefined tables.

7.2.5 Renaming Tables

The **Rename** option allows you to change the name of any table that you have defined.

The table name will be used as the title in printed reports. You cannot rename any of the predefined tables. If you need to rename a predefined table, duplicate it first and then rename it.

7.2.6 Resetting Tables

Reset Units to the Current Unit System – This option is only available for tables that are in Local Units mode. Local Units mode allows the table to maintain its own "local" set of column properties (units, precision, etc.). You can use this option to reset all units in the selected table to the defaults for the current unit system, which refers to the units used in the current project. You will be prompted to confirm before this action is performed.

Reset to Factory Defaults – You can reset any of the predefined tables to the factory defaults (this option is not available for tables that you create).

7.3 Table Setup Dialog

The **Table Setup** dialog allows you to customize any table, through the following options:

- **Table Type –** This field allows you to specify the type of network elements that will appear in the table (e.g. only pipes will appear in a "pipe" table).
- **Available Columns** – This list contains all the attributes that are available for your table design, and will change based on the **Table Type** field.
- **Selected Columns** – This list contains attributes that will appear in your custom designed table. When you open the table, the selected attributes will appear as columns in the table in the same order that they appear in the list.
- **Allow Duplicate Columns –** This is an advanced feature that allows you to place two identical columns in the same table and set them to different unit systems.
- **Column manipulation buttons –** These buttons allow you to select or deselect columns to be used in the table, as well as to arrange the order in which the columns will appear.

> **The number next to the Selected Columns label indicates the number of columns that will appear in your table.**

To access the **Table Setup** dialog from the **Table Manager** dialog, highlight the table you wish to edit and select **Edit** from the **Table Management** menu button.

7.3.1 Table Type

The **Table Type** field allows you to specify the types of elements that will appear in the table (e.g. only pipe elements will appear as rows in a "pipe" table). The table type also provides a filter for the attributes that appear in the **Available Columns** list. When you choose a table type, the available list will only contain attributes that can be used for that table type (e.g. only pipe attributes will be available for a "pipe" table).

7.3.2 Available Table Columns

The **Available Columns** list is located on the left-hand side of the **Table Setup** dialog. 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 columns, while those displayed in white represent editable columns.

7.3.3 Selected Table Columns

The **Selected Columns** list is located on the right-hand side of the **Table Setup** dialog. The attributes in this list will appear as columns in the table when it is opened. The columns will appear in the same order as the attributes in the selected list.

To add columns to the **Selected Columns** list:

1. Select one or more attributes in the **Available Columns** list.
2. Click the **Add** button [>] or drag the highlighted attributes to the **Selected Columns** list.

7.3.4 Table Manipulation Buttons

The **Add** and **Remove** buttons are located in the center of the **Table Setup** dialog.

[>] Adds the selected item(s) 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 item(s) from the **Selected Columns** list.

[<<] Removes all items from the **Selected Columns** list.

To rearrange the order of the attributes in the **Selected Columns** list:

1. Highlight the item to be moved.
2. Move it up or down in the list by clicking the up button or the down button (located below the **Selected Columns** list), or by simply dragging it to the desired location.

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 will provide group selection behavior. Holding down the Control key will provide single element selection behavior.

The items displayed in yellow represent non-editable columns (e.g. columns that contain calculated data) while those in white represent editable columns (e.g. columns that contain input data).

7.3.5 Allow Duplicate Columns

Set this check-box to allow duplicate columns in a table. **Allow Duplicate Columns** is an advanced feature that allows you to place two identical columns in the same table and set them to different unit systems.

7.4 Table Window

The **Table** window is where you will perform much of your data input and review. It has many features to assist you with data entry, data formatting, report customization, and output generation. To access the **Table** window, select a table from the **Table Manager** and click **OK** to open it. Here are some of the topics that will be covered in this section:

- Table Navigation

- Table Customization

Options:

- Table Sorting
- Table Filtering
- Changing Column Headings
- Globally Editing Data
- Local vs. Synchronized Units
- Mixing Units in a Tabular Report
- Abbreviated Labels
- Changing Column Display Properties

Output:

- File (Export Table to ASCII File)
- Table Copy to Clipboard
- Table Print
- Table Print Preview

Columns:

See the Glossary for information regarding the definitions of the columns in the **Table** window.

Use the Scenario control located at the top of the Table Window to quickly view the data for different scenarios.

To access the **Table** window open the **Table Manager** dialog, select the table you wish to open, and click **OK**.

7.4.1 Editing Tables

Editable Table Columns

Editable table columns correspond to input data that you can change. The values in these columns can be modified either directly or through the **Global Edit** option. These columns are displayed with a white background.

Non-editable table columns correspond to model results calculated by the program. You cannot modify the values in these columns. These columns are displayed with a yellow background.

Table Navigation

The **Table** window supports two modes: Table Navigation Mode and Cell Navigation Mode. By pressing the **F2** key, you can toggle between them.

Table Navigation Mode

The Arrow keys, Home, End, PgUp, PgDn, Ctrl+<arrow> keys navigate to different cells in a table. Table Navigation Mode is the default mode when editing a table. To edit within a single cell of a table, press the **F2** key to switch to cell navigation mode.

Cell Navigation Mode (Edit Mode)

In Cell Navigation Mode, the Arrow keys, Home, and End keys navigate within a single cell. When cell navigation mode is active, the word "EDIT" will appear on the status pane at the bottom of the window. Cell Navigation Mode will automatically terminate when you press any extended key except for Left, Right, Home, End, Delete, or Backspace.

Globally Editing Data

You can globally change the values of any editable column in a table. Right-click the column that you wish to globally change and choose the **Global Edit** menu item.

For numeric columns:

1. Choose the operation to be performed (Add, Divide, Multiply, Set, or Subtract).
2. Enter the value you wish to use.
3. Press **OK** and the values in the entire column will be updated to reflect this change.
4. For non-numeric columns:
5. Enter the new value.
6. Click **OK**, and the values in the entire column will be updated to reflect this change.

Global Edit is available only for editable columns.

Global Edit is not available in two-row tables (in SewerCAD or StormCAD).

You can use Global Edit in conjunction with Filtering to globally edit a subset of elements.

7.4.2 Sorting/Filtering Tables

Sorting Tables

Tables can be sorted based on a single column, multiple columns, or network order.

Custom Sort

You can sort elements in the table based on one or more columns, in ascending or descending order. For example, the following table is given:

Slope (ft/ft)	Depth (ft)	Discharge (cfs)
0.001	1	4.11
0.002	1	5.81
0.003	1	7.12
0.001	2	13.43
0.002	2	19.00
0.003	2	23.27

A custom sort is set up to sort first by Slope, then by Depth, in ascending order. The resulting table would appear in the following order:

Slope (ft/ft)	Depth (ft)	Discharge (cfs)
0.001	1	4.11
0.001	2	13.43
0.002	1	5.81
0.002	2	19.00
0.003	1	7.12
0.003	2	23.27

Filtering Tables

To access the filtering operations, use the **Options** button at the top of the **Table** window (in the case of a FlexTable), or right-click the column header by which you wish to filter. Filters allow you to change the table so that only rows that match the specified criteria will appear.

- **Quick Filter** – Quickly set up a simple filter by right-clicking the column by which you wish to filter.
- **Custom Filter –** Set up a custom filter based on one or more criterion.
- **Reset –** Turn off the active filter, causing all available rows in the table to be displayed.

Another way to select which elements are displayed in the table consists of first selecting elements (graphically, or by using the Selection Set tool), then right-clicking any of the selected elements and choosing Edit Group from the pop-up menu that appears. This will display the Table Manager dialog. Only the selected elements will appear in any of the tables you open at this point.

When you perform a **Quick Filter** or a **Custom Filter**, the **Filter** dialog will open allowing you to specify your filtering criterion.

Each filter criterion is made up of three items:

- **Column** – 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 criterion elements can be added to a filter. Multiple filter criterion are implicitly joined with a logical "AND" statement. When multiple filter criterion 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, or the Table Window is closed.

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 appear in a highlighted color.

Table filtering allows you to perform global editing on any subset of elements. Only the elements that appear in the filtered table will be edited.

7.4.3 Table Customization

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. Use the **Table Manager** to rename your table.
- **Adding/Removing Columns –** You can add, remove, and change the order of columns by using the **Table Setup** dialog. Use the **Table Manager** to access the **Table Setup** dialog.
- **Drag/Drop Column Placement –** With the **Table** window open, select the column that you would like to move by holding down the left mouse button on its column heading. Drag the column heading to the left or right, and release the mouse button to drop the column into its new location.
- **Resizing Columns –** With the **Table** window open, place your pointer over the vertical separator line between column headings. Notice that the cursor changes shape to indicate that you can resize. Hold down the left mouse button and drag the mouse to the left or right to "stretch" the column to its new size. When you are satisfied, release the mouse button to set the new column width.
- **Changing Column Display Properties –** With the **Table** window open, right-click in the heading area of the column you wish to change and choose the **Properties** menu item. The current column properties will be displayed in the **Set Field Options** dialog. Refer to the section on Local Units for additional information.
- **Changing Column Headings –** With the **Table** window open, right-click the column heading that you wish to change and choose **Edit Column Label**. Refer to section on Changing Column Headings for additional information.

Changing Column Headings

To change the label of any column in the **Table** window, right-click the column heading that you wish to change and choose **Edit Column Label** from the context menu. The backslash character (\) can be used to insert a line-break wherever you want the title to be split into multiple lines. If you enter an empty label, the column heading will be restored to the default label.

Abbreviated Labels

Using label abbreviations will allow columns to take up less space. This will permit more data to fit on each page when printing a report.

To toggle the Use Abbreviated Labels option on and off, select **Options\Use Abbreviated Labels** in the **Table** window.

Changing Column Display Properties

You can change the display properties (e.g. units, precision) of any numeric column in the **Table** window. Right-click the label of the column that you wish to change and select **Properties** from the pop-up menu. This opens the **Set Field Options** dialog, where you can change the display properties of the column.

Local vs Synchronized Units

Use the **Options** button at the top of the **Table** window to access the **Use Local Units** menu item. Click the menu item to toggle between **Local Units** and **Synchronized Units**. A check mark will appear next to the **Use Local Units** menu item to indicate that Local Units mode is active. Otherwise, Synchronized Units mode is active.

- **Synchronized Units –** This is the default mode that allows the table to stay synchronized with the active project. If you have one project in US Customary and one project in SI units, the Table will match the units in the project that is currently open.
- **Local Units –** Local Units mode allows the table to maintain its own "local" set of column properties (units, precision, etc). This is a powerful feature that gives you the ability to build tables that are always in a fixed unit system, no matter what unit system the active project is currently using. This is a useful feature for printing reports for the same project in different unit systems.

When the **Table** window is open, the current unit synchronization mode is displayed in the status pane at the bottom of the window ("Local Units" or "Synchronized Units").

Mixing Units in a Tabular Report

This software allows for duplicate columns in a table, thus giving you the ability to display an attribute in two different units.

For example, to see two "Pipe Length" columns in a Table, one in feet and one in meters:

1. Open the **Table Manager** dialog.
2. Click the **Table Management** button, and select **New** to create a new table.
3. Select the pipe table type from the pull-down menu list and enter a name for your new table. Click **OK** and you will be taken to the **Table Setup** dialog where you can customize your table.
4. In the **Table Setup** dialog, activate the **Allow Duplicate Columns** check box (located at the lower left corner of the dialog).
5. Add the "Length" column to the **Selected Columns** list.

> **The Length column will still appear in the Available Columns list, but will be displayed in a lighter color (indicating that it has already been selected).**

6. Add the "Length" column again.
7. Click **OK** to close the **Table Setup** dialog. From the **Table Manager** dialog, highlight the table you have just created, and click **OK** to open the table.
8. Click the **Options** button at the top of the window and select the **Use Local Units** menu item to turn Local Units on. You will be prompted to verify that you want to use local units. Click **Yes**.
9. Right-click the first **Length** column and select **Length Properties** to set the units in the column to "ft." Then, right-click the second **Length** column to set the units to "m."

7.4.4 Table Output

File (Export Table to ASCII File)

You may export the data shown in the **Table Window** to an ASCII text file in either tab or comma-delimited format.

To export a table to an ASCII File format select **File\Export Data** and either **Tab Delimited** or **Comma Delimited** from the **Table** window.

Table Copy to Clipboard

The **Copy** button at the top of the **Table** window allows you to copy tab delimited data to the Windows clipboard. Tab delimited data can be pasted directly into your favorite spreadsheet program or word processor.

Table Print

The **Print** button at the top of the **Table** window is used to output the table directly to the printer.

Table Print Preview

Click the **Print Preview** button at the top of the **Table** window to view the report in the format that will be printed.

Using label abbreviations will allow some columns to be narrower, permitting more data to fit on each page. Use the Options button at the top of the Table window to access this option.

Printing with landscape orientation will also allow more columns to fit on a single page. From the Print Preview window, use the Options\Print Setup menu item to access orientation.

Chapter 8

Scenarios and Alternatives

The scenario management feature allows you to easily analyze and recall an unlimited number of "*What If?*" alternative calculations for your model. The powerful two-level design – which uses **Scenarios** that contain **Alternatives** – gives you precise control over changes to the model, while eliminating any need to input or maintain redundant data.

We have spent many hours coming up with a system that offers the power and flexibility that you demand, with the ease of use that you have come to expect from us. If you are like most users, you will want to jump right in without having to spend a lot of time reading. When you are ready to create your first scenario, you will find that you will be able to accomplish what you want easily and quickly.

The Scenario Wizard is designed to get you started quickly, while slowly exposing you to the power behind scenarios and alternatives.

When you are ready to model more complex scenarios, you will appreciate the power and flexibility provided by the various scenario management features.

If you are a beginning user, try the Scenario Wizard, and run the Scenario tutorial. Also, refer to the Scenario Management Reference Guide in Appendix C. If you are an advanced user, be sure to read about Alternatives, and investigate the **Alternatives Manager** dialog.

8.1 Alternatives

Alternatives are the building blocks behind scenarios. 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. The different types of alternatives are as follows:

- Physical
- Demand
- Initial Settings
- Operational
- Age
- Constituent
- Trace
- Fire Flow

- Cost
- User Data

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.

Scenarios are composed of alternatives, as well as other calculation options, 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.

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 a child alternative consists of data inherited from its parent, and the data altered specifically by you (local data).

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 will behave, 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.

Scenarios allow you to specify the alternatives you wish to analyze. 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.

Remember that all data inherited from the base alternative will be changed when the base alternative changes. Only local data specific to a child alternative will remain unchanged.

8.1.1 Alternatives Manager

The **Alternatives** dialog is the central location for managing the alternatives in your project. It allows you to edit, create, and manage the various types of alternatives. It also gives you more advanced capabilities, such as merging alternatives and creating child alternatives.

The available alternatives of each type are conveniently organized in a tree view. The network element data is grouped into the following types:

- Physical
- Demand
- Initial Settings
- Operational
- Age
- Constituent
- Trace
- Fire Flow
- Cost
- User Data

On the right side of the dialog are a number of buttons that provide functions for managing the alternatives. The following list provides a brief description of the function of each of these buttons.

Add – Create a new base alternative, first prompting for a name, and then opening an Alternatives editor. Base alternatives are initialized with the first data set entered either in tables or specific element dialogs.

Add Child – Create a new child alternative that inherits from the selected alternative. This allows you to automatically share the majority of the records from a parent alternative, while modifying only selected records in the child alternative.

Edit – Open the tabular record editor for the selected alternative. This tabular record contains all the values that are used by the selected alternative.

Merge – Move all records from the selected child alternative into its parent alternative, and then remove the selected alternative. The records in the selected alternative will replace the corresponding records in the parent. This is helpful when you have been experimenting with changes in a child alternative, and you want to permanently apply those changes to the parent alternative. All other alternatives that inherit data from that parent alternative will reflect these changes.

Rename – Rename an existing alternative. This invokes an in-place editor in the tree view of the available alternatives. Make the desired changes to the existing name and press the **Enter** key

Duplicate – Create a new alternative filled with records copied from the selected alternative. Use this if you wish to copy the data from an alternative, but not create a child. The two alternatives will be independent.

Delete – Remove the selected alternative and its records. Deleting an alternative will also delete all of the input data associated with that alternative.

Report – Generate a Print Preview of a summary report of the selected alternative, all alternatives, or the selected alternative and all of its children in that hierarchy.

> **You will not be allowed to merge or delete an alternative that is referenced by one or more scenarios. When you attempt to perform the operation, you will be provided with a list of the scenarios that reference the alternative.**
>
> **If you are attempting to merge an alternative that is referenced, you will need to edit the scenario(s) that references the child alternative that you are merging from, and make them reference the parent alternative that you are merging to. Use the Scenario Manager window to edit the scenario(s), and the Alternatives tab to make the scenario point to the parent alternative.**

To access the **Alternative Manager** dialog select **Analysis\Alternatives** from the pull-down menu.

8.1.2 Alternatives Editor

The Alternatives 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. The tables are a great way to edit data. 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 basically the same 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 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 will be reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

> **As you make changes to records, the check box will automatically become checked. If you want to reset a record to its parent's values, simply uncheck the corresponding check box.**
>
> **Many columns support Global Editing, allowing you to change all values in a single column. Right click a column header to access the Global Edit option.**
>
> **The checkbox column will be disabled when you edit a base alternative.**

To access the Alternatives Editor for a particular alternative, select it in the **Alternatives** dialog and click the **Edit** button.

8.1.3 Physical Alternative

One of the most common uses of a water distribution model is the design of new or replacement facilities. During design, it is common to try several physical alternatives in an effort to find the most cost effective solution. For example, when designing a replacement pipeline, it would be beneficial to try several sizes and pipe materials to find the most satisfactory combination. Our powerful **Alternative Manager** allows you to set up an unlimited number of design alternatives and apply them in different scenarios.

Each type of network element has a specific set of physical properties that are stored in a physical properties alternative, as listed below:

- Pipe Physical Properties
- Pump Physical Properties
- Valve Physical Properties
- Junction Physical Properties
- Reservoir Physical Properties
- Tank Physical Properties

Physical Alternative Editor for Pipes

The Physical Alternative editor for pipes is used to create various data sets for the physical characteristics of pipes. The following columns are available:

- **Material –** Type of material from which the pipe is constructed (e.g. Ductile Iron, PVC, Steel).
- **Diameter –** Internal diameter of the pipe. The nominal diameter of the pipe is commonly used in water distribution modeling with little practical impact.
- **Roughness** – A measure of the pipes internal roughness, based on the chosen friction method.
- **Minor Loss Coefficient** -Appurtenances such as valves, bends, and tees contribute to local flow disturbances resulting in energy loss. Click the **ellipsis (…)** button to edit the composite minor loss element for the pipe.

- **Check Valve** – True or false status indicating the presence of a check valve.

Physical Alternative Editor for Pumps

The Physical Alternative editor for pumps is used to create various data sets for the physical characteristics of pumps, which consists of the following:

- **Elevation** – Elevation of the pump, typically measured from the Mean Sea Level.
- **Pump Type** – The attributes that define the pump's operating characteristics. Click this field, then click the **ellipsis (…)** button to edit the parameters for the active pump type.

Physical Alternative Editor for Valves

The Physical Alternative editor for valves used to create various data sets for the physical characteristics of valves. The following columns are available:

- **Elevation** – Elevation of the valve, typically measured from the Mean Sea Level.
- **Diameter** – The internal diameter of the valve. The nominal diameter of the valve is commonly used in water distribution modeling with little practical impact.
- **Minor Loss Coefficient** – The wide-open minor loss coefficient. Click the **ellipsis (…)** button to edit the Minor Loss Library.

Physical Alternative Editor for Junctions

The Physical Alternative editor for PressureJunctions is used to create various data sets for the physical characteristics of junctions, which consist of the following:

- **Elevation** – Elevation of the junction, typically measured from the Mean Sea Level.
- **Zone** – Specify the zone the junction belongs to. You may click the **ellipsis (…)** button to access the Zone Manager, which allows you to edit or add zones.

Physical Alternative Editor for Reservoirs

The Physical Alternative editor for reservoirs is used to create various data sets for the physical characteristics of reservoirs, which consist of the following:

- **Elevation** -Elevation of the reservoir, typically measured from the Mean Sea Level.
- **Zone** – Specify the zone the tank belongs to. You may click the **ellipsis (…)** button to access the Zone Manager, which allows you to edit or add zones.

Physical Alternative Editor for Tanks

The Physical Alternative editor for tanks is used to create various data sets for the physical characteristics of tanks. The following columns are available:

- **Elevation** – Ground elevation of the tank.
- **Base Elevation** – The vertical distance of the tank's base above a known datum. Typically, Mean Sea Level is the datum used. The base elevation is the elevation from which all tank levels are computed.
- **Minimum Elevation (or Level)** – This is the lowest possible water surface elevation for the tank. If the tank drains below this level, it will shut off from the system.
- **Maximum Elevation (or Level)** – This is the highest possible water surface elevation for the tank. If the tank fills above this level, it will shut off from the system.

- **Section –** The physical parameters that define the tank cross sectional geometry. There are two (2) types of tank sections, Constant Area and Variable Area. Click this field, they click the **ellipsis (…)** button to edit the parameters for the active tank section type.
- **Zone –** Specify the zone the tank belongs to. You may click the **ellipsis (…)** button to access the Zone Manager, which allows you to edit or add zones.

8.1.4 Demand Alternative

The Demand Alternative allows you to model the response of the pipe network to different sets of demands, such as the current demand and the year 2010 demand.

The demand alternative table includes the following columns:

- **Label** – Identifying label of the junction element.
- **Type** – Demand type, **Demand** or **Inflow**. Direct editing of this item is disabled if the junction has multiple demands (see Demand Summary below).
- **Demand** – Hydraulic load attributed to the junction for Steady-State Analysis, or the hydraulic load before applying the Pattern time step multiplier used for Extended Period Analysis. If the junction has multiple demands, this field displays a single "calculated" Baseline Load, and direct editing of the field is disabled (see Demand Summary below).
- **Pattern** – Name of the Pattern that applies the time-step multiplier to the Baseline Load. If the junction has multiple demands, direct editing is disabled and the pattern name is shown as "Composite".
- **Demand Summary** – A summary displaying the calculated Baseline Load, the EPS Pattern applied to the Baseline Load, and the calculated demand Type (**Demand** or **Inflow**) for the junction. Clicking twice on a Demand Summary opens an editing dialog for working with multiple demands on a junction.

Setting up multiple demand alternatives makes it possible to easily manage different loading conditions for any network. For example, an "Average Day" demand alternative contains the average demands for each junction in the network, and a "Peak Day" demand alternative contains the peak demands for each junction in the network. The Alternative Manager allows you to create any number of demand alternatives.

This program allows multiple demands to be attributed to a single junction.

8.1.5 Initial Settings Alternative

The following types of network elements have initial settings:

- Pipes
- Pumps
- Tanks
- Pressure Valves
- FCVs (Flow Control Valves)
- TCVs (Throttle Control Valves)

Initial Settings Alternative Editor for Pipes

The Initial Settings Alternative for pipes is used to specify if the pipe status is initially Open or Closed.

Initial Settings Alternative Editor for Pumps

The Pump Initial Settings Alternative editor allows you to analyze various initial settings for pumps.

The fields for each record are as follows:

- **Status –** Indicates whether the pump is initially On or Off.
- **Relative Speed Factor** – Determines the initial speed of the pump impeller relative to the speed at which the pump curve is defined.

Initial Settings Alternative Editor for Tanks

The Tank Initial Settings Alternative editor allows you to analyze various water surface elevations (hydraulic grades) in your tank at the beginning of the simulation.

Initial Settings Alternative Editor for Pressure Valves

The Pressure Valve Initial Settings Alternative editor allows you to analyze various initial settings for pressure valves (Pressure Breaker, Pressure Reducer, and Pressure Sustaining Valves) .

The fields for each record are as follows:

- **Valve Status** – Indicates whether the pressure valve is initially Active, Inactive (wide-open), or Closed.
- **Hydraulic Grade / Pressure** – Initial setting for the valve. Depending on the input mode, the setting is entered and displayed in terms of hydraulic grade or pressure.

Initial Settings Alternative Editor for FCVs

The FCV Initial Settings Alternative editor allows you to analyze various initial settings for Flow Control Valves.

The fields for each record are as follows:

- **Valve Status** – Indicates whether the FCV is initially Active, Inactive, or Closed.
- **Discharge –** Initial flow setting for the valve.

Initial Settings Alternative Editor for TCVs

The TCV Initial Settings Alternative editor allows you to analyze various initial settings for Throttle Control Valves.

The fields for each record are as follows:

- **Valve Status –** Indicates whether the TCV is initially Active, Inactive, or Closed.
- **Headloss Coefficient –** Initial headloss coefficient for the valve.

8.1.6 Operational Alternative

The Operational Alternative allows you to specify controls on pressure pipes, pumps, as well as valves (in WaterCAD). The **Controlled** field contains a Boolean (true or false) statement that indicates whether the network element is controlled. Clicking in this field activates a button that allows you to access the **Controls** dialog and edit the controls for this element.

8.1.7 Age Alternative

The Age Alternative is used when performing a water quality analysis for modeling the age of the water through the pipe network. This alternatives allows you to analyze different scenarios for varying water ages at the network nodes.

8.1.8 Constituent Alternative

The Constituent Alternative contains the water quality data used to model a constituent concentration throughout the network when performing a water quality analysis.

Selecting a constituent from the **Constituent** scroll-down list provides default values for table entries. This software provides a user editable library of constituents for maintaining these values, which may be accessed by clicking the **ellipsis (…)** button next to the **Constituent** scroll-down list.

- **Label** – Identifying label of the element represented by the row.
- **Initial Constituent** – Concentration of the constituent at the beginning of the analysis.
- **Bulk Reaction –** Reaction rate constant used to model reactions of the constituent within the bulk flow.
- **Wall Reaction –** Reaction rate constant used to model reactions that occur with the material along the pipe wall.
- **Constituent Source –** True or false check to determine whether a node is a source of the constituent.
- **Constituent Baseline Load** – Load attributed to the element before applying the Pattern time step multiplier used for an Extended Period Analysis.
- **Constituent Pattern –** Name of the Pattern that applies the time step multiplier to the Baseline Load.

Depending on the type of highlighted network element in the table, the **Use Defaults** button will reset the reaction coefficients for that element to the constituent default values as specified in the constituent library, or it will reset the initial constituent concentrations to 0.0.

8.1.9 Trace Alternative

The Trace Alternative is used when performing a water quality analysis to determine the percentage of water at each node coming from a specified node. The Trace Alternative data includes a Trace Node, which is the node from which all tracing is computed.

The trace alternative table includes the following columns:

- **Label** – The identifying label of node elements.
- **Initial Trace –** A percentage value representing the starting condition at the node.

8.1.10 Fire Flow Alternative

The Fire Flow Alternative contains the input data required to perform a fire flow analysis. This data includes the set of junction nodes for which fire flow results are needed, the set of default values for all junctions included in the fire flow set, and a record for each junction node in the fire flow set.

Default Flow and Pressure Constraints

Each fire flow alternative has a set of default parameters that are applied to each junction in the fire flow set. When a default value is modified, you will be prompted to decide if the junction records that have been modified from the default should be updated to reflect the new default value.

The default constraints are grouped in the **Flow Constraints** and **Pressure Constraints** sections, as follows:

- **Needed Fire Flow –** Flow rate required at a fire flow junction to satisfy demands.
- **Fire Flow Upper Limit –** Maximum allowable fire flow that can occur at a withdrawal location. It will prevent the software from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures.
- **Apply Fire Flows By** – There are two methods for applying fire flow demands. The fire flow demand can be added to the junction's baseline demand, or it can completely replace the junction's baseline demand. The junction's baseline demand is defined by the Demand Alternative selected for use in the Scenario along with the fire flow alternative.
- **Residual Pressure** – Minimum residual pressure to occur at the junction node. The program determines the amount of fire flow available such that the residual pressure at the junction node does not fall below this target pressure.
- **Minimum Zone Pressure –** Minimum pressure to occur at all junction nodes within a zone. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone associated with it, which can be located in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another zone.
- **Use Minimum System Pressure Constraint –** Toggle indicating whether a minimum pressure is to be maintained throughout the entire pipe system.
- **Minimum System Pressure –** Minimum pressure allowed at any junction in the entire system as a result of the fire flow withdrawal. If the pressure at a node anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied.

Selection Set

Set of selected elements where fire flows need to be analyzed. You can choose between 'All Junctions' or a 'Subset of Junctions' that you can edit by clicking the **ellipsis (…)** button and accessing the Selection Set editor.

Fire Flow Loads

The table on the fire flow alternative editor displays the fire flow loads for the junctions in this set. The values in this table reflect the default values entered, unless a change is made. The columns in the table are as follows:

- **Label –** Label of the junction whose fire flow record is being displayed.
- **Needed Fire Flow –** Flow rate required at the junction to meet fire flow demands. This value will be added to the junction's baseline demand, or it will replace the junction's baseline demand, depending on the default setting for applying fire flows.

- **Fire Flow Upper Limit** – Maximum allowable fire flow that can occur at a withdrawal location. This value will prevent the software from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures. This value will be added to the junction's baseline demand, or it will replace the junction's baseline demand, depending on the default setting for applying fire flows.
- **Residual Pressure, Minimum Zone Pressure, Minimum System Pressure** – Set of pressure constraints for each Fire Flow node. See description for each of these parameters in the **Default Flow and Pressure Constraints** section above.

Use Defaults

Click the **Use Defaults** button to reset the selected row to the default values for the Fire Flow Alternative. This does not cause the record to inherit from its parent, only causes it to reflect the Default values.

The Defaults for a fire flow alternative can only be set in the root alternative. All child alternatives inherit the Default values from the root. The set of junction nodes is also inherited from the root and cannot be altered in the child alternatives.

8.1.11 Cost Alternative

The Cost Alternative contains the data allowing you to perform cost estimating of the whole network or parts of it.

This dialog contains the following tabs:

- Pressure Pipe
- Nodal Elements (Pressure Junction, Pump, Valve, Reservoir, and Tank)

Pressure Pipe Cost Alternative

The **Pressure Pipe** tab in the **Cost Alternative** editor contain the data associated with pipe cost estimating, as described below:

- **Include Element in Cost Calculation** – Toggle allowing you to estimate the cost of only part of the network.
- **Total Fixed Cost –** Cost of this pipe resulting from any fixed cost as defined in the **Fixed Cost Table**, accessed by double-clicking in the **Fixed Cost Description** field.
- **Fixed Cost Description** – Button accessing the **Fixed Cost Table**. The button is labeled as Simple or Composite when there are more than one fixed cost item for this pipe.
- **Unit Cost Function –** Allow you to associate a unit cost function selected from the scroll-down list. You may click the **ellipsis (…)** button to access the **Unit Cost Functions** Manager, which allows you to edit or add unit cost functions, in a tabular or formula format based on a pipe attribute, such as pipe diameter.
- **Additional Unit Cost –** You may also define or add a unit cost specific to this pipe.

Nodal Element Cost Alternative

The **Pressure Junction**, **Pump**, **Valve**, **Reservoir**, and **Tank** tabs in the **Cost Alternative** editor contain the data associated with the cost estimating for each nodal element, as described below:

- **Include Element in Cost Calculation –** Toggle allowing you to estimate the cost of only part of the network.

- **Total Fixed Cost** – Cost of this element resulting from any fixed cost as defined in the **Fixed Cost Table** accessed by double-clicking in the **Fixed Cost Description** field.
- **Fixed Cost Description** – Click on this field for a button accessing the **Fixed Cost Table**. The button is labeled as Simple, or Composite when there are more than one fixed cost itsem for this element.

8.1.12 User Data Alternative

The User Data Alternative allows you to edit the data defined in the User Data Extension for each of the network element types. The User Data Alternative editor contains a tab for each type of network element.

8.2 Scenarios

A Scenario contains all the input data, calculation options, results, and notes associated with a set of calculations. A set of calculations may include a Hydraulic, a Fire Flow or Water Quality Calculation, and a Cost Analysis calculation run. 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 as often as you wish.

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.

There are two types of scenarios:

- **Base scenarios** – Contain all of your working data. When you start a new project, you will 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 even 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) elements in child scenarios.

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.

8.2.1 Scenario Selection

You can change the current scenario by simply using the **Scenario** drop-down list located on the Analysis Toolbar on the main application window. When you select a different scenario, your current input data, calculation options, and calculated results (if available) will reflect the selected scenario and the alternatives it references.

8.2.2 Editing Scenarios

Once scenarios and alternatives are created, you do not need to take any special steps to input data into the alternatives referenced by the current scenario. This happens automatically as you make changes to your data. Changes to your data are always applied to the alternatives in your active scenario. For example, consider that a pipe has a 6" diameter in the alternative storing data for the Base scenario. Then you switch to Scenario 2, which references another alternative, and change the pipe diameter to 8". The new value will automatically be associated with the alternative in Scenario 2. If you switch back to the Base scenario, the pipe diameter will revert to 6".

You can also enter data directly into an alternative using the Alternatives Editor. This editor allows you to see all of the changes that you have made in a single alternative. If you make an unintended change to the active child scenario and you wish to remove it, go to the tabular editor for the type of input data you changed and uncheck the leading check box on the record(s) for the elements you wish to restore.

Scenarios currently only track modifications to the input data associated with existing network elements. They do not allow you to track modifications to the network topology itself (e.g. additions and deletions of network elements).

8.2.3 Scenario Manager

The **Scenario Manager** allows you to 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.

The **Scenario Manager** window is divided into four sections:

1. The three buttons that run across the top of the window:

   - **Tutorial** – Open the tutorials.
   - **Close –** Close the **Scenario Manager**.
   - **Help** – Open the on-line help.

2. The series of five buttons running along the left side of the window:

- **Scenario Wizard –** Open the **Scenario Wizard**, which walks you step-by-step through the creation of a new scenario.
- **Scenario Management –** Offers a menu of options for creating, editing, and managing scenarios:
  - **Add –** Prompts for a name, then creates a new child or base scenario. If you create a child scenario, it will be based on the scenario that is currently highlighted.
  - **Edit –** Open the **Scenario Editor** dialog for the scenario that is currently highlighted.
  - **Rename –** Rename an existing scenario. Note this invokes an in-place editor in the tree view of the available scenarios. Make the desired changes to the existing name and press **Enter**.
  - **Delete –** Delete the scenario that is currently highlighted.
  - **Report –** Generate a summary report for the scenario that is highlighted, including alternatives, calculation options, notes, and results.

- **Alternative –** Open the **Alternative Manager** for creating, editing, and managing alternatives.
- **Batch Run –** Open the **Batch Run** dialog for selecting from among the available scenarios and initiating calculations.
- **Scenario Comparison –** Open the **Annotation Comparison Wizard**, which allows you to create drawings displaying the differences in input and output variables between two scenarios.

3. The pane in the center of the dialog:

- **Scenarios Pane** – Available scenarios in a hierarchical tree showing the parent-child relationships. You can right-click any scenario to perform scenario management functions on it. You can double-click parent scenarios to expand or collapse the child scenarios beneath them.

4. The pane on the right side of the dialog, which displays a variety of information depending on which of the following tabs is selected:

- **Alternatives –** Alternatives referenced by the highlighted scenario, showing the type and name for each alternative. An icon distinguishes whether the alternative belongs to the scenario or is inherited from its parent scenario. Double-click any alternative to open the Alternatives editor.
- **Summary –** Summary of the calculation options for the highlighted scenario and any notes you have associated with it.
- **Results –** Summary of the last calculation performed for the highlighted scenario.

> **When you delete a scenario, keep in mind that because scenarios never actually hold calculation data records (alternatives do), you are not losing data records. The alternatives and data records referenced by that scenario will still exist until you explicitly delete them. By accessing the Alternative Manager, you can delete the referenced alternatives and data records.**

To open the **Scenario Manager** window, select **Analysis\Scenarios**. Or, click the button next to the scenario drop-down list in the main application window.

Batch Run

Performing a batch run allows you to set up and run multiple scenarios at once. This is helpful if you want to queue a large number of calculations, or simply manage a group of smaller calculations as a set. The list of selected scenarios for the batch run will remain with your project until you change it.

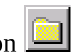
Using the dialog is simple. First, check the scenarios you want to run and click the **Batch** button. Each scenario will be calculated. You can cancel the batch run between any scenario calculation.

When the batch is completed, the scenario that was current will remain current, even if it was not one that was calculated. Select a calculated scenario from the main window drop-down list to see the results throughout the program, or select it from the **Scenario Manager** and click the **Results** tab to preview the results.

Creating Scenarios to Model "What-if" Situations?

The scenario management feature was designed to let you model "what-if" situations by easily switching between different input data sets without having to re-enter data, and by comparing different output results just as easily.

To create a new scenario:

1. Open the **Scenario Manager** dialog by clicking the Scenario Manager button  next to the drop-down scenario list in the main application window.
2. Open the **Scenario Wizard** by clicking its button in the upper left of the **Scenario Manager** dialog.
3. Complete each step in the **Scenario Wizard –** Name the new scenario, choose which scenario to base it on, and choose the alternatives to be included. Click **Next** between each step, and click **Finish** when you are done.
4. Close the **Scenario Manager** dialog. Notice the scenario you have just created is displayed as the current scenario in the **Scenario** drop-down list in the main application window.
5. Proceed to modify your model with the changes you want recorded in the new scenario.

8.2.4 Scenario Wizard

The Scenario Wizard will guide you step-by-step through the process of creating a new scenario.

These are the basic steps for creating a new scenario:

- **Name –** Name the scenario, and add some comments if you wish.
- **Base –** Select a scenario on which to base the new scenario.
- **Calculation –** Choose the type of calculation (Steady State or Extended Period) that you would like to perform, as well as other calculation options.
- **Alternatives –** Specify the alternative types with which you would like to work.
- **New/Existing** – Create and/or Select alternatives for your new scenario.
- **Preview** – Preview the scenario, and create it when satisfied.

To access the Scenario Wizard, open the **Scenario Manager** and click the **Scenario Wizard** button.

Scenario Wizard – Step 1

Here you can enter a unique name and an optional note for the new scenario that you are creating.

The name field allows you to input a distinguishing name for this scenario. A default name is provided, but we recommend that you change it to something more descriptive. If the new scenario will be based on another scenario, you may want a name that indicates what will be different about the new scenario. For example: "Post Development".

The next field is optional, and allows you to input free-form text that will be associated with the new scenario. Use it to make detailed notes about the conditions the scenario will model.

Click the **Next** button to proceed to the next step in defining a new scenario.

Scenario Wizard – Step 2

Click the existing scenario on which you would like to base your new scenario. Your new child scenario will inherit data from this parent scenario, and will be initialized with the same calculation settings and options. The **Scenario Wizard**, designed to introduce the user to scenarios, does not allow you to create new base scenarios.

Existing scenarios in your project are displayed in a "tree" structure, giving you a graphic depiction of the parent-child relationships.

Press the **Next** button to proceed to the next step.

Scenario Wizard – Step 3

This step of the **Scenario Wizard** allows you to specify the type of calculation to be associated with the scenario you are creating.

If you select to be in Steady State mode, you are also given the option to perform an automated fire flow analysis.

If you select to be in Extended Period mode, you are also given the option to perform one of the Water Quality Analysis (Age, Constituent, or Trace analysis).

Scenario Wizard – Step 4

Check the boxes next to the types of alternatives you want to include in the new scenario. The alternatives for boxes you do not check will be inherited from the specified parent scenario. You will be free to add or remove alternatives to the scenario after you create it.

Click the **Next** button to proceed to the next step in defining a new scenario.

Scenario Wizard – Step 5

Here you are asked to specify the source for each alternative you have requested in the previous tab.

Create New Alternative – If you choose to create a new alternative, it will inherit from the same type of alternative in the specified or parent scenario – meaning it will initially use all the same input data values. Enter a unique and descriptive name for the new alternative.

Use Existing Alternative – If you choose to use an existing alternative, you will be shown the tree of existing alternatives from which to choose. In this case you will not be creating a new alternative for use in the scenario, and instead may actually be *sharing* an alternative with another scenario.

Click the **Next** button to proceed to the next step.

Scenario Wizard – Step 6

The last step of the **Scenario Wizard** displays a summary of the scenario you have defined and are about to create.

In the left pane is a preview of the scenario as it relates to its parent and other scenarios. In the right pane is a list of the alternatives it references, showing their labels and types. An icon indicates whether a given alternative is local to the new child scenario, or if it is inherited from the specified or parent scenario.

If you are satisfied, click the **Finished** button to create the new scenario.

8.2.5 Scenario Editor

The **Scenario Editor** dialog is the control center for each analysis. It is the place where you access or change all the information for performing a single calculation (alternatives, calculation type, calculation options, results, and notes). It is organized in the following tabs:

- **Alternatives –** Edit or view the alternatives to be used by this scenario.
- **Calculation –** Specify the type of hydraulic/water quality calculations to be performed, and click the **GO** button to run these calculations.
- **Results –** View the hydraulic/water quality calculation results summary.
- **Notes** – Edit or view notes for this scenario.

To open the **Scenario Editor** dialog for the active scenario, press the **GO** button from the toolbar. To open the **Scenario Editor** dialog for any scenario, select **Analysis\Scenarios** from the pull-down menu to open the **Scenario Manager**, right-click the scenario that you wish to edit, and select **Edit** from the pull down menu that appears. Or highlight the scenario you wish to edit, press the **Scenario Management** button, and select **Edit**.

Scenario Editor – Alternatives Tab

The **Alternatives** tab, located in the **Scenario Editor**, allows you to specify the alternatives that will be used by this scenario. There is one row for each Alternative Type. You need only concern yourself with the rows that correspond to the changes you would like to model using this scenario.

To specify the alternatives you would like to work with, simply click the check box next to the alternative type. For example, if you would like to see how your system behaves by changing the shape or sizes of a few pipes, then click the check box next to the Physical alternative row.

If you would like to use an existing alternative that you have already set up, use the drop-down list to choose the desired alternative. If you would like to create a new alternative, click the **New** button. You will be asked to name the new alternative, and the **Alternatives Editor** will open.

The **Scenario Wizard** will walk you through all of the steps required to create a new scenario. If you are unsure how to specify the alternatives that you would like to work with, we recommend that you use this wizard.

Remember, when this scenario is active, the alternatives that you specify here will be active.

Changes that you make to your model will be made in these alternatives.

When you calculate this scenario, these are the alternatives that will be used.

This tab will take on a different appearance depending on whether you are editing a Base scenario or Child scenario. When editing a base scenario, the checkbox column (described above) will not be present. You can use the ellipsis (…) button located to the right of each drop-down list to access the associated Alternatives Manager.

Scenario Editor – Calculation Tab

This dialog is the control center for each network analysis. This program is capable of performing both a *Hydraulic Analysis* and a *Water Quality Analysis.* Also, *Extended Period Analysis* which considers time-variable hydraulic demands and constituent source concentrations is available. Patterns are used to define the time-variable aspects of these system loads. Also contributing to time-variable hydraulic conditions are tank characteristics and controls associated with Pipes, Pumps, and Valves.

This dialog allows you to specify the following data and calculation modes:

- **Steady State/Extended Period Simulation –** If you have selected Extended Period calculation, a set of extended period options become available for editing: **Start Time**, **Duration**, and **Hydraulic Time Step**.
- **Analysis Modes –** In addition to performing a standard hydraulic analysis, you are given the option to perform a Water Quality Analysis (in Steady State mode) or a Fire Flow Analysis (in Extended Period Simulation mode).
- **Calibration** – Feature used to "tweak" the two most commonly used parameters during model calibration: junction demands and pipe roughness, without permanently changing the value of the input data. It allows you to experiment with different calibration factors until you find the one that causes your calculation results to most closely correspond with your observed field data.
- **Check Data/Validate –** This feature, allows you to validate your model against typical data entry errors, hard to detect topology problems, and modeling problems. When the **Validate** box is checked, the model validation is automatically run prior to calculations. It can also be run at any time by clicking the **Check Data** button. The process will produce either a dialog stating "No Problems Found", or a status log with a list of messages.
- **Calculation Options –** Specify parameters affecting hydraulic and water quality calculations.

Clicking the **GO** button will perform the calculations.

The GO button, the Validate toggle and the Check Data button are not available when the Scenario Editor is accessed from the Scenario Manager. In order to validate the data or calculate the current scenario, the Scenario Editor should be accessed by clicking the GO button on the main window.

Calibration is one of the most important steps in developing a hydraulic and/or water quality model. This program provides an easy-to-use calibration feature that lets you "tweak" the input data to help you match data observed in the field.

Each calculation depends upon a number of parameters that can optionally be configured using the Calculation Options dialog.

If the model has not been calculated, or if the input data has been changed since the last calculation, the word Compute (displayed in Red) will appear in the status pane in the lower right corner of the main editing window. This is a signal that the model needs to be recalculated.

Scenario Editor – Results Tab

The **Results** tab contains a summary of the last calculation performed using this scenario. Click the **Save** button to save the results to an ASCII text file. Click the **Print Preview** button to preview the Scenario Results Summary Report.

Immediately after you run the calculations, the Results tab automatically displays. You will notice a green, yellow, or red light in that tab indicating how successful the computations were. This light is not displayed once you close this dialog box.

Scenario Editor – Notes Tab

The memo field on the **Notes** tab allows you to input free-form paragraph text that will be associated with the new scenario. Use it to make detailed notes about the conditions that the scenario will model.

Chapter 9

Modeling Capabilities

WaterCAD provides unmatched modeling capabilities, allowing you to model and optimize practically any distribution system aspect, including the following operations:

Hydraulic Analysis

- Perform a steady-state analysis for a "snapshot" view of the system, or perform an extended-period simulation to see how the system behaves over time.
- Use any common friction method: Hazen-Williams, Darcy-Weisbach, or Manning's.
- Take advantage of scenario management to see how your system reacts to different demand and physical conditions, including fire and emergency usage.
- Control pressure and flow completely by using flexible valve configurations. You can automatically control pipe, valve, and pump status based on changes in system pressure (or based on the time of day). Control pumps, pipes, and valves based on any pressure junction or tank in the distribution system.
- Perform automated fire flow analysis for any set of elements and zones in the network.

Water Quality Analysis

- Track the growth or decay of substances (such as chlorine) as they travel through the distribution network.
- Determine the age of water anywhere in the network.
- Identify source trends throughout the system.

9.1 Steady-State/Extended Period Simulation

WaterCAD gives the choice between performing a steady-state analysis of the system or performing an extended-period simulation over any time period.

Steady-State Simulation

Steady-state analyses determine the operating behavior of the system at a specific point in time or under steady-state conditions (flow rates and hydraulic grades remain constant over time). This type of analysis can be useful for determining pressures and flow rates under minimum, average, peak, or short term effects on the system due to fire flows.

For this type of analysis, the network equations are determined and solved with tanks being treated as fixed grade boundaries. The results that are obtained from this type of analysis are instantaneous values and may or may not be representative of the values of the system a few hours, or even a few minutes, later in time.

Extended Period Simulation

When the effects on the system over time are important, an extended period simulation is appropriate. This type of analysis allows you to model tanks filling and draining, regulating valves opening and closing, and pressures and flow rates changing throughout the system in response to varying demand conditions and automatic control strategies formulated by the modeler.

While a steady-state model may tell whether or not the system has the capability to meet a certain average demand, an extended period simulation indicates whether or not the system has the ability to provide acceptable levels of service over a period of minutes, hours, or days. Extended period simulations can also be used for energy consumption and cost studies, as well as water quality modeling.

Data requirements for extended period simulations are greater than for steady-state runs. In addition to the information required by a steady-state model, you also need to determine water usage patterns, more detailed tank information, and operational rules for pumps and valves.

The following additional information is required only when performing Extended Period Simulation, and therefore is not enabled when Steady-State Analysis has been specified.

- **Start Time** – The start time format is military time, so 0.00 hr. is midnight and 24.00 hr. is midnight of the following day. This very powerful feature allows you to run an extended period analysis for any period of the day without redefining any time dependent input data such as patterns.
- **Duration** – The duration can be any positive real number.
- **Hydraulic Time Step** – Enter the time interval between hydraulic solutions for this calculation. The hydraulic time step is the maximum amount of time that the hydraulic conditions of the network are assumed to be constant.

Each of the parameters needed for an extended period analysis has a default value. You will most likely want to change the values to suit your particular analysis.

Occasionally the numerical engine will not converge during an extended period analysis. This is usually due to controls (typically based on tank elevations) or control valves (typically pressure regulating valves) toggling between two operational modes (on/off for pump controls, open/closed for pipe controls, active/closed for valves). When this occurs, try adjusting the hydraulic time step to a smaller value. This will minimize the differences in boundary conditions between time steps, and may allow for convergence.

Whether the model is run in steady-state or over an extended period is specified on the **Calculation** tab of the **Scenario Editor**, which can be accessed from the **Analysis\Scenarios** menu item.

9.2 Optional Analysis

In addition to performing a standard hydraulic analysis, you are given the option to perform a water quality analysis or a fire flow analysis:

- **Water Quality Analysis** – This check box configures the calculation to analyze for water quality. When this box is checked, you need to specify the type of water quality analysis to perform. This software is capable of performing three types of water quality analyses:
  - **Age** – Determine how long the water has been in the system.
  - **Constituent –** Determine the concentration of a constituent at all nodes and links in the system.
  - **Trace** – Determine the percentage of the water at all nodes and links in the system. The source is designated as a specific node.
- **Fire Flow Analysis** – This check box configures WaterCAD to analyze the system for available fire flow.

Water quality calculations are time variable in nature, and therefore are only available when the calculation is configured for extended period analysis. Be sure that the Extended Period Analysis radio button in the Hydraulic Analysis portion of the Calculation dialog is selected.

Fire Flow calculations are based on a steady-state calculation. Therefore, if the calculation is configured to perform an Extended Period Analysis, the Fire Flow Analysis check box is disabled. Be sure that the Steady State Analysis radio button in the Hydraulic Analysis portion of the dialog is selected.

Use the Scenario Manager to set up and manage multiple water quality data sets.

Use the Alternative Manager to set up and maintain multiple Fire Flow data sets.

These optional analyses are specified in the **Analysis** section in the **Calculation** tab of the **Scenario Editor**, which can be accessed from the **Analysis\Scenarios** menu item.

9.3 Calibration

Calibration is an important part of the development of hydraulic and water quality models. It is a powerful feature for "tweaking" the two most commonly used parameters during model calibration: junction demands and pipe roughness.

One of the first steps performed during a calculation is the transformation of the input data into the required format for the numerical analysis engine. If a factor and operator are present in the calibration fields when the **GO** button is clicked, the factor is used during this transformation. This does not permanently change the value of the input data, but allows you to experiment with different calibration factors until you find the one that causes your calculation results to most closely correspond with your observed field data.

The **Calibration** section contains the following data:

- **Demand** – Use this calibration field to temporarily adjust the individual demands at all junction nodes in the system that have demands for the current scenario. For example, assume node J-10 has two demands, a 100 gpm fixed pattern demand, and a 200 gpm residential pattern demand, for a total baseline demand of 300 gpm. If you enter a demand calibration multiplier of 1.25, the input to the numerical engine will be 125 gpm and 250 gpm respectively, for a total baseline demand of 375 gpm at node J-10. If you use the Set operator to set the demands to 400, the demand will be adjusted proportionally to become 133 and 267 gpm, for a total baseline of 400 gpm. Also, if a junction has an inflow of 100 gpm (or a demand of -100 gpm), and the calibration operation is Set Demand to 200 gpm, then the inflow at that junction will be -200 gpm (equivalent to a demand of 200 gpm).

- **Roughness** – Use this calibration field to temporarily adjust the roughness of all pipes in the distribution network.

- **Apply –** Click the **Apply** button to permanently adjust the demands or roughness. Generally, you will use this button after experimenting with the calibration factors on successive calculations, and comparing the calculation results to observed field data.

The calibration data is located on the **Calculation** tab of the **Scenario Editor**, which can be accessed from the **Analysis\Scenarios** menu item.

9.4 Check Data/Validate

This feature allows you to validate your model against typical data entry errors, hard to detect topology problems, and modeling problems. When the **Validate** box is checked, the model validation is automatically run prior to calculations. It can also be run at any time by clicking the **Check Data** button. The process will produce either a dialog stating "No Problems Found" or a status log with a list of messages.

The validation process will generate two types of messages. A warning message means that a particular part of the model (i.e. a pipe's roughness) does not conform to the expected value, or is not within the expected range of values. This type of warning is useful but not fatal. Therefore, no corrective action is required to proceed with a calculation. Warning messages are often generated as a result of a topographical or data entry error and should be corrected. An error message, on the other hand, is a fatal error, and the calculation cannot proceed before it is corrected. Typically, error messages are related to problems in the network topology, such as a pump or valve not being connected on both its intake and discharge sides.

The check data algorithm performs the following validations:

- **Network topology** – Checks that the network contains at least one boundary node, one pipe, and one junction. These are the minimum network requirements. It also checks for fully connected pumps and valves and that every node is reachable from a boundary node through open links.

- **Element validation –** Checks that every element in the network is valid for the calculation. For example, this validation ensures that all pipes have a non-zero length, a non-zero diameter, a roughness value that is within the expected range, etc. Each type of element has its own checklist. This same validation is performed when you edit an element in a dialog. The dialog will not close until each item on the checklist is satisfied.

In earlier versions of the software, it was possible to create a topological situation that was problematic but was not checked for in the network topology validation. The situation could be created by "morphing" a node element such as a junction, tank, or reservoir into a pump or valve. This situation is now detected and corrected automatically, but it is strongly recommended that you verify the flow direction of the pump or valve in question. If you have further questions or comments related to this, please contact Haestad Methods Support.

Warning messages related to the value of a particular attribute being outside the accepted range can often be corrected by adjusting the allowable range for that attribute.

The **Check Data** button and the **Validate** check box are located on the **Calculation** tab of the **Scenario Editor**, which can be accessed from the **Analysis\Scenarios** menu item.

9.5 Calculate Network

The following needs to be completed before performing hydraulic calculations for a network.

1. Set the Calculation mode to **Steady-State** or **Extended Period**. If **Extended Period** is selected, then specify the starting time, the duration, and the time step to be used.
2. Optionally, in Extended Period mode, you may perform a Water Quality Analysis. Set the **Water Quality** toggle On and select one of the three available types of calculations: **Age**, **Constituent** or **Trace**.
3. Optionally, in Steady-State mode, you may also perform a Fire Flow Analysis by setting the **Fire Flow Analysis** toggle.
4. Optionally, in the **Calibration** section, you may modify the demand or roughness values of your entire network for calibration purposes. If a factor and operator are present in the calibration fields when the **GO** button is clicked, the factor is used during this calculation. This does not permanently change the value of the input data, but allows you to experiment with different calibration factors until you find the one that causes your calculation results to most closely correspond with your observed field data. To permanently change the value of the input data, select **Apply**.
5. Optionally, click the **Options** button to verify general algorithm parameters used to perform Hydraulic and Water Quality calculations.
6. Set the **Validate** toggle On, or click the **Check Data** button to ensure that your input data does not contain errors.
7. Click on the **GO** button to start the calculations.

The Check Data button performs a quick check of your input data and displays any errors found. It is recommended to run this function before the actual run of the calculations. Note, however, that the data is automatically checked when you perform the calculations if the Validate toggle is On.

9.6 Fire Flow Analysis

One of the goals of a water distribution system is to provide adequate capacity to fight fires. WaterCAD's powerful fire flow analysis capabilities can be used to determine if the system can meet the fire flow demands while maintaining various pressure constraints. Fire flows can be computed for a single node, a group of selected nodes, or all nodes in the system. A complete fire flow analysis can comprise hundreds or thousands of individual flow solutions – one for each junction selected for the fire flow analysis.

Fire flows are computed at user-specified locations by iteratively assigning demands and computing system pressures. The model will check the computed fire flow and ensure that all pressure constraints are met. New demand will automatically be assigned and system pressures recomputed if one of the constraints is violated. Iterations continue until one of the fire flow constraints is matched, or until the maximum number of iterations is reached.

The flow is not actually withdrawn from the node. The baseline pressures are the pressures that are modeled under the standard steady-state demand conditions in which fire flows are not exerted. The analysis only serves as a check to determine how much can be withdrawn given the pressure constraints (Residual Pressure, Minimum Zone Pressure and optionally, Minimum System Pressure). The program will compute the available fire flow to be zero in situations where the baseline system pressures violate one of the pressure constraints.

Results of fire flow calculations, which are obtained from calculations performed separately for an automatic batch run, are only reported in the Fire Flow tab and in the Fire Flow Tabular Report (accessed from the Report\Tables menu or the Tabular Reports icon). Results reported in the other element editor tabs do not take into account any fire flow, unless you explicitly entered the fire flow as a demand at a specific junction.

All parameters defining a fire flow analysis, such as the residual pressure or the minimum zone pressure, are explained in detail in the Fire Flow Alternative and in the Fire Flow tab topics.

An on-line tutorial on fire flow can be found by selecting the Help\Tutorials menu.

9.6.1 Fire Flow Results

After performing a fire flow analysis, calculation results are available for each junction node in the fire flow selection set. These results can be viewed in the predefined **Fire Flow Report** (in tabular format), accessed by clicking the **Tabular Reports** button highlighting **Fire Flow Report**, and clicking **OK**. Note that results for the nodes that were not included in the fire flow selection set are reported as N/A.

9.6.2 Not getting Fire Flow at a Junction Node

Perform the following checks if you are not getting expected fire flow results:

1. Check the Available Fire Flow. If it is lower than the Needed Fire Flow, the fire flow conditions for that node are not satisfied. Therefore, **Needed Fire Flow Constraints** is false.

2. Check the Calculated Residual Pressure. If it is lower than the Residual Pressure Constraint, the fire flow condition for that node is not satisfied. Therefore, **Needed Fire Flow Constraints** is false.

3. Check the Calculated Minimum Zone Pressure. If it is lower than the Minimum Zone Pressure Constraint, the fire flow condition for that node is not satisfied. Therefore, **Needed Fire Flow Constraints** is false.

If you are not concerned about the pressure of a node that is NOT meeting the Minimum Zone Pressure constraint, move this node to another zone. Now, the node will not be analyzed as part of the same zone.

4. If you checked the box for **Minimum System Pressure Constraint** in the **Fire Flow Alternative** dialog, check to see if the Calculated Minimum System Pressure is lower than the set constraint. If it is, **Satisfies Fire Flow Constraints** will be False.

9.7 Water Quality Analysis

9.7.1 Age Analysis

An age analysis determines how long the water has been in the system and is more of a general water quality indicator than a measurement of any specific quality. To configure for an age analysis:

1. Choose **Compute** from the **Analysis** menu or click the **GO** button.

2. Activate the **Extended Period** radio button.

3. Check the box labeled **Water Quality Analysis**.

4. Select the **Age** radio button.

5. Assuming you have not already set up an Age alternative for this scenario (including defining the trace node), go to the **Alternatives** tab, click the **ellipsis (…)** or **New** button next to the **Age** scroll-down list, and add or edit an Age alternative. Back in the **Alternatives** tab, choose the desired alternative from the **Age Alternative** choice list.

6. Return to the **Calculation** tab and click the **GO** button.

Water quality analysis can only be performed for extended period simulations.

9.7.2 Constituent Analysis

A constituent is any substance, such as chlorine and fluoride, for which the growth or decay can be adequately described through the use of a bulk reaction coefficient and a wall reaction coefficient. A constituent analysis determines the concentration of a constituent at all nodes and links in the system. Constituent analyses can be used to determine chlorine residuals throughout the system under present chlorination schedules, or can be used to determine probable behavior of the system under proposed chlorination schedules. To configure for a constituent analysis:

1. Choose **Compute** from the **Analysis** menu or click the **GO** button.

2. Activate the **Extended Period** radio button.

3. Check the box labeled **Water Quality Analysis**.

4. Select the **Constituent** radio button.

5. Assuming you have not already set up a Constituent alternative for this scenario (including the selection of the constituent), go to the **Alternatives** tab, click the **ellipses (…)** or **New** button next to the **Constituent** scroll-down list, and add or edit a Constituent alternative. Specify the Constituent, which is defined in the Constituent Library and accessed by clicking the **ellipsis (…)** button. You are missing a few Back in the **Alternatives** tab, choose the desired alternative from the **Constituent Alternative** choice list.

6. Return to the **Calculation** tab and Click the **GO** button.

9.7.3 Trace Analysis

A trace analysis determines the percentage of the water at all nodes and links in the system. The source is designated as a specific node in the system and is called the trace node. In systems with more than one source, it is common to perform multiple trace analyses using the various trace nodes in successive analyses. The source node and initial traces are specified in the **Trace Alternative** dialog. To configure for a trace analysis:

1. Choose **Compute** from the **Analysis** menu, or click the **GO** button.

2. Activate the **Extended Period** radio button.

3. Check the box labeled **Water Quality Analysis**.

4. Select the **Trace** radio button.

5. Assuming you have not already set up a Trace alternative for this scenario (including defining the trace node), go to the **Alternatives** tab, click the **ellipses (…)** or **New** button next to the **Trace** scroll-down list, and add or edit a trace alternative. Specify the trace node to be used for this analysis and provide the appropriate data. Back in the **Alternatives** tab, choose the desired alternative from the **Trace Alternative** choice list.

6. Return to the **Calculation** tab and click the **GO** button.

Water quality analysis can only be performed for extended period simulations.

9.8 Calculation Options

Calculations depend on a variety of parameters that may be configured by you.

This program provides defaults for each of the calculation options. If you make changes to the calculation options and decide that you would like to return to the default settings, use the **Reset** button on the **Calculation Options** dialog.

The dialog is divided into two groups:

- Hydraulics
- Water Quality

To access the **Calculation Options** dialog, click the **Options** button on the **Calculation** tab of the **Scenario Editor**, or select the **GO** button and click **Options**.

9.8.1 Hydraulic Analysis Options

The following hydraulic analysis parameters are available for user configuration:

- **Trials** – Unitless number that defines the maximum number of iterations to be performed for each hydraulic solution. The default value is 40.
- **Accuracy –** Unitless number that defines the convergence criteria for the iterative solution of the network hydraulic equations. When the sum of the absolute flow changes between successive iterations in all links is divided by the sum of the absolute flows in all links, and is less than the Accuracy, the solution is said to have converged. The default value is 0.001 and the minimum allowed value for Accuracy is 1.0e-5.

The number of trials specifies the maximum number of iterations to be performed for each time step in an extended period simulation, not the total number of iterations for the entire analysis.

In most cases, the default values are adequate for the hydraulic analysis. Under special circumstances, the accuracy may need to be adjusted downward. This is necessary when the model converges, yet there are larger than acceptable discrepancies between the total inflow and outflow at individual nodes.

9.8.2 Water Quality Analysis Options

The following water quality analysis parameters are available for user configuration:

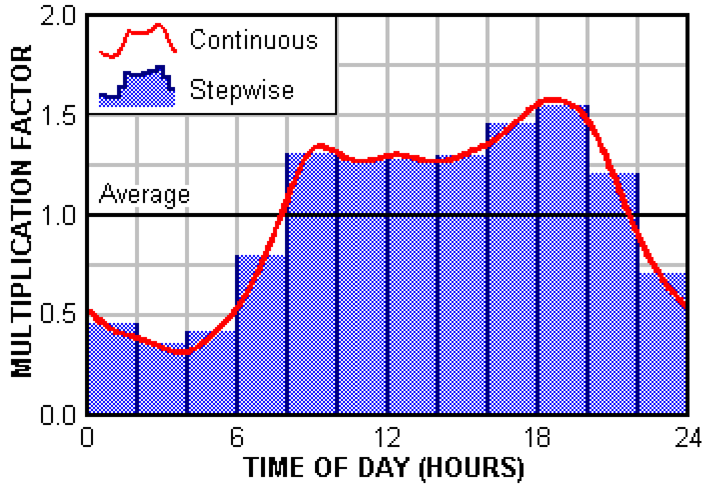
- **Maximum Pipe Segments –** Unitless number that defines the maximum number of segments that any pipe can be separated into during a water quality analysis. The default value is 100.
- **Minimum Pipe Travel Time** – Minimum time of travel through a pipe. If travel time through any pipe is computed to be less than this number, it is set equal to it. The default value for minimum pipe travel time is 10% of the hydraulic time step, and default units are hours.
- **Set Quality Time Step** – Check this box if you want to manually set the water quality time step. By default, this box is not checked and the water quality time step is computed internally by the numerical engine.
- **Quality Time Step –** Time interval used to track water quality changes throughout the network. By default, this value is computed by the numerical engine and is equivalent to the smallest travel time through any pipe in the system.

9.9 Patterns

The extended period analysis is actually a series of steady-state analyses run against time-variable hydraulic demands and constituent source concentrations. Patterns allow you to apply automatic time-variable changes within the system. The most common application of patterns is for residential or industrial demands. Diurnal curves are patterns which relate to the changes in demand 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.

Typical Diurnal Curve



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 time, 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. On the continuous pattern in the figure above, the value 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.

9.9.1 Pattern Manager

Patterns provide an effective means of applying time-variable system loads to the distribution model. There are two separate managers for organizing and editing your patterns: one for hydraulic demand, the other for constituent source loading (used when performing a water quality analysis to analyze a constituent concentration). The **Pattern Manager** allows you to do the following:

- **Add –** Click the **Add** button. This will open the **Pattern** dialog where the specifics of the pattern can be entered.
- **Edit –** Select the label of the pattern you wish to edit, and click the **Edit** button. The **Fixed** pattern cannot be edited.
- **Duplicate –** Select the label of the pattern you wish to duplicate, and press the **Duplicate** button. The **Fixed** pattern cannot be duplicated.
- **Delete –** Select the label of the pattern you wish to delete, and click the **Delete** button. The **Fixed** pattern cannot be deleted.

In this program, an individual junction can support multiple hydraulic demands. Furthermore, each demand can be assigned any hydraulic demand pattern. This powerful functionality makes it possible to model any type of extended period simulation.

To access the **Pattern Manager** select the **Analysis\Patterns\Hydraulic** or **Analysis\Patterns \Constituent** menu item, or click the **ellipsis (…)** button next to any pattern choice list.

9.9.2 Pattern Editor

A pattern is a series of time step values, each having an associated multiplier value. During an extended period analysis run, multipliers are selected for use by comparison of their time step value with the current time steps of the Extended Period Analysis run. The selected multiplier is applied to any baseline load that is associated with the pattern.

Defining Patterns

- **Label** – A required name to uniquely identify the pattern. This name appears in the choice list when applying patterns to hydraulic demands or constituent source loads.
- **Start Time –** A value between 0 and 24 that specifies the first time-step point in the pattern. All other pattern time step points are referenced from this start time. This program automatically adjusts your pattern when you start an Extended Period Analysis at a time other than zero.
- **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).

Time Step Points

- **Time From Start** – The amount of time from the **Start Time** of the pattern to the time step point being defined.
- **Multiplier** – The multiplier value associated with the time step point.

Format

- **Stepwise Format** – 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 Format** – 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.

Patterns must begin and end with the same multiplier value. This is because patterns will be repeated if the duration of the Extended Period 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.

An Extended Period Analysis is actually a series of steady-state analyses where the boundary conditions for 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 Junction can support multiple hydraulic demands. Furthermore, each demand can be assigned any hydraulic demand pattern. This powerful functionality makes it easy to combine two or more types of demand patterns (such as residential and institutional) at a single junction.

Use the Report button to view or print a graph or detailed report of your pattern.

To access the **Pattern** editor, click the **Add** or **Edit** button on the **Pattern Manager** dialog.

Chapter 10

Cost Estimating

This feature allows you to calculate a planning level estimate of the capital cost associated with an entire system or any portion of a system. This makes it easy to compare the costs associated with the various scenarios, thus helping to ensure that the most cost-effective design is chosen.

The costs associated with a particular element are broken down into two categories, fixed costs and unit costs. The total cost for each element is calculated as the unit cost times the number of units plus the total of the itemized fixed costs associated with the element. For pipes the unit costss are specified in terms of a cost per unit of length, and for nodal elements, the unit cost is specified in terms of a cost per element. The total cost for the elements in the system is then simply the sum of the total cost for each element selected to be included in the cost analysis.

The unit costs for an element are specified in terms of a unit cost function. A unit cost function describes the relationship between an attribute of the element and the unit cost. For example, you may wish to specify the unit cost of a pipe as a function of the diameter, where the unit cost of the pipe increases as the diameter increases.

In addition to a unit cost, every element can have an itemized list of fixed costs. Fixed costs are specified by the user and provide flexibility in specifying the cost of elements for which the unit cost might not accurately capture the actual cost of installing that element. For instance, if an easement is needed for a particular pipe this cost will probably not be reflected in the unit cost for that pipe.

A unit cost can not be associated with some elements, such as pumps and tanks, whose cost cannot be easily correlated to an attribute of the element. However, you can include the costs associated with these elements in the cost analysis by assigning fixed costs to these elements. This provides the user with the flexibility to distinguish between the different costs associated with these types of elements. For instance, if you are costing a pump you may want to differentiate between the cost of the pump itself and the cost of the pump housing.

In addition to specifying the costs for each element in the system, you can also make adjustments on a system wide basis to the total cost of all the elements included in the cost analysis. This makes it easy to account automatically for contingencies and adjustments that need to be made to the total cost of the physical system.

You do not need to have a hydraulically valid network to perform a cost analysis. You can quickly calculate the cost associated with a system at any time through the Cost Manager.

10.1 Cost Manager

The Cost Manager allows you to quickly compute and compare the costs associated with your different scenarios. This dialog provides you with a convenient place to view, edit, and calculate project level cost data. This dialog can be divided into three sections that are described below.

The first section is the series of four buttons on the left side of the window:

- **Unit Cost Functions –** This button opens the **Unit Cost Function** manager which is the place to add new functions describing the relationship between a model attribute and the unit cost for the element. For pipes, this might be a table of data relating the pipe material to the cost per unit length.
- **Cost Alternatives –** This button opens the **Capital Cost Alternatives** manager where you can quickly create different cost alternatives. For example, you may wish to compare the cost associated with different cost functions or different portions of the system.
- **Compute –** This button opens the **Cost Scenario Editor** for the scenario that is selected in the pane to the right of this column of buttons. The **Cost Scenario Editor** is the location to enter the cost adjustments and to run a cost calculation for the selected scenario.
- **Batch Run –** Performing a batch run allows you to set up and run multiple cost scenarios at once. This is helpful if you want to queue a large number of calculations or simply manage a group of smaller calculations as a set.

The second section is the pane to the right of this column of buttons. When you open the **Cost Manager** this pane will contain all the scenarios in the project. Each scenario will have an icon located to the left of it indicating whether a cost calculation has been performed for that scenario. If the icon is a green file folder, then the scenario has not been calculated. If the icon is a file folder with a dollar symbol then the cost results are available for that scenario. If cost results are available for the scenario, then the total cost of the scenario will be displayed to the right of the scenario label.

Clicking on a scenario, for which cost results are available, will expand the components comprising the total cost of the scenario. These components include the following types of sections:

- A section for each cost function that is used in the cost analysis
- A section detailing additional unit costs specified for the pipes in the system
- A section detailing the fixed cost specified for each element in the system
- A section detailing any cost adjustments made to the total cost of all the elements included in the cost analysis

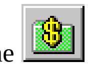
If there is no cost data associated with one or more of these sections then those sections will not be listed.

Just above the right side of the center pane listing the available scenarios is a row of three buttons, each of which is described below.

- **Properties –** This button opens the **Cost Scenario Editor** for the currently selected scenario. In this dialog the system cost adjustments applied to the total cost computed for the elements in the system can be edited. The cost calculation for the selected scenario can also be run from this dialog.
- **Graph –** This button opens a pie chart of the items comprising the total cost of the scenario.
- **Report –** This buttons opens a printed tabular report on any item selected in the pane below these buttons. If the scenario label is selected, then a tabular report of all the items comprising

the scenario are included in the report. If one of the components listed under the scenario is selected then only the information for this one item will be included in the report.

The third section on this dialog is the pane on the right side. This pane is used to display an expanded view of the contents of the item selected in the center pane. The table listed in this pane will appear in the printed report that is opened when the report button described above is clicked.

To open the **Cost Manager** window, select **Analysis\Compute Costs** or click the  button in the analysis toolbar.

10.1.1 Cost Alternatives Manager

The **Cost Alternatives** manager dialog allows you to edit, create, and manage your cost alternatives. It also gives you more advanced capabilities, such as merging alternatives and creating child alternatives.

On the right side of the dialog are a number of buttons that provide functions for managing the alternatives. These buttons are identical to the buttons found in the **Alternatives Manager** dialog. See the **Alternatives** manager topic in the Scenarios/Alternatives chapter for a descriptions of these button's function.

To access the **Cost Alternatives** manager dialog select **Analysis\Alternatives** from the pull-down menu and select the **Cost** tab, or click on the **Cost Alternatives** button in the **Cost Manager**.

10.1.2 Unit Cost Functions

A unit cost function is a description of the relationship between an element attribute and the unit cost for that element. For example it might describe the relationship between pipe diameter and the cost per unit length or it could relate the depth of a gravity structure to the unit cost for that structure. You can specify the relationship between the unit cost and the value of the attribute as either tabular data or as a formula.

Tabular Unit Cost Function – Relates attribute values to unit costs as a series of data points. This is the only way to enter unit cost data for non-numeric attributes such as material. If the attribute for which you are supplying the cost data is numeric then values between the data points that you enter will be linearly interpolated. If the unit cost is requested for an attribute value that falls outside of the range of data that you supplied in the table, the model will assume that the unit cost is equal to the unit cost at the most extreme point closest to the value that was requested. For example if the following points had been entered (8 in, 30$/ft) and (12 in, 40 $/ft) and the unit cost was requested for a 16 in diameter pipe the value returned would be 40 $/ft. When a cost calculation is performed you will be warned about all elements whose unit cost falls outside of the range of data that you supplied.

Formula Unit Cost Function – Represents the unit cost as a function of the selected numeric attribute of the following form:

$$\text{Cost} = d + a(x - c)^b$$

Where:
Cost = Linear cost of the pipe (local currency/m, local currency/ft)
x = Selected attribute (unit depends on the type of attribute)
a,b,c,d = Parameters you specify (units depend on local currency and the type of attribute)

For certain values, such as when x is less than c and b is not an integer, this equation will be invalid. Under these conditions the unit cost returned by the function will be zero.

Unit Cost Functions Manager

You can add, delete, and edit the Unit Cost Functions for your project through this manager. You will be able to assign the cost functions defined here to one or more of the elements of the appropriate type in your system. For example, if you define a cost function here for pipes, you will be able to select this cost function from the drop down box on the cost tab of the pipe element editor dialog.

Use the Save command (accessed from the File menu button on the bottom of this dialog) to save the unit cost functions listed in the Unit Cost Functions Manager. You can then import them into another project using the Import command (accessed from the File menu button in this dialog).

To access the **Unit Cost Functions Manager**, select **Analysis\Compute Costs** from the pull-down menus to display the **Cost Manager** and click the **Unit Cost Functions** button.

New Unit Cost Function Dialog

When you add a new unit cost function you will be prompted with this dialog containing two fields, **Unit Cost Function Type** and **Unit Cost Function Attribute**. This is the information that is needed to initialize the new unit cost function that you are about to create.

The **Unit Cost Function Type** field allows you to select whether you would like to enter your cost function data in a tabular or formula format. We recommend that you quickly familiarize yourself with both formats to see which is most convenient for you. If you wish to base your unit cost on an attribute that is not numeric, such as material, you must choose a tabular format.

The **Unit Cost Function Attribute** field is for selecting the attribute of which your unit costs are a function. For example the unit cost of a pipe might be based upon its diameter or its material. Remember that attributes that are not numeric can only be selected if the unit cost function type is tabular.

Tabular Unit Cost Function

For Unit Cost Functions defined with tabular data, this tab contains the data for this unit cost function grouped as follows:

- **General –** General information identifying the unit cost function.
- **Attribute Value Range –** Display the range of the selected attribute in the current scenario. This information can be useful to make sure you specify cost data for the entire range of values in your model.
- **Unit Cost Data** – Specify the tabular data relating unit cost to the value of the selected attribute.

In order to help you enter and visualize the function, use one of the buttons at the bottom of the dialog:

- **Plot –** Plot the tabular data relating cost to the value of the selected attribute.
- **Initialize Range** – Initialize the minimum and maximum values in the **Attribute Value Range** section, based on all the elements present in your project, for the current scenario.

If the attribute you have selected to define the cost unit function for is outside the defined range for some elements in your network, the unit cost used will be the cost of the minimum or maximum value of the attribute you defined in the table.

Unit Cost Table General Data

This section contains general information identifying the unit cost function, as follows:

- **Label –** Unique identifier for your unit cost function.
- **Element Type –** Displays the type of element to which the function applies, which is always Pressure Pipe in WaterCAD, but could also be Gravity Pipe, Junction, Inlet, Manhole, or Junction Chamber in SewerCAD or StormCAD.
- **Attribute Label –** Element attribute that controls the unit cost, such as pipe diameter. This attribute is selected when you add a new function in the **Unit Cost Function Manager**.

Attribute Value Range

This section displays the Minimum and Maximum values of the attribute that controls the unit cost in your current network. Click the **Initialize Range** button to have these values calculated.

Unit Cost Data Table

Define the unit cost function in a tabular format here, preferably defining the costs associated with the entire range of values present in your network. To display the current range of values in your model, initialize the **Attribute Value Range** section by clicking the **Initialize Range** button.

Formula Unit Cost Function

The data defining formula based unit cost functions is grouped as follows:

- **General –** General information identifying the unit cost function.
- **Valid Cost Data Range –** Specify the range for which the function is valid for the attribute used to define the unit cost function.
- **Coefficients** – Coefficients defining the formula relating the unit cost to the attribute value.

In order to help you enter and visualize the function, use one of the buttons at the bottom of the dialog:

- **Plot –** Display a graph of the unit cost function.
- **Initialize Range –** Initialize the minimum and maximum values of the attribute used to define your unit cost function based on all the elements present in your project.

If the function is invalid for any interval within the Valid Cost Data Range, it is set to 0.0 in that interval. Click the Plot button to visualize any problem with the function.

If the attribute you have selected to define the Unit Cost Function is outside the Valid Cost Data Range for any element in the network, the formula will still be applied to calculate that element unit cost, but an error message for that element will be reported when computing the cost for the system.

Formula Unit Cost General Data

This section contains general information identifying the unit cost function, as follows:

- **Label** – Unique identifier for your unit cost function.
- **Element Type** – Displays the type of element to which the function applies, which is always Pressure pipe in WaterCAD, but could also be Gravity Pipe, Junction, Inlet, Manhole, or Junction Chamber in SewerCAD or StormCAD.
- **Attribute Label –** Element attribute that controls the unit cost, such as pipe diameter. This attribute is selected when you add a new function in the **Unit Cost Function Manager**.
- **Local Unit** – Unit of the attribute that controls the unit cost. This unit is used for defining the formula coefficients.

Valid Cost Data Range

This section specifies the range of values for which the function is valid for the attribute used to define the unit cost function. Clicking the **Initialize Range** button will initialize these two values based on the range of values present in your current network.

Coefficients

In this section you can enter the coefficients defining the unit cost function. The x parameter, which represents the value of the attribute on which the unit cost function is based, is expressed through the unit specified in the **Local Unit** field in this tab.

Unit Cost Function Notes

Optional notes that you can enter related to the unit cost function.

10.1.3 Cost Scenario Editor

The **Cost Scenario Editor** dialog is the control center for each cost analysis. It is the place where you access or change all the information for performing a single calculation (alternatives, system cost adjustments, results, and notes). It is organized into the following tabs:

- **Alternatives –** Edit or view all cost alternatives.
- **Cost Calculation –** Edit or view the cost adjustments for the total system cost and run a cost calculation for a single scenario.
- **Results** – If a cost calculation has been performed for a scenario this log will contain summary information for the calculation as well as any warnings or errors that were generated during the calculation.
- **Notes** – Edit or view the cost analysis notes for this scenario.

To open the **Cost Scenario Editor** dialog, open the Cost Manager by clicking the  button. Then highlight the scenario that you wish to edit, and click the **Compute** button.

Cost Scenario Editor – Cost Calculation Tab

This tab contains scenario level data for the cost calculation and is the location from which you can run a cost calculation for a single scenario.

The scenario level data that are contained on this tab are the cost adjustments that will be made to the total cost computed for all the elements included in the cost analysis. Each cost adjustment includes a label, operation, and factor. The label is the name such as 'Contingency Fee' or 'Adjustment' used to identify the cost adjustment. The operation is the operation (Add, Multiply, or Subtract) that will be used against the factor to compute the total cost associated with the adjustment. The factor is a number that will be applied to the operation to compute the cost associated with the adjustment. For example if you wanted to add a contingency fee of ten percent to the total cost computed for the elements in the system, you would specify an operation of ’Multiply’ and a factor of ’0.10.’ So if the total cost for the elements was $2,000,000, the contingency fee would be $200,000 and the total cost reported for the scenario would be $2,200,000.

You can include as many cost adjustments as you wish in the cost for a particular scenario. The cost adjustments are applied in the order that they appear in the table on this dialog. For instance if you included a contingency of ten percent and a flat adjustment of $100,000, the amount of the contingency would depend on whether it came before or after the flat adjustment in the cost adjustment table. If the contingency fee was listed first in the table and the total cost of the system was $2,000,000, the amount computed for the contingency would be $200,000 giving you a total cost of $2,300,000 after the flat adjustment is included. If the contingency was listed after the flat adjustment, then the amount of the contingency would be ten percent of $2,100,000 or $210,000 and the total cost of the system would then be $2,310,000.

The easiest way to understand how cost adjustments are computed is to perform a few simple cost calculations and view the resulting values. The amount of each cost adjustment is itemized in the cost analysis results for each scenario.

There is also a **GO** button on this tab that you can use to run the cost calculation for the scenario you are editing.

Cost Scenario Editor – Results Tab

The **Results** tab contains a summary of the last cost calculation performed for this scenario. Click the **Save** button to save the results to an ASCII text file. Click the **Print Preview** button to preview the printed version of the contents of the pane in the **Results** tab.

> **Immediately after you run the calculations, the Results tab is automatically displayed, giving you a quick indication of whether the calculation was successful.**

10.1.4 Cost Scenario Batch Run

Performing a batch run allows you to set up and run cost calculations for multiple scenarios at once. This is helpful if you want to queue a large number of calculations, or simply manage a group of smaller calculations as a set. The list of selected scenarios for the batch run will remain with your project until you change it.

Using the dialog is simple. First, check the scenarios you want to run and click the **Batch** button. Each scenario will be calculated. You can cancel the batch run between any scenario calculation.

When the batch is completed, the scenario that was current will remain current, even if it was not one of the scenarios that was calculated. Select a calculated scenario from the center pane of the **Cost Manager** dialog to view the results for that scenario. You can also open the **Cost Scenario Editor** and select the **Results** tab to view a summary of the results and any errors that were generated during the calculation.

Notes

Chapter 11

Presenting Your Results

This chapter covers the various methods that are provided for viewing, annotating, graphing, and reporting your data. It also presents the tools available for generating contours, profiles, and color coding elements based on any attribute.

11.1 Element Annotation

Element annotations allow you to display detailed information such as pipe lengths or node ground elevations in your drawing. You can add one or more annotations for any type of element in the system. Annotations update automatically. For example, annotations will display newly calculated values and will be refreshed as you change scenarios.

The annotations and their format are defined by using the Annotation Wizard. In Stand-Alone mode, the annotation format can also be easily modified in the Attribute Annotation dialog that displays when you double-click the annotation text in the main view window.

Pipe annotations can be aligned with the pipes or displayed horizontally, depending on the Pipe Text setting specified in the Drawing Options dialog.

You can control the angle at which the text flips from one side of the pipe to the other (reading in the opposite direction) to maintain readability when the pipe direction on a plot is nearly vertical. By default, the text flips direction when the pipe direction is 1.5 degrees measured counter-clockwise from the vertical. You can modify this value by inserting a **TextFlipAngle** variable in the Haestad.ini file located in your Haestad directory, under the [WTRC] section. That angle is measured in degrees, counter-clockwise from the vertical.

For instance, if you want the text to flip when the pipe direction is vertical, you should add the following line to the Haestad.ini file:

TextFlipAngle=0

Reasonable values fall in the range from 15.0 degrees to -15.0 degrees.

11.1.1 Attribute Annotation Dialog

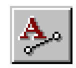
In the main view in Stand-Alone mode, double-click the annotation text to display the corresponding **Attribute Annotation** dialog. Here you can easily modify the format of that attribute annotation without going through the **Annotation Wizard** again. The parameters "%v" and "%u" represent the attribute's value and unit respectively.

11.1.2 The Annotation Wizard

You can use the **Annotation Wizard** to add annotations to the drawing, as well as to remove or modify existing annotations in the drawing.

The wizard is divided into three steps:

- **Select Elements** – Select the types of elements to annotate.
- **Choosing Attributes** – Select the attributes to annotate.
- **Summary** – Summary of the annotation settings you have selected.

To access the Annotation Wizard, click the annotation tool  in the toolbar, or select **Tools\Element Annotation** from the main menu.

Annotation Wizard – Select Elements

This step allows you to specify the types of elements you wish to annotate. Check the appropriate box for each type of element you wish to annotate. You may annotate more than one type of element at a time by checking all the desired element types. If you have already annotated your drawing, you can remove annotations for a particular type of element by unchecking the corresponding box.

Annotation Wizard – Choosing Attributes

The next step(s) allows you to specify the attributes you wish to add for each element type in the selection set you are annotating.

Specify the set of elements… – Choose **All Elements** from the scroll-down list for color coding to be applied to all elements (nodes or links) in the network, or choose a selection set from the scroll-down list. Click on the **ellipses (…)** button to access the **Selection Set Manager** to edit or add selection sets.

For each element type, you will be presented with a table where you can specify the attributes you wish to annotate, and the "mask" for each attribute.

- **Attributes –** This column contains a list of all available attributes for the current element type. Click in this field, and choose the attributes you wish to annotate by selecting from the list that appears.
- **Mask** – This column allows you to customize the way the annotation is displayed. The parameters "%v" and "%u" represent by the attribute's value and unit respectively. By default, the mask is setup as follows: <attribute name>: %v %u. The mask can be customized by editing this field.

> **When annotating pipe diameters, the default mask is "Diameter: %v %u". The default annotation for a 150 millimeter pipe would be "Diameter: 150 mm". By changing the mask to "%v%u", the resulting annotation is "150 mm". You can choose to omit the %u parameter altogether, and instead substitute a common symbol like "inches," or "feet."**

- **Preview** – This column displays a preview of how the annotation will look when it is added to the drawing.

Annotation Wizard – Summary

The last step of annotating your drawing allows you to review the choices you have made. If you would like to make changes at this time, simply click the **Back** button to step back through the wizard. When you are satisfied, click the **Finished** button to add the annotations to the drawing.

> **You can turn annotation visibility on or off by editing the Drawing Options. Your annotation settings will be retained.**
>
> **If the Drawing Options indicate that element annotations should not be displayed, clicking the "Finished" button will automatically turn annotations on.**
>
> **You can double-click an annotation element in the drawing to edit the associated mask.**
>
> **The Annotation Text Height can be adjusted from the Drawing Tab of the Options Dialog, accessed by selecting Tools\Options from the pull-down menus.**

11.2 Color Coding

Color coding allows you to assign colors to elements in the drawing based on a variety of input and output attributes. For any attribute, you can supply a color scheme or have the application generate one for you. For example, you can supply a color scheme to display all pipes sizes between 2" and 8" in green, those between 10" and 24" in blue, and those between 27" and 48" in red.

To access Color Coding, select **Tools\Color Coding...** from the main menu, or click the **Color Coding** (rainbow) button on the main toolbar, or double-click a color coding legend figure in the drawing.

11.2.1 Color Coding Dialog

At the top of the **Color Coding** dialog are two tabs, **Link** and **Node**. You can set up color coding for both links and nodes, or just one of the two.

Attribute – Select the attribute by which you would like to color code, or select **<None>** to turn color coding off.

Selection Set – Choose **All Elements** from the scroll-down list for color coding to be applied to all elements (nodes or links) in the network, or choose a selection set from the scroll-down list. Click on the **ellipses (…)** button to access the **Selection Set Manager** to edit or add selection sets.

Minimum/Maximum – Calculated minimum and maximum for the specified attribute for elements in the specified selection set.

Calculate Range – Automatically determine the minimum and maximum for the specified attribute and selection set.

Initialize – Automatically calculate a set of default color coding ranges for the specified attribute and selection set, based on the values in your project.

Use the **Initialize** and/or the **Insert** buttons to define your color coding map. Then click **OK** and the specified colors will be assigned to the elements in the drawing that belong to the specified selection set(s).

Color coding legends can be added to any location in the drawing by clicking the Legend Button on the Tool Palette.

Color coding will automatically update as input or results change. For example, after performing a calculation, colors will update based on the calculated values.

If the results for the selected attribute are not available, or if all values for that attribute are the same, automatic range initialization will not be performed. You can enter your own custom range in this case.

A color map can have any number of color assignments.

The Quick View window can be used to display a summary of the active link or node color coding parameters.

11.3 Reports

11.3.1 Predefined Reports

This application provides several predefined reports that can be used in your projects. This feature makes report generation a simple point-and-click exercise. Simply select the elements for which you want a report and send them to your printer.

The following types of Predefined Reports are available:

- Element Details Reports
- Element Results Reports
- Tabular Reports
- Scenario Summary Reports
- Project Inventory Reports
- Calculation Results Table
- Plan View Reports

Detailed reports can be copied to the Windows clipboard in RTF format for use in your favorite word processing program. Refer to the Print Preview window for more information.

To access the Predefined Reports, select **Report** from the menu bar and select the report of your choice.

11.3.2 Element Details Report

The **Detailed Reports** dialog allows you to print detailed reports for all elements or any subset of elements in the system.

From the **Detailed Reports** dialog, select elements to be printed by holding down the **Shift** key or the **Control** key while clicking with the mouse. Holding down the **Shift** key will provide group selection behavior. Holding down the **Control** key will provide single element selection behavior. Alternately, use the **Select** button to open the **Selection Set** dialog. This provides more powerful selection functions. When you are satisfied, click the **Print** button to output the selected reports.

You can graphically select elements that you would like to print before opening the Detailed Reports dialog. The selected elements will be highlighted in the list of elements to print. This is done by holding down the Shift key and selecting elements or by dragging a window around the area of interest.

You can print a detailed report for a single element without using the Detailed Reports dialog. Simply open the element editor for the desired element and click the Report button.

To access the **Detailed Reports** dialog, select **Report\Element Details** from the menu bar.

11.3.3 Element Results Report

The **Element Results** dialog allows you to print or preview a single report containing the results for any number of elements in the system.

From the **Element Results** dialog, you can select elements to be printed by holding down the **Shift** key or the **Control** key when clicking with the mouse. Holding down the **Shift** key will provide group selection behavior and holding down the **Control** key will provide single element selection behavior. Alternately, use the **Select** button to open the **Selection Set** dialog. This provides more powerful selection functions. When you are satisfied, click the **Preview** button to view the selected reports, or click the **Print** button to print the selected reports.

You can graphically select elements that you would like to print before opening the Element Result Reports dialog. The selected elements will be highlighted in the list of elements to print. This is done by holding down the Shift key and selecting elements or by dragging a window around the area of interest.

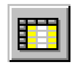
When working with large systems, the preview option can require a great deal of system resources. You can reduce resource requirements by selecting a small subset of elements with which to work, or by printing a subset of the time step results, as described above. The print option has lower system resource requirements than the preview option.

To access the **Element Results** dialog, select **Report\Element Results** from the menu.

11.3.4 Tabular Report

Using the powerful FlexTables feature you can very quickly generate a tabular report containing any attribute (in columns) and any network element (in rows).

All tabular data in this program can be copied to the Windows Clipboard by right-clicking the desired table and selecting Copy in the context menu. You can then paste this data into your favorite spreadsheet or word processor to generate custom reports and graphs.

To access the Table Manager, click the **Table** button  on the main toolbar, or choose **Report\Tables** from the main menu.

11.3.5 Scenario Summary Report

The **Scenario Summary** provides a detailed report of the active scenario, including alternatives and a brief summary of the calculation options.

To access the **Scenario Summary Report**, select **Report\Scenario Summary** from the menu bar.

11.3.6 Project Inventory Report

The **Project Inventory** report provides a detailed report that includes a summary of the active scenario, a network inventory, and a detailed pipe inventory (grouped by pipe section).

To access the **Project Inventory Report**, select **Report\Project Inventory** from the menu bar.

11.3.7 Calculation Results Table

The calculation results for each element in a network can be viewed in a table format. This table is predefined and you cannot change it. It displays the set of the most commonly desired output attributes for the type of element for each reporting time step in the hydraulic analysis. The contents of the table can be copied to the Windows clipboard to transfer the data to another application such as a spreadsheet or word processing document.

To copy the data to the Windows clipboard, right-click the table and select **Copy** from the context menu.

You can change the reporting time step increment on the Analysis Toolbar.

To view the **Calculation Results Table** for a particular element, select **Table** from the **Report** button on the editor dialog for the desired element.

11.3.8 Plan View Report

Generate print previews for the plan view of the network for either the current drawing display (Current View) or the entire drawing extents (Full View).

To generate a preview of the current view or the entire network, choose either **Report\Plan View\Current View** or **Report\Plan View\Full View**, respectively, from the main menu.

11.3.9 Calculation/Problem Summary Report

After running hydraulic calculations, the **Results** tab of the **Scenario Editor** is displayed. This tab contains a summary of the calculation results, or a list of problems encountered with the data, in case the calculation could not be completed.

This report can be previewed before being printed (or copied to the clipboard) by clicking the **Printer** button on the **Results** tab. They can also be printed to a text file by clicking the **Printer** button on the **Results** tab.

This **Results** tab can be accessed from the **Scenario Manager**, reporting the results corresponding to the highlighted scenario in the scenario tree view. It can also be accessed for the currently active scenario by clicking on the **GO** button, which opens the **Scenario Editor**.

11.3.10 Contour Plan View

A preview of the Contour Plan View Report, showing all countours as displayed in the **Contour Plot** window, can be obtained by clicking on the **Print Preview** button in the **Contour Plot** window.

11.4 Graphs

11.4.1 Pump Curve

To generate a pump curve, open the **Pump Editor** for the pump of interest, click the **Report** button, and choose the **Pump Curve** menu item.

11.4.2 Tank Storage Curve

To generate a plot of the tank storage volume versus the elevation, open the **Tank Editor** for the tank of interest, click the **Report** button, and choose the **Tank Curve** menu item.

11.4.3 Junction Demand Graph

To generate a graph of the total demand at a junction over time, open the **Junction Editor** for the junction of interest, select the **Demand** tab and click the **Graph** button.

11.4.4 Pattern Graph and Report

You can generate a graph or a full report of a pattern, representing the multiplier variable of the pattern over time. To do so, open the **Hydraulic** or **Constituent Pattern Manager** dialog to access the pattern for which you would like to generate output. From the **Pattern Editor** dialog, click the **Report** button, and select **Graph** or **Detailed Report**.

11.4.5 Plotting a Variable vs Time

Graph Setup

The **Graph Setup** dialog allows you to graph calculated results for any element in the system.

- **Independent Variable** – The Independent Variable "Time" will be plotted on the x-axis.
- **Dependent Variable –** The selected dependant variable will be plotted on the y-axis.
- **Available Scenarios –** Select and compare various scenario computations.

The Graph Setup option is only available for Extended Period Analysis.

When the Plot window is open, click the Options button to graph other dependent variables.

The **Graph Setup** dialog can be accessed for any element in the system. For the element you would like to graph, open its Element Editor by double-clicking the element in the main window, then click the **Report** button and select the **Graph** menu item.

Available Scenarios

This feature allows you to select which scenario(s) you wish to view and compare on the current graph. Place a check mark by the scenario(s) you wish to display.

By default, the base scenario is named "Base".

11.5 Contours

11.5.1 Contour Map Manager

The **Contour Map Manager** contains the information required to generate contours for a *calculated* network, organized as follows:

- **Contour –** Choice list used to select the attribute that is to be contoured.
- **Selection Set** – Contours can be generated using all elements in the network or a subset of elements, defined in the Selection Set Manager. Click the **ellipsis (…)** button to access the Selection Set Manager.

In addition to using selection sets, you can also add nodes to a special zone that will ensure that they are excluded from the contouring point set. Cybernet 2 users might recall that Zone number 99 was reserved for this purpose. You should create a zone named 'Do not contour'. You can then add the nodes that you do not want to be included in the contour set. You can change the name of the contour exclusion zone by editing the file, haestad.ini, and setting the variable, 'ExcludeFromContouringTag' equal to any string label. The exclusion label is not case-sensitive.

If you want to exclude some spot elevations from the contouring point set, set their Description field to 'Do not contour' (or whatever value is set in the haestad.ini 'ExcludeFromContouringTag' variable).

- **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.
- **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.
- **Color by Index –** The standard contours and index contours have separately controlled colors so you can make the index contours more apparent.
- **Color by Range –** Contours are colored based on attribute ranges. Use the **Initialize** button to create five evenly spaced ranges and associated colors.

> **Initialization can be accomplished by clicking the Initialize button. This program will then automatically generate values for the minimum, maximum, and so on, to create an evenly spaced contour set. These values may or may not be the desired range for your purposes, but should at least give you a better concept of the range of values with which you are dealing.**

To access the **Contour Map Manager**, click on the Contour button on the main toolbar or select **Tools\Contouring** from the menus.

11.5.2 Contour Plot

The **Contour Plot** window displays the results of a contour map specification as accurate, straight-line contours.

The plot can be printed or exported as a DXF file using the **File** button at the top of the window. In AutoCAD mode, you can export the contours directly to your AutoCAD drawing by clicking **File\Export to AutoCAD**.

> **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.**

> **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.**

11.5.3 Contour Smoothing

The Contour Smoothing option displays the results of a contour map specification as smooth, curved contours.

The plot can be printed or exported as a DXF file using the command buttons at the top of the window.

11.5.4 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.

11.5.5 Spot Elevations

In addition to the elevations at junction nodes and other network elements, supplemental spot elevations can be entered throughout the model without adding unnecessary model nodes.

These spot elevations have no effect on the network model, but can better define the terrain surface throughout the drawing. The result is that elevation contours and enhanced pressure contours can be generated with more detail. This gives the modeler a better prediction of the system's behavior, even in areas where the model has been skeletonized.

Because spot elevations are not included in the actual piping network, there is very little information in the spot elevation editor. The data consists of the following:

- **Spot Elevation Input Data –** General characteristics defined by the user.
- **Spot Elevation Calculated Results –** Values calculated from the model results.

To access the **Spot Elevation** editor:

Stand-Alone:	Double-click the spot elevation you wish to edit or right-click the element and select **Edit** from the drop-down menu.
AutoCAD R14:	Pick the Select tool and click the spot elevation you wish to edit, or if the Right-Click Context Menu is selected you can right-click the element and select **Edit** from the drop-down menu.
AutoCAD 2000:	Pick the Select tool and click the spot elevation you wish to edit or select the element and select **Edit** from the drop-down menu.

Spot Elevation Input Data

Spot Elevations have user-defined characteristics, including:

- **Label** – Unique "name" by which a spot elevation element will be referenced in reports, error messages, and tables.
- **X-Coordinate** (Easting) – The location may be presented as an X-value, or defined as an Easting value, depending on individual preferences.
- **Y-Coordinate** (Northing) – The location may be presented as a Y-value, or defined as a Northing value, depending on individual preferences.
- **Elevation –** Elevation of the junction.
- **Description –** Optional notes describing the element.

Spot Elevation Calculated Results

Because spot elevations are not directly tied to the hydraulic network model, there are only a few values that are calculated from the model results:

- **Hydraulic Grade –** Interpolated hydraulic grade at this location.
- **Enhanced Pressure –** Pressure based on the interpolated hydraulic grade.

These values are obtained by interpolating between three adjacent model nodes. The hydraulic grade is determined from this interpolation, and the pressure is then computed as a function of the interpolated hydraulic grade and the elevation.

For spot elevations that are outside the model bounds, there may not be three adjacent model nodes. If this is the case, the hydraulic grade will be determined to be zero, which may result in negative pressures. This does not necessarily demonstrate that there are poor conditions in the system. It simply indicates that the spot elevations may cover a wider area than the model itself.

11.6 Profile

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.

Although profiles in general are not limited to a specific alignment, piping network models are usually concerned with a specific profile alignment type called a network walk.

11.6.1 Profile Setup

Setting up a profile is a matter of simply selecting the walk and the attribute on which the profile is to be based. The **Profile Setup** dialog includes:

- **Attribute –** Parameter to be plottted on the vertical axis of the profile.
- **Elements –** List of elements that define the walk to be profiled.

In addition, the network walk can be manipulated by using some of the associated options:

- **Select From Drawing** – Return to the drawing in a protected mode to select and deselect elements for inclusion in the walk.
- **Reverse** – Reverse the order of the walk. The first node in the list becomes the last, and the last node becomes the first.
- **Remove All –** Remove all elements from the current walk.
- **Remove All Previous –** Remove all elements that appear before the selected element in the list. If the selected element is a pipe, the associated node will not be removed.
- **Remove All Following** – Remove all elements that appear after the selected element in the list. If the selected element is a pipe, the associated node will not be removed.

When everything is set up to your satisfaction, click the **Profile** button to generate the graph.

To open the **Profile Setup** dialog:

Click on the **Profile** button on the toolbar of the main window.

- or -

Select **Tools\Profiling** from the main menu.

11.6.2 Profile Plot

The **Profile Plot** window displays the results of an analysis in a profile format. The plot can be copied to the Windows clipboard or printed out directly. By selecting the **Options\Graph Options** menu button, you can also adjust the titles, axes, colors, and other characteristics of the graph.

There is also a time toolbar on the **Profile Plot** window that allows you to follow the profile through extended period simulation results.

For an extended period simulation, the extents of the axes are determined based on the minimum and maximum attribute values for the entire time step, not just the current time step. This is done so that stepping through the time steps gives a more accurate portrayal of the system behavior without rescaling.

11.6.3 Export Profiles (in AutoCAD Mode)

Profiles can be exported to the AutoCAD drawing using the **File** menu on the **Profile Plot** dialog. Profiles will be exported to an insertion point below the current drawing extents.

11.6.4 Walk

A walk is a collection of nodes and pipes that follows a specific path through the network. It can include any type of network element, but cannot include annotations or spot elevations.

A walk is a non-branching path through the network, and can only be extended at either end. Pipes cannot be added along the midsection of the walk. Likewise, elements in the midsection of the walk cannot be deselected without first deselecting all of the elements between one end and the undesired element.

A walk cannot double back on itself, so once a pipe has been selected it cannot be included elsewhere in the walk.

11.6.5 Walk Selection

After clicking the **Select From Drawing** button to define a walk, you will be returned to the drawing editor. If there are already elements in the current walk, they will be displayed in a highlighted mode. Otherwise, you need to begin a new walk by simply clicking any pipe. The pipe and its end nodes will then be highlighted. Continue clicking pipes to add them to the walk, or click highlighted end pipes to remove them from the current walk.

Once you have selected a walk, press the Escape button on your keyboard or right-click with the mouse and select **Done** from the context menu.

11.7 Scenario Comparison

The data calculated in different scenarios can be compared through the use of the **Scenario Comparison** window. This allows you to display the differences in the values calculated for each scenario on the model map as annotations.

11.7.1 Annotation Comparison Wizard

The **Annotation Comparison Wizard** is used to create a drawing that contains text elements displaying the differences between specific attributes of two scenarios. The **Annotation Comparison Wizard** is identical to the **Annotation Wizard** except it has one additional step. This step involves selecting the two scenarios you wish to compare.

- **Scenario 1** – Choose the baseline scenario.

- **Scenario 2** – Choose the scenario you wish to compare to Scenario 1.

The value in Scenario 1 is substracted from the value of Scenario 2, and the difference is displayed. Therefore, if any specified attribute's value is greater in Scenario 2 than it is in Scenario 1, the difference is displayed as a positive number. If the value is smaller in Scenario 2 than in Scenario 1, it is displayed as a negative number.

For example, suppose your model contains two scenarios. One is named 2002 Conditions, and the other is named 2010 Conditions. To create a drawing that displays the difference in velocity in a pipe between the 2002 scenario and the 2010 scenario, you would use the **Annotation Comparison Wizard**. You could choose the 2002 scenario as Scenario 1, and the 2010 scenario as Scenario 2. You would then complete the rest of the steps in the wizard. The drawing produced would show positive values where the velocity increased under 2010 conditions and negative values where the velocity decreased under 2010 conditions.

> **For applications that support extended period simulations, you can choose the same scenario for Scenario 1 and Scenario 2 effectively comparing a scenario to itself to annotate the differences between two time steps of that scenario.**

To access the **Annotation Comparison Wizard**, open the **Scenario Manager** and click the **Scenario Comparison** button.

11.7.2 Scenario Comparison Window

The **Scenario Comparison** window allows you to view, print, export, and modify scenario comparison annotations.

Along the top of the window is a row of buttons that perform the various functions listed below:

- **File\Export To DXF** – Export the drawing in the standard DXF file format.
- **File\Export To AutoCAD (available only in AutoCAD mode)** – Export the drawing to the current AutoCAD drawing.
- **Zoom Tools** – Provide standard zoom capabilities for navigating the drawing.
- **Options\Annotation Manager** – Open the **Annotation Comparison Wizard** to add, delete, or modify the scenario comparison annotations.
- **Options\Annotation Height Multiplier –** Modify the text height for the scenario comparison annotations.
- **Options\Find Element –** Allows you to locate an element by its label.
- **Print Preview** – Open the **Print Preview** window to view how the printed page(s) will look.
- **Close** – Close the **Scenario Comparison** window.
- **Help** – Get quick access to this Help topic.

Several user interface elements are available to let you modify the scenarios that are being compared, and to control when the scenario comparison annotations are updated. These interface elements are described in more detail below.

- **Scenario 1 –** This row of controls is similar to the Analysis Toolbar on the main window. This field allows you to choose, from the list of available scenarios, the one that will be the baseline in the comparison.

- **Scenario 2 –** This row of controls is identical to those described above in Scenario 1, but instead of defining the baseline for the comparison, they define the scenario to be compared to the baseline.
- **Update –** Click this button to refresh the scenario comparison annotations. This button is used when Auto Update (described below) is off, and you have changed either Scenario 1 or Scenario 2.
- **Auto Update –** A check in this box indicates that **Auto Update** is on, and that the scenario comparison annotations will be refreshed whenever Scenario 1 or Scenario 2 is changed. With Auto Update off, you can select the desired combination of Scenario 1 and Scenario 2 time steps, then click the **Update** button. With **Auto Update** on, the annotations will refresh even if you are not interested in the current combination.

The **Scenario Comparison** window is accessed by pressing the **Scenario Comparison** button in the **Scenario Manager** and then completing the **Annotation Comparison Wizard**.

11.8 Graphic Annotation

In Stand-Alone mode, several **Graphic Annotation** tools are provided for enhancing the appearance of your drawing. Graphic annotations can be manipulated like any other element in the Graphical Editor. You can add, move, and delete them just as you would network elements.

To add graphic annotation to your drawing, use the **Tools\Layout\Graphic Annotation** and **Tools\Layout\Legend** menu item, or use the tool-palette located along the left side of the main window. The available tools are:

- **Line Tool –** Add polylines or polygons for things like drawing roads or outlining catchments.
- **Border Tool** – Add rectangles to your drawing for creating borders.
- **Text Tool –** Add text to your drawing for adding things like explanatory notes or a title to the drawing.
- **Legend Tool –** Add a link or node color coding legend to your drawing.

The program will calculate the area of a closed polyline. Right-click the polyline of which you wish to determine the area and select Enclosed Area.

To open or close a polyline, right-click the polyline and select Close. A check will appear next to the menu item to indicate that the polyline is closed.

To add bends or vertices to a polyline, right-click the polyline at the location you would like to add a bend and select Bend\Add Bend.

To remove bends or vertices from a polyline, select the polyline, right-click the bend you would like to remove, and select Bend\Remove Bend.

11.8.1 Legend

Legends are used to display the ranges of the active link and node color coding. The legend tool adds a color coding legend to the drawing. This legend is automatically updated as the color coding is modified.

Editing of the legend figure is not required. In Stand-Alone mode, multiple legends may be placed in the drawing to assist you when printing only regions within the drawing.

> **You can double-click a color coding legend in the drawing to edit the associated color coding parameters.**

11.9 Preview Windows

11.9.1 Plot Window

The **Plot** window, that displays when generating a profile, contour map, etc., contains four option buttons at the top of the **Plot** window:

- **Copy –** Outputs the plot onto the clipboard for use in other applications.
- **Print** – Outputs the contents of the **Plot** window to the printer.
- **Options\Graph Options –** Customizes the Plot. This feature allows you to change the graph's axes, fonts, titles, etc.
- **Close** – Closes the **Plot** window.

11.9.2 Print Preview Window

This window provides you with a preview of what will be printed. The top of the window contains the following buttons:

- **PgUp/PgDn** – Navigate between pages of the report.
- **Copy** – Copy the page(s) to the Windows clipboard.
- **Print –** Output the contents of the **Print Preview** window to the printer.
- **Options\Print Setup** – Change printer options, such as portrait or landscape page layout.
- **Options\Fit to Page –** The **Fit to Page** check box will not appear if the **Preview** window does not contain a drawing or if the drawing is in schematic mode (not to scale). When checked, the drawing will be scaled to fit within a single page. When not checked, the drawing will be output using the drawing scale.
- **Close –** Close the **Print Preview** window.

11.9.3 Graph Options

These features allow you to customize the way a graph or pie chart (used in SewerCAD) looks. The dialog is divided into several tabs:

Titles

- **Titles** – There are three sets of titles for a graph: Graph title, X-Axis title and Y-Axis title. Each title set contains two levels: title and subtitle. A pie chart simply has a title and a subtitle.
- **Title Font** – This feature allows you to select and change the text font type for specific items on the graph or pie chart. Use the selection list to choose the item for which to change the font, then click the **ellipsis (…)** button to select the desired font type from the list of available fonts currently installed on your PC.

Axis (for graphs only)

- **Automatic Scaling –** By default, the program uses the Automatic Scaling options for setting the X and Y-axis minimum, maximum, and increment values. To customize an axis, turn the check mark off and enter the desired values for the minimum, maximum, and increment. If desired, you can customize a single axis while leaving the other in the Automatic Scaling mode.
- **Log Scale –** Place a check mark in this box to use a log scale for this axis. You can use a log scale for one or both axes.

Grid (for graphs only)

- **X-Axis –** Place a check mark in this box to view grid lines corresponding to the X-Axis labels.
- **Y-Axis** – Place a check mark in this box to view grid lines corresponding to the Y-Axis labels.
- **Line Color –** Use this selection list to define the color to use for both axes grid lines.
- **Line Style –** Use this selection list to define the line type (solid, dashed, etc) to use for both axes grid lines.
- **Fill Color** – Use this selection list to define the color to use for background fill within the plotting boundaries of the graph.
- **Save as Default** – Place a check mark in this box to save the current grid settings as the default for subsequent graphs.

You can choose to use grid lines for one or both axes.

Display (for pie charts only)

- **Data Labels** – Allows you to annotate the pie-charts with percentages, labels, or both.
- **Chart View** – Allows you to generate a 3D-view pie chart.
- **Legend Location** – Allows you to place the legend (if any) on the left, right, top, or bottom of the pie chart.
- **Percentages** – Indicates how many decimals are to be displayed for the percentage figures.

Legend

- **Show Legend** – A check mark designates that the legend will be included on the graph or pie chart. Turn the check mark off if you do not wish to show the legend.
- **Series –** Each series represents a different curve on the graph or a slice on the pie chart. If the graph contains only one curve, or the pie chart contains only one slice, then it is designated as Series 1. Scroll through the list and select the desired curve or slice (series number) for which to customize the color and/or line type. Then, use one of the options below to customize it:
  - **Label** – Name for the selected curve (series).
  - **Line Color –** Color for the selected curve (series).
  - **Line Style (for graphs only)** – Style for the selected curve (series).
  - **Line Width (for graphs only)** – Width for the selected curve (series).
  - **Symbol (for graphs only)** – Data point symbol to use for the selected curve (series).

- **Save as Default –** Place a check mark in this box to save the current legend settings as the default for subsequent graphs.

11.10 Status Log

Several commands generate a status log showing the results of that command (for example, a database connection). The main window will show you status information based on your action. The dialog contains the following buttons:

- **Print –** Print the status log results.
- **Save –** Export the status log results as an ASCII file.
- **Copy –** Copy the status log results to the Windows clipboard.
- **Close** – Close the **Status Log** dialog after design calculations.

Notes

Chapter 12

Engineering Libraries

The Haestad Methods' engineering libraries and library managers are powerful and flexible facilities for managing 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, pipe sections (in StormCAD and SewerCAD), and dry weather loads (in SewerCAD only). You can modify engineering libraries and the objects they contain by using the **Tools\Engineering Libraries** menu or by clicking the **ellipsis (…)** buttons available next to the fields in dialog boxes that make use of library objects.

The data for each engineering library is stored in a tabular ASCII file with the extension .HLB.

> **We strongly recommend that you only edit these files using the built-in facilities available from the Tools\Engineering Libraries menu. If absolutely necessary, these library files may be edited or repaired using any ASCII editor.**

The standard set of engineering libraries shipped with your Haestad Methods product reside in the product's program directory. By default, each project you create will use the objects in these 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, and setting the path in the engineering library manager for the project to the path for the custom library.

When you change the properties for an object in an engineering library, those changes will affect all projects that use that library. At the time a project is loaded, all of its engineering library objects are synchronized to the current library. Objects are synchronized based on their label. If the label is the same, then the object's values will be made the same. If any library referenced in a library manager path cannot be located, then the standard library in the program directory will be used. Once a project is created, it is not necessary to have access to the engineering library in order for that project to be edited or analyzed.

12.1 Engineering Library Manager

The **Engineering Library Manager** dialog consists of a table of five columns: **Library**, **Current Path**, **Browse**, **Edit**, and **New**. There is one row for each kind of engineering library used in your project. You cannot create library types different from the set of standard libraries shipped with the product. The columns in the table are as follows:

- **Library** – This column lists the kind of object stored in the referenced library.

- **Current Path** – This column lists the path to the library to be used for objects of a certain kind within the current project. By default, the path will reference the standard library shipped with your Haestad Methods product. To browse for other libraries of the same type that you may have already created, click the **Browse** column.
- **Browse** – Click this column and the button that appears if you wish to search your computer or network and locate other engineering libraries. To reference a library in the path field, the library must already exist. To create it you may copy a standard library using Windows File Manager or Explorer, or click **New** as described below.
- **Edit** – Click this column and the button that appears if you wish to add, delete, or edit the objects within a specific kind of engineering library.
- **New** – Click this column and the button that appears if you wish to create a new library.

Most users do not need to create custom libraries or edit the library paths. You only need to change path values if you wish to create and use custom libraries.

The **Engineering Library Manager** can be accessed by selecting **Tools\Engineering Libraries** from the pull-down menu.

12.1.1 Engineering Library Editor

The **Engineering Library** dialogs consist of a table of two columns:

- **Label** – This column contains a textual description of the object. In general, objects are considered to be the same if their labels are the same. For example, when a project is loaded, the engineering library objects are synchronized to the current library based on label.
- **Available in WaterCAD** – This column contains a checkbox indicating whether the library object on the given row is enabled for use. If an object is enabled, it will appear in choice lists as a candidate for use in the project. If an object is disabled, it will remain in the library and be editable, but it will not be offered as a candidate for any operations in the program. If a disabled object has already been used in a project, then it will remain in use. Disabling it will not affect the existing project in any way.

The following command buttons appear on the **Engineering Library** dialog:

- **Insert** – Insert a new, unlabeled object into the current library. You must then click the **Edit** button to edit the label and add the appropriate values before the library will be valid. Library objects will be sorted by label in ascending alphabetical order the next time you open the **Engineering Library** dialog.
- **Duplicate** – Create a copy of the currently highlighted object at the bottom of the list.
- **Delete** – Delete the object represented by the highlighted row. Note that this command always deletes objects from the library, but never deletes an object from your current project if it is in use. To change the library object that is currently in use by a project, proceed to the dialog containing the field where the library object is referenced and select a different library object.
- **Edit** – Access the object properties editor.
- **Usage** – Only applies to the material engineering library. Use this button to specify specific uses for the material.

The **Engineering Library** dialogs can be accessed by selecting **Tools\Engineering Libraries** from the pull-down menus and clicking the **Edit** column and the button that appears next to the Library you want to edit.

12.1.2 Material Properties

A user-editable library of materials is provided. Pipes (and ditches/channels in StormCAD) are constructed from various materials. It is often useful to specify the material of the pipes and channels/ditches in your hydraulic and hydrologic models. Materials provide the pipe or channel with a default value for the roughness coefficient used in the friction equations. Therefore, a material must be defined with the following properties:

- **Label** – Name of the material as it will appear in material choice lists.
- **Culvert Inlet Material Type** – Limits the type of culvert inlets that are available in the material (used in CulvertMaster). The inclusion of this property allows the sharing of libraries among Haestad Methods' products.
- **Manning's Coefficient** – Default value for Manning's n. This is a unitless number generally between 0.009 and 0.300.
- **Roughness Height** – Default value for absolute roughness height. This will be used in conjunction with the Darcy-Weisbach friction equation. The roughness height has units of length, typically mm or ft.
- **Kutter's n Coefficient** – Default value for Kutter's n. This is a unitless number generally between 0.009 and 0.300.
- **C Coefficient** – Material's default value for Hazen William's C. This is a unitless number generally between 60 and 150.

The check boxes next to each item specify whether or not the friction method will be available for the material. For example, some materials, such as asphalt, may only have Manning's n defined.

Usage

This dialog only applies to a Material Engineering Library. It specifies the uses for which the material will be available. This dialog is used frequently in StormCAD, but typically is not needed for WaterCAD.

[>] Adds the selected item(s) from the **Available Items** list to the **Selected Items** list.

[>>] Adds all of the items in the **Available Items** list to the **Selected Items** list.

[<] Removes the selected item(s) from the **Selected Items** list.

[<<] Removes all items from the **Selected Items** list.

12.1.3 Minor Loss Properties

An editable library of minor losses is provided. Minor losses are used on pressure pipes and valves to model headlosses due to pipe fittings or obstructions to the flow. A minor loss is defined with the following properties:

- **Label** – Name of the minor loss as it will appear in choice lists.
- **Type** – General type of fitting or loss element. This field is used to limit the number of minor loss elements available in choice lists. For example, the minor loss choice list on the valve dialog only includes minor losses of type valve. You cannot add or delete types.
- **K Coefficient** – Headloss coefficient for the minor loss. This unitless number represents the ratio of the headloss across the minor loss element to the velocity head of the flow through the element.

12.1.4 Liquid Properties

An editable library of liquids is provided. All hydraulic or hydrologic networks transport a particular liquid. Liquids are defined with the following properties:

- **Liquid Label** – Name of the liquid as it will appear in choice lists.
- **Kinematic Viscosity –** Ratio of the liquid's dynamic, or absolute, viscosity to its mass density. This is a common parameter in fluid mechanics. The units of kinematic viscosity are length squared per unit time (typically m^2/sec or ft^2/sec).
- **Specific Gravity –** Ratio of the specific weight of the liquid to the specific weight of water at 4°C(39°F). Specific gravity is a unitless number.
- **Temperature –** Reference temperature for the liquid. This is required because the two parameters listed above are generally a function of the temperature. The default temperature for new liquids is room temperature (20°C / 68°F).

Certain friction methods (i.e. Manning's, Hazen Williams) were developed experimentally, and are only applicable to water at room temperature (20°C / 68°F). This program will ask you to confirm a liquid choice that is inconsistent with the chosen friction method, but it will not prevent you from using it.

Specify the liquid to be modeled in the Project Options dialog.

12.1.5 Constituent Properties

An editable library of constituents is provided. Constituents are used in water quality analyses where the tracking of the growth or decay of a constituent is desired. Constituents are defined with the following properties:

- **Label** – Name of the constituent as it will appear in choice lists.
- **Bulk Reaction –** Reaction rate constant used to model reactions of the constituent within the bulk flow. This value is used as the default bulk reaction rate constant for all pipes and tanks. This constant has units of per unit time (typically /day).
- **Wall Reaction –** Reaction rate constant used to model reactions that occur with the material along the pipe wall. This value is used as the default wall reaction rate constant for all pipes. This constant has units of length per unit time (typically m/day or ft/day).
- **Diffusivity** – Molecular diffusivity of the constituent. This value is only used when pipe wall reactions are considered in the water quality analysis. Diffusivity has units of length squared per unit time (typically m^2/s or ft^2/s).
- **Unlimited Concentration** – Check this box if the constituent does **not** have a limiting concentration or potential. If this box is checked, the **Concentration Limit** field will not be available for editing. If it is **not** checked, the **Concentration Limit** field is applicable. This box will typically be checked, but certain constituents, such as trihalomethanes (THM's), have a limiting concentration or formation potential that needs to be modeled.
- **Concentration Limit** – Limiting concentration or potential for the constituent. When a concentration limit is given, reaction rates will be proportional to the difference of the current concentration and the concentration limit. Concentration limit has units of mass per unit volume (typically mg/l or lbs/million gal.)

The constituent library shipped by Haestad Methods includes a single constituent labeled "Constituent". The values for this constituent do not correspond to anything in particular. It is not intended that you use this constituent for an actual analysis, but it was provided simply as a starting point for the library.

A negative value for the reaction rate constants indicates constituent decay, and a positive value indicates constituent growth.

Notes

Chapter 13

GIS and Database Connections

Haestad Methods' GIS Database Connection feature provides the modeler with the ability to dynamically exchange data with a variety of applications. You can establish a "Connection" between your hydraulic model and relational and non-relational database management systems (RDBMS and DBMS), spreadsheets, and ESRI Shapefiles. Throughout the rest of this chapter, the term "external file" will be used to generically refer to any one of these types of files. Where information pertains to a specific type of external file, that type will be used.

The GIS/Database Connection system is extremely powerful. It can be used to update hundreds or thousands of database records with a few clicks of the mouse. This chapter provides a detailed look at the structure and behavior of the system so that you can use it in the most effective manner.

The purpose of the GIS/Database Connection system is to provide you with a safe and convenient means of exchanging data with external files. This system has several advantages over simply providing an open file format for direct manipulation by the end user.

Generality – Open file formats have a specific form that must be adhered to. This restrictiveness is problematic for both the developer and the end user. Developers are now under additional constraints when modifying the software. They must be cognizant of the fact that users may depend on this format, and are therefore less free to modify it. The end user, on the other hand, has no control over this format, and is therefore at the mercy of the developer. A new version may change the format completely, and all of your existing data must be converted. In addition, the file format is rarely convenient for an end user, since it is typically chosen for efficient processing by the program. The GIS/Database Connection system allows you to exchange data between the model and any arbitrarily defined external files. This flexibility allows you to set up a database or spreadsheet, and it frees the developer to use a file format that is efficient for the program.

Data Protection – Open file formats can typically be modified by anyone, often without the knowledge of the modeler. By providing an interface to exchange data, the model is protected from inadvertent changes. The modeler is in complete control of when and how the model or external files are updated.

Type Coercion – Quite often the external files do not store the data using the format expected by the hydraulic model. For example, a database may store the length of a pipe using single precision floating point numbers, whereas the model works with double precision floating point numbers. When exchanging data between the model and the external file using the GIS/Database Connection system, the data is coerced from one type to the other automatically.

Unit Conversion – The quantities used in hydraulic models almost always have some unit associated with them. For example, pipe lengths are typically expressed in meters or feet. General purpose database and spreadsheet applications do not support the concept of unitized numbers. A pipe length, for example, is simply represented as 100.0. Is that 100.0 meters or

100.0 feet? The GIS/Database Connection interface allows you to specify the database unit, so that the numbers can be converted from the model unit to the database unit and vice versa.

Virtually all model inputs and calculated results can be exchanged through the GIS/Database Connection system. The system not only supports the update of existing model elements and external file records, but also the creation and deletion of these elements and records. For example, by performing a **Sync In** operation (explained in detail below), an entire hydraulic model can be built from data stored in a spreadsheet. Likewise, an empty spreadsheet can be completely populated with data from an existing hydraulic model by performing a **Sync Out** Operation. The spreadsheet can be kept synchronized with the hydraulic model over the course of a project as new elements are added or deleted, and the input and output data is modified.

The GIS/Database Connection system has a three-tiered architecture:

- Connections
- Table or Shapefile Links
- Field Links

The first tier is the Connection. Connections are organized and managed by Connection Managers. There are two types of Connection Managers: a Database Connection Manager and a Shapefile Connection Manager. As the names imply, the first manages connections to databases and spreadsheets, and the second manages connections to Shapefiles. The Connection Managers are similar, and provide an interface for adding, editing, deleting, duplicating, and synchronizing Connections. To exchange data between the model and external files, a Connection must be created and then synchronized. The two synchronization operations that can be performed on a Connection are **Sync In** and **Sync Out**. **Sync In** synchronizes the model to the data contained in external files. In this case, the model acts as a "consumer" of the data, and external files act as the data "provider". **Sync Out** synchronizes external files to the data contained in the model. Thus, for **Sync Out**, the model is the data provider and external files are the consumers. Exactly what data is exchanged during synchronization depends on how the Connection is defined. Intuitively, a Connection must specify which files are to be connected to the model, and what data in each file is to be exchanged.

The second tier is the Table or Shapefile Link. A Database Connection uses these links to gather and store information. Each Connection can contain one or more Table or Shapefile Links. Each of these links specifies the type of external file with which to exchange data (implied with Shapefile links), the name of the file, and, if the file contains multiple tables, which table within the file is of interest.

The third tier of the system is the Field Link. Each Table or Shapefile Link uses one or more Field Links to specify exactly what data in the external file is going to be exchanged. A Field Link defines the fundamental mapping between a field in an external file and a field in the model. For example, a field link may be used to "map" the GRND_FT field of an external database file to the Ground Elevation attribute of the model.

In summary, a Connection defines a link between the model and external files. Table or Shapefile Links and Field Links are used to specify files, tables, and fields to be linked. Once a Connection is created, it can be synchronized in or out. The synchronization action will update models ("in" direction) or the external files ("out" direction).

The rest of this chapter provides details on the dialogs and windows used to interact with the GIS/Database Connection system. Although Database Connections and Shapefile Connections are similar in concept, there are differences in the interfaces and options. Therefore, they will be discussed in separate sections.

13.1 Database Connections

13.1.1 Database Connection Manager

This manager, accessed from the **File** pull-down menu, helps you track and work with database connections. On the left-hand side of this dialog is a list of the current database connections.

There are several options available from the manager, including:

Add – Create a new database connection using the **Database Connection Editor**.

Edit – Change the configuration of the currently selected connection. This will open the **Database Connection Editor**, where you can rename the connection, change the associated database files, and perform other changes to the connection configuration.

Duplicate – Create an identical connection to the selected one. This feature is very helpful when defining two or more connections with many similar attributes.

Delete – Remove the selected connection from the list.

Synchronize In – Update the network attributes from the databases defined in the selected connection.

Synchronize Out – Update all databases in the connection from the current status of the model.

Reset – Return a highlighted standard database import or export connection to default settings.

When synchronizing in, output fields such as hydraulic grade line or computed pipe flow will not be updated. If an attempt is made to update an output field during a **Synchronize In** operation, a "Read Only Warning" will be issued to the status log, indicating which attribute could not be updated.

When synchronizing out, all mapped information will be overwritten in the database files, including input and output conditions. If you do not want your input values overwritten upon synchronizing out, simply duplicate the connection. Then, edit one connection such that it includes only the values you want to synchronize in, and one that includes only the values you want to synchronize out.

When synchronizing out, be sure that the model element labels are of the same data type as the database column you are mapping to. For example, if you were to synchronize in from a database where your pipe identifier was a numeric value, then any changes or additions to the pipes in the model should also use a numeric labeling scheme. Select Element Labeling from the Tools menu and remove the appropriate element prefixes before any changes are made to the model. Otherwise, synchronizing out to the database will yield erroneous results.

To access the **Database Connection Manager**, select the **File\Synchronize\Database Connections** menu item. From this dialog, there are two ways to get to the **Database Connection Editor**. You can click **Add** to create a new connection, or select **Edit** to change an existing connection.

13.1.2 Standard Database Import/Export

The **Database Connection Manager** is initialized with four database connections for importing and exporting model data using simple **File** menu commands. These standard connections are as follows:

1. **[Project Export – SI]** – Used for the **File\Export\Database** command when the global unit system is set to System International.

2. **[Project Export – US]** – Used for the **File\Export\Database** command when the global unit system is set to US Customary.

3. **[Project Import – SI]** – Used for the **File\Import\Database** command when the global unit system is set to System International.

4. **[Project Import – US]** – Used for the **File\Import\Database** command when the global unit system is set to US Customary.

The purpose of the standard database connections is to provide a powerful yet easy to use method of exposing the model data to external applications using a standard database format, Microsoft Access database (.mdb). This method is powerful because it provides you with all the flexibility and functionality of a user-defined database connection, such as unit conversion and type coercion. It is easy to use because it is predefined with all of the standard model data, and requires nothing more than a file name to execute.

The standard database connections are almost identical to user-defined database connections with the following exceptions:

- Standard connections cannot be deleted.
- The label of a standard database connection cannot be changed.
- The target database for a standard database connection is determined at the time it is synchronized. During a **Synchronize In** operation, you will be prompted to choose an existing Microsoft Access Database (.mdb). During a **Synchronize Out,** you will be prompted for the name of a new Access database. If an existing filename is chosen, a warning will indicate that the existing file will be overwritten.
- The field names of the external database tables are editable from within the **Table Link** editor.
- The **Database Type** (on the **Table Link** editor) cannot be changed.
- Standard connections can be reset to their factory default values. To do this, select a standard connection from the list in the **Database Connection Manager** dialog, and click the **Reset** button.

By default, the standard database connections include a table link for each element type, and field links for all the attributes related to that element type, with some minor exceptions. The default units for the specified unit system (SI or US) are used for unitized attributes. The **Key/Label** field is designated as the key field for each of the table links, and it is created as an index for the table during database creation. No duplicates are allowed.

As noted above, the field links external field names can be edited directly within the table link editor. It is valid to have more than one internal attribute "mapped" to a single external field name. Although this is not the case for the standard connections in their factory default state, you can create this condition. Under this condition the following behaviors will be observed:

- **Import (Synchronize In)** – All of the attributes will be populated with the value of the database field if it is a valid value for the specified attributes.
- **Export (Synchronize Out) –** The database field will be populated with the last non-nil attribute value.

If an existing filename is chosen during export, the existing database file will be overwritten. Therefore, any custom tables, queries, forms, etc., present in that database will be lost.

> **Model data that are typically a collection of data (e.g. SewerCAD dry weather unit loads, StormCAD watershed areas and rational C coefficients, and WaterCAD junction demands) cannot be written to a single record, and are therefore not exported to the database. However, if these collections only contain a single item, that single item will be transferred to and from the database during export and import, respectively.**
>
> **By default, the Standard Database Export creates Office 2000 Access files. These files cannot be read with Office 97. If you want to use Office 97, you need to use a text editor to edit the HAESTAD.INI file located in your HAESTAD directory and replace the line:**
>
> ```
> ConnectionDatabaseFormat=0
> ```
>
> **With:**
>
> ```
> ConnectionDatabaseFormat=3
> ```
>
> **Basically, a value of 3 results in the program creating an Office 97 Access file, whereas a value of 0 will have the program generate an Office 2000 Access file.**

Use the **File\Import\Database** menu item to import data using the standard database connections.

Use the **File\Export\Database** menu item to export data using the standard database connections.

Use the **File\Synchronize\Database Connections** menu item to view or edit the standard database connections.

13.1.3 Database Connection Editor

The **Database Connection Editor** is used for defining the group of table links to be included in the connection. The **Database Connection Editor** is a tabbed dialog, with tabs for **Database Connection** and **Synchronization Options**.

There are three standard operation buttons at the bottom of the dialog:

- **OK –** Accept the current condition of the connection including any changes that have been made.
- **Cancel** – Close the **Database Connection Editor** without saving any changes.
- **Help** – Open the context-sensitive Help system.

To use the Database Connection Editor, select **File\Synchronize\Database Connections** from the main menu. This will open the **Database Connection Manager**. From this dialog, there are two ways to get to the **Database Connection Editor**. You can click **Add** to create a new connection, or select **Edit** to change an existing connection.

Database Connection Tab

The **Database Connection** tab of the **Database Connection Editor** provides an interface for the standard attributes of a connection. It contains the following:

- **Connection Label –** A required unique alphanumeric identification for the connection. This is the label that appears in the list on the **Database Connection Manager** dialog.
- **Table Links** – Provide basic information about each table link, such as the referenced database file, the specific table within the database, and the type of table that is referenced. A table link can be highlighted from the list, at which point the following commands can be performed using the buttons on the right side of the dialog:

- **Add –** Add a new table link. If there are no table links currently defined for this connection, this will be the only button available.
- **Edit** – Change the characteristics of the selected table link, such as the referenced file or table, or the mapping of the table's field links.
- **Duplicate –** Duplicate the selected table. This command is very helpful when defining two or more table links with similar attributes.
- **Delete** – Delete the selected table link from the connection.

Select **File\Synchronize\Database Connections** from the main menu. This will open the **Database Connection Manager**. Click **Add** to create a new connection, or select **Edit** to change an existing connection. From the **Database Connection Editor**, click the **Database Connection** tab.

Synchronization Options Tab

The **Synchronization Options** tab of the **Database Connection Editor** provides an interface for some of the behaviors of the connection. These options cannot be accessed until the Table Links are defined, and are as follows:

- **Add Objects to destination if present in source –** If this option is selected, when performing a synchronize out for example, elements that are present in the model but are not found in the database file will be created in the database. If this is not checked, only the elements that are present in both the model and the database will be updated.
- **Prompt before adding object –** If this is checked, you will get a dialog notifying you of each unmapped element in the source and asking if you would like to create a new element in the destination. If this is not checked, the additional elements will be automatically created in the database.
- **Remove from destination if present in source** – If this is checked when synchronizing out, elements that are present in the database but not in the model will be deleted from the database. If this is not checked, the unmapped elements will be ignored.
- **Prompt before remove** – If this is checked, you will get a dialog notifying you of each unmapped element in the destination and asking if you would like to remove that element. If this is not checked, the additional elements will be automatically removed from the database.

In order to be successfully created from the database, Pipe Elements must have a Start and Stop node associated with them. This association can be established by mapping the '+ Start Node' and '+ Stop Node' attributes in the pipe table link, or by the '+ In Link' or '+ Out Link' of a node table link. Mapping both the pipe table and node table attributes may result in the reading of redundant data causing the connection to fail.

By default, elements created from a database are located at coordinate (0,0). This behavior can be overridden by mapping the X and Y or Northing and Easting attributes of the node elements.

Select **File\Synchronize\Database Connections** from the main menu. This will open the **Database Connection Manager**. Click **Add** to create a new connection, or select **Edit** to change an existing connection. From the **Database Connection Editor**, click the **Synchronization Options** tab.

Database Table Link Editor

The **Table Link** editor provides an editing tool for defining or modifying a table link. This dialog is separated into two groups, one dealing with the file and table information, and the other dealing with the field links (attribute mapping).

The general table link information includes:

- **Database Type –** Type of database to which the link will be made. There are many types of external files that can be linked to the model. Among these are Btrieve, Dbase, Excel, FoxPro, Jet (.mdb files, such as Access), Lotus, and Paradox, as well as Oracle, Sybase, and SQl server or any other Open Database Connectivity (ODBC) compliant database.
- **Database File** – File referenced by the table link. To browse directories and specify a file path, click the **ellipsis (...)** button.
- **Database Table** – Once the external file has been selected it will be scanned for tables (or worksheets), which will then be available for selection from this field. Only one table can be linked to for each table link, but table links can be easily duplicated and edited from the **Database Connection** editor.
- **Table Type –** Define the type of data that can be mapped for this particular table link. For example, a *Pipe* type of table link means that the available model attributes to be mapped are items such as material, roughness coefficient, flow rate, and velocity.
- **Key / Label Field** – Key by which the entire database-model mapping is defined. The model references each element by a unique alphanumeric label, and the database must contain the same labels in one of the columns.

The Field Links group is a manager for the attribute mapping. The tabular list in this group has three field columns:

- **Model** – Each item in this column is an attribute in the model that is being mapped to the database. The list of available attributes depends on the type of the table.
- **Database –** Each item in this column is a heading from the database table, which correlates to the item in the model being mapped.
- **Unit –** This column defines the units of the values in the database. During a synchronization operation, the values will automatically be converted to the appropriate units to maintain the desired unit systems in both the model and the database. No conversion on your part is required.

In addition to the standard table operations of **Insert**, **Duplicate,** and **Delete**, the Field Links manager offers the following additional operation:

- **Select –** Opens the **Select Field Links** dialog for an efficient method of selecting the fields of interest from the available model fields.

Select **File\Synchronize\Database Connections** from the main menu. This will open the **Database Connection Manager**. Click **Add** to create a new connection, or select **Edit** to change an existing connection. From the **Database Connection** tab of the **Database Connection** editor, click **Add** or **Edit**.

Select Field Links

The **Select Field Links** dialog provides an easy-to-use interface for populating the Field Links group of the **Table Link** editor or **Shapefile Link** editor.

The dialog contains two lists:

- **Available Items** – Model attributes that are available for mapping in the current Table or Shapefile Link.
- **Selected Items –** Model attributes that have been selected for mapping.

The following buttons are provided to move items from one list to the other:

- **[>]** Move the selected item or items from the **Available Items** list to the **Selected Items** list.
- **[>>]** Move all items from the **Available Items** list to the **Selected Items** list.
- **[<]** Move the selected item or items from the **Selected Items** list to the **Available Items** list.
- **[<<]** Move all items from the **Items Selected** list to the **Available Items** list.

13.1.4 ODBC

ODBC, which stands for Open Database Connectivity, is a standard programming interface developed by Microsoft for accessing data in relational and non-relational database management systems (DBMSs). Using ODBC, applications such as Haestad Methods' engineering software can access data stored in many different PC, minicomputer, and mainframe DBMSs, even though each uses a different storage format and programming interface.

The ODBC architecture conceptually consists of three parts:

1. **The application program** – Supplied by Haestad Methods.
2. **The Data Source Administrator Program –** Embedded in Microsoft Windows.
3. **The low-level drivers for accessing specific databases** – Supplied by your database vendor.

Although most computers with Windows will have ODBC present, the exact databases you can interface via ODBC will depend on the databases and drivers installed on your computer.

ODBC is powerful because it is very generic and can access many database systems, including mainframe, GIS, and legacy systems. However, because ODBC must be very general, it is slower, more complex, and more difficult to use than working directly with a database. When you have the option to work directly with a database, you will usually find it is faster and easier than going through ODBC.

For specific information about ODBC in your environment, see your database vendor's documentation. For general information on ODBC, see the online Help for the ODBC Data Source Administrator Program. To find the Administrator Program, go to the Control Panel for your computer and double-click the **ODBC** icon. Choose the **Help** button on the dialog that appears, then go to the Help Contents.

ODBC Database Type

The first field of the database connection **Table Link** editor is for the Database Type. The list box displays the external databases and versions supported by the Database Connection feature. One of the Database Types you can select is ODBC. This does not refer to a specific database or version. Rather than being a specific database this is actually a link to the ODBC Data Source Administrator Program running on your computer. This link will provide an interface between the Haestad Methods' Database Connection feature and a specific DBMS and source database file.

ODBC Database File

If you have selected ODBC as the Database Type, when you click the **ellipsis (...)** button next to the **Database File** field the ODBC Data Source Administrator Program will take over and offer a list of the ODBC data sources installed on your computer. Depending on how your computer is configured, you may see database systems or actual database files from which to choose.

> **You will also see database systems such as Microsoft Jet or Excel that are also supported directly via choices in the Database Type list. In general, the Database Connection feature will work faster and better by choosing these database systems directly rather than going through ODBC.**

If you choose a data source from the Administrator Program, upon returning to the **Table Link Editor** you will see an ODBC "connect string" in the Database File field, rather than a file path. This connect string is a series of key = value pairs, separated by semicolons. It specifies the database location, security parameters, and access options needed by the particular ODBC driver you are using. In general, you should not edit this string in any way as you could introduce an error that would prevent the ODBC driver from accessing the data source you have selected.

> **If you are unable to successfully synchronize to the data source using the default form of ODBC string, it is possible that you may need to add some parameters to the string that are specific to your environment. See your database vendor's ODBC documentation for details.**

Synchronizing Via ODBC

Once you have successfully created and entered the data for a database connection that uses ODBC, the **Synchronize In** and **Synchronize Out** operations perform as they do for any other database format. However, note that ODBC databases are accessed with slightly different internal mechanisms, and thus may generate different error conditions. If a synchronization fails to complete, see the status log for error messages. Note the project or database object the program was processing when the error occurred. Refer to your database vendor's documentation for detailed information on any errors reported.

Using ODBC to access SQL Server databases will result in an error #3197 if the synchronization attempts to delete a database record. The only known workaround currently is to uncheck "Remove Objects" on the **Synchronization Options** tab of the **Database Connection Editor**.

ODBC Database Tables and Fields

There are many complexities in successfully accessing ODBC databases. You will know if there are problems on your machine because the Database Table or other database-related fields will not have any entries in the associated drop-down lists.

If this happens, confirm that ODBC is installed and operating correctly on your computer. Double-check that the ODBC data source you are trying to reference actually exists and is accessible by other programs in your environment. Check the HAESTAD.LOG file for error messages pertaining to ODBC. If none of these steps helps you correct the problem, please call HMI Technical Support.

Given the diversity of ODBC database drivers and the difficulty of reproducing your networked computing environment, we cannot guarantee that the Database Connection feature will function with all ODBC databases. However, we will try to determine the source of your problem and offer a fix or workaround if possible.

If you edit the connect string manually, you will need to re-enter the dependent fields such as Database Table and Field Links.

13.1.5 Sharing Database Connections between Projects

When WaterCAD works with database connections, it is using a file with a .HDC extension, which stores the information regarding database files, table links, and field mapping.

When you open a WaterCAD project file (.WCD), WaterCAD first looks for a file in the same directory and with the same filename but with the .HDC extension. If it finds this file, it uses the database connectivity information contained therein. If it does not find this file, then it defaults to a file in the installed WaterCAD directory called WTRC.HDC.

If you are working on a local drive and you have several project files, all of which reference common connection information, let your project files automatically default to the WTRC.HDC file. Any connectivity changes that you work on in one project will be automatically reflected when you open any other project.

If there are several people working on different projects on different computers, but they still wish to have common connectivity information, the appropriate .HDC file can be copied (and renamed if necessary) to the individual local drives.

Preventing Database Connectivity Sharing between Projects

There are times when shared connectivity can be more cumbersome than helpful, such as when there are many projects, each with different database connectivity. At these times, it is more useful to have the connectivity associated with one specific project rather than with all projects. To do this, simply copy the WTRC.HDC file from the installed WaterCAD directory to the same location as your project file, and rename it to the same name as your .WCD file.

For example, if your WaterCAD project file is PROJECT1.WCD, rename WTRC.HDC to PROJECT1.HDC. The connections in PROJECT1.WCD can then be modified without the effects being reflected in any other projects.

13.1.6 Database Connection Example

To connect your model to an external file, take the following steps:

1. From the **File** menu, select **Synchronize\Database Connections** to open the **Database Connection Manager**. Click **Add**.
2. In the **Database Connection Editor**, type a label for your Connection.
3. Click **Add** to create a new table link. This will take you to the **Table Link** editor.
4. Select the type of file to which you would like to link, then click the **ellipsis (…)** button to browse for and select your database file.
5. Choose the table to which you would like to link, and the type of table.
6. Choose the **Key/Label Field** to define the column in the database that contains the labels of the elements to be synchronized.
7. Define as many field links as you want by selecting the model attribute and the associated database column and unit.
8. Click **OK** to exit the **Table Link Editor**.
9. Click **OK** again to exit the **Database Connection Editor**.
10. You should be back at the **Database Connection Manager**. You can leave this dialog and return to the model, or you can choose to **Synchronize In** to the model from the database, or **Synchronize Out** to the database from the model.

13.2 Shapefile Connections

13.2.1 Shapefile Connection Manager

This manager is identical to the **Database Connection Manager**, except that it helps you to track and work with Shapefile Connections rather than Database Connections. Only a brief description of each dialog control is presented here. Please refer to the **Database Connection Manager** topic for a more detailed explanation.

- **Add –** Create a new Shapefile connection. This will open the **Shapefile Connection Wizard**.
- **Edit** – Change the configuration of the currently selected connection. This will open to the **Shapefile Connection Editor**.
- **Duplicate** – Duplicate the selected connection.
- **Delete** – Delete the selected connection from the list.
- **Synchronize In** – Update the network attributes from the Shapefiles defined in the selected connection.
- **Synchronize Out –** Update all Shapefiles within the connection from the current status of the model.

> **See the Overview at the beginning of this chapter for a general discussion of Shapefile Connections.**

To open the **Shapefile Connection Manager**, select the **File\Synchronize\Shapefile Connections** menu item. Select **No** when prompted to create a new shapefile connection.

Shapefile Connection Wizard

The **Shapefile Connection Wizard** provides an easy-to-use interface for defining a new Shapefile Connection. It is similar to the **Shapefile Import Wizard**, but has a few additional steps. The major steps in the wizard are as follows:

- **Label –** Enter an alphanumeric label to uniquely identify the Shapefile Connection.
- **Select Element Types –** Choose the types of network elements you wish to connect to Shapefiles.
- **Shapefile Synchronization Options –** Specify the spatial data unit, and configure other options.
- **Import Shapefile Link Editor –** Choose the Shapefile to which you want to connect and specify the details of the link.
- **Synchronize Now?** – Choose whether or not you want to synchronize the Shapefile Connection when finished with the wizard. You can choose to synchronize in either direction.

Shapefile Connection Label

The **Shapefile Connection Label** window allows you to enter a unique alphanumeric label for your Shapefile Connection. This window is presented during both the **Import** and **Export Shapefile Connection Wizards**, as well as the **Shapefile Connection Wizard**.

Synchronize Now?

The last step in the **Shapefile Connection Wizard**, the **Synchronize Now** window, allows you to specify that you wish to synchronize the Shapefile Connection immediately after editing it in the wizard. The following options are available:

- **Synchronize Shapefile Connection –** Check this box if you wish to synchronize the connection immediately upon clicking the **Finished** button. By default this box is checked. If you uncheck it, you will return to the **Shapefile Connection Manager** after clicking the **Finished** button.
- **In** – Click this radio button if you wish to synchronize the connection in to the model. This will update the model data from the Shapefiles defined in the connection.
- **Out** – Click this radio button if you wish to synchronize the connection out to the Shapefiles defined in the connection. This will update the Shapefiles from the model.

13.2.2 Shapefile Connection Editor

The **Shapefile Connection Editor** is similar to the **Database Connection Editor**. It offers the tabs for **Shapefile Connection** and **Synchronization Options**.

Shapefile Connection

The **Shapefile Connection** tab of the **Shapefile Connection Editor** is similar to the **Database Connection** tab of the **Database Connection Editor**. It contains the following:

- **Connection Label** – A unique alphanumeric identification for the connection. This is the label that appears in the list on the **Shapefile Connection Manager** dialog.
- **Table Links –** This list is similar to all of the program's managers. It provides basic information about each Shapefile link, such as the referenced Shapefile, the feature type of the Shapefile, and the type of element which is referenced. As with the other managers, a Shapefile link can be highlighted from the list, at which point the following commands can be performed using the buttons on the right side of the dialog:
  - **Add** – Define a new Shapefile link. If there are no table links currently defined for this connection, this will be the only button available. Clicking this button invokes the **Shapefile Link Wizard**.
  - **Edit –** Change the characteristics of the selected Shapefile link, such as the referenced file or the mapping of the Shapefile's field links. Clicking this button also invokes the **Shapefile Link Wizard**.
  - **Duplicate** – Create an identical Shapefile link to the selected one. This is very helpful when defining two or more Shapefile links with similar attributes.
  - **Delete –** Remove the selected Shapefile link from the connection.

To use the **Shapefile Connection Editor**, do the following:

Select **Synchronize\Shapefile Connections** from the **File** menu.

- If you do not currently have any Shapefile connections defined, you will be prompted to indicate if you wish to create one now. If you answer **Yes**, you will be automatically taken to the **Shapefile Connection Wizard**.
- If there are connections already defined, or if you answer **No** to the prompt to create one now, you will be taken to the **Shapefile Connection Manager**. Select **Add** to open the **Shapefile Connection Wizard**.

13.2.3 Shapefile Link Wizard

The **Shapefile Link Wizard** is used when adding new Shapefile Links to a Shapefile Connection, or when editing the existing links of a Shapefile Connection. The first step of the wizard is bypassed when editing an existing link. The basic steps of the wizard are as follows:

- **Select Element Type** – Similar to the **Select Element Types** window for importing Shapefiles, except that radio buttons are used rather than check boxes. This is because a Shapefile Connection represents a single element type.
- **Import Shapefile –** Choose the Shapefile to which you would like to connect, and choose the **Key/Label Field** to specify the column in the Shapefile that contains the matching element labels in the network. Define as many field links as necessary. For each link, specify the model attribute, the associated Shapefile column, and the Unit in which the Shapefile attribute is stored.
- **Shapefile Link Summary –** Quick review of the details specified in the wizard.

As with all wizards, you can move forward or backward through the process to make changes. Click the **Finished** button when you are done making changes to the Shapefile Link.

Shapefile Link Summary

The **Shapefile Link Summary** window provides an opportunity to review the details of the Shapefile Link before completing the editing process. The following information is provided in the summary window:

- **Type** – Type of element represented by this Shapefile Link.
- **Shapefile** – Full path and file name of the Shapefile referenced by this Shapefile Link.
- **Key/Label Field –** Shapefile field used to map Shapefile records to their corresponding network elements in the model.
- **Attributes Mapped –** Number of Field Links mapped in this Shapefile Link.

13.2.4 Import Shapefile Wizard

The **Import Shapefile Wizard** will guide you step-by-step through the process of importing ESRI Shapefiles. These are the basic steps for importing Shapefiles:

- **Select Element Types –** Choose the type of network elements you wish to import.
- **Shapefile Synchronization Options –** Specify the spatial data unit and configure other options.
- **Import Shapefile –** Choose the Shapefile you would like to import, and choose the **Key/Label Field** to specify the column in the Shapefile that contains the matching element labels in the network. Define as many field links as necessary. For each link, specify the **network** attribute, the associated **Shapefile** column, and the **Unit** in which the Shapefile attribute is stored.
- **Create Shapefile Connection? –** Choose whether or not you want to establish a Shapefile Connection. The Shapefile Connection allows you to update the Shapefile with values from your model, or to update your model from the Shapefile.

While using the wizard, you can move forward or backward through the process to make changes by clicking the **Next** and **Back** buttons. Click the **Finished** button when you are done making changes to import Shapefiles.

To access the **Import Shapefile Wizard**, select **File\Import\Shapefile** from the pull-down menu.

Select Element Types

The **Select Element Types** window is used for selecting the types of network elements that are of interest when importing and exporting Shapefiles, or when creating a Shapefile Connection. The window contains a list of network element types, and a check box precedes each type.

To select an element type for Shapefile Import, Export, or Connection, put a check mark in the corresponding box.

Shapefile Synchronization Options

Several options are available to customize the Shapefile synchronization process. The Shapefile Synchronization Options are available for editing in the **Import Shapefile Wizard** and editing a Shapefile Connection.

The first group of options is only available when editing a Shapefile Connection. These options are exactly the same as their counterparts in Database Synchronization Options, and are as follows:

- **Prompt before add**
- **Prompt before remove**

Unlike the Database Synchronization Options, the Shapefile Synchronization Options do not allow for optionally adding or removing elements. When synchronized, Shapefiles and the model will contain exactly the same number of records for the specified element type. For example, suppose a Shapefile contains a record for the junction labeled 'J-1'. When this Shapefile is synchronized into the model, the model will automatically add a junction labeled 'J-1' if none currently exists. Likewise, if 'J-1' is removed from the model and then synchronized out to the Shapefile, the record for 'J-1' will automatically be removed from the Shapefile. You have no control over this.

The rest of the options are available during the **Shapefile Import Wizard** and from the **Shapefile Connection Editor**.

Shapefile Unit – Choose a unit from the available list. This is the unit of the spatial data in the Shapefile. For example, if the X- and Y-coordinates of the Shapefile represent feet, choose feet from the choice list. If they represent meters, choose meters. This unit must be the same for every Shapefile in the Shapefile Connection. If you wish to import Shapefiles that have different spatial data units, create a separate connection for each unit.

When Missing Connectivity Data

As noted in the **Table Link Editor** topic, to create a pipe from an external file it is necessary for a pipe to have a start node and stop node associated with it. Typically, these "connectivity" associations are created by synchronizing the '+ Start Node' and '+ Stop Node' attributes of the pipe. Since a Shapefile contains spatial data, it is also possible to establish these associations based on the location of nodes relative to the end points of the pipe. The following options allow you to customize this behavior:

- **Establish By Spatial Data** – Check this box to configure the synchronization so that any missing connectivity data (start node, stop node, or both) for a pipe will be established from the spatial data if possible.
- **Tolerance** – This value represents the distance to be searched when trying to locate nodes for establishing connectivity for a pipe. All nodes within the tolerance of a pipe's end point will be collected and the closest node will be selected for connection.
- **Create Nodes if None Found –** Check this box if you would like nodes to be created during the synchronization when no nodes are found within the specified tolerance of a pipe's end point. If this box is not checked, and no nodes are found within the tolerance, the pipe will not be created because it has insufficient connectivity data.

Import Shapefile Link Editor

The **Import Shapefile Link Editor** is similar to the **Database Table Link Editor**. Refer to that topic for detailed information on the following Shapefile Link parameters:

- **Shapefile –** Location of the file that is being referenced by the Shapefile link. This is identical to the Database File parameter of the **Table Link Editor**.
- **Key/Label Field –** Key by which the entire Shapefile/model mapping is defined.
- **Field Links** – Identical to the Field Links group of the Database Table Link Editor.

Create Shapefile Connection

The **Create Shapefile Connection** window provides an opportunity during a Shapefile Import or Shapefile Export to specify that a persistent connection containing the Shapefile Links and Synchronization Options be created. This Connection can be used at a later time to synchronize the model and the Shapefiles. The Create Shapefile Connection window has the following parameters:

- **Add Shapefile Connection** – Check this box if you wish to add a persistent Shapefile Connection to the Shapefile Connection Manager. By default this box is checked.
- **Label –** Specify an alphanumeric label for the Connection. This field is only editable when the **Add Shapefile Connection** box is checked.

To access the **Shapefile Import Wizard**, select **File\Import\Shapefile** from the pull-down menu.

Shapefile Import Example

Follow these steps to import one or more Shapefiles into a new or existing model:

1. From the **File** menu, select **Import\Shapefile** to access the **Import Shapefile Wizard**.
2. Choose the element types that you wish to import by clicking one or more of the check boxes in the list, then click the **Next** button.
3. Configure the options for this import. First, select the unit for the spatial data of the Shapefile. Then, if appropriate for your situation, click the **Establish by Spatial Data** check box in the **When Missing Connectivity Data** group, and enter a value in the **Tolerance** field. For more information regarding these options, refer to the section on Shapefile Synchronization Options. Click the **Next** button to proceed to the Shapefile Link Editors.
4. You will be presented with an **Import Shapefile Link Editor** for each element type you chose to import. Perform the following steps for each **Import Shapefile Link Editor**:

   Enter the name of the Shapefile you wish to import for the specified element type. Click the **ellipsis (…)**button to interactively browse for and select your Shapefile.

   Choose the Key/Label Field to define the column in the Shapefile that maps to the element labels in the model.

   Define as many field links as necessary by selecting the model attribute and the associated Shapefile column and unit, if appropriate. Use the **Select** button for making the selection process more efficient. Click the **Next** button.

5. Click the **Add Shapefile Connection** check box if you wish to create a persistent link between the Shapefile(s) you are importing and the model. If you choose to create a Shapefile Connection, enter an alphanumeric label to identify the connection. Click the **Finished** button to import the Shapefiles.

13.2.5 Export Shapefile Wizard

This program has the capability of exporting network elements in the ESRI Shapefile Format. The ESRI Shapefile is actually three files that together define the spatial and non-spatial attributes of a map feature. In the case of Haestad Methods hydraulic models, map features are network elements (e.g. pipes, junctions). Exporting Shapefiles creates brand new files. If you are exporting a Shapefile to a directory that already contains a Shapefile of the same name, the existing Shapefile will be completely overwritten. If you wish to update the Shapefile rather than overwriting it, use the Shapefile Connection feature.

The major components of the Wizard are as follows:

- **Select Element Types –** Choose the type of network elements you wish to export. Each type of network element will have its own Shapefile associated with it. This component is identical to the Import Wizard's **Select Element Types** component.

- **Export Shapefile Link Editor –** Enter a name for each Shapefile you wish to create. Each Shapefile name must be no more than eight characters in length, and should not be duplicated. Define as many field links as necessary. For each link, specify the *network* attribute. The shapefile variable will default to a preset calue which can be edited.

- **Create Shapefile Connection –** Choose whether you want to establish a Shapefile Connection for this Shapefile or not. The Shapefile Connection allows you to update the Shapefile with values from your model, or to update your model from the Shapefile. This component is identical to the Import Wizard's **Create Shapefile Connection** component.

While using the Wizard, you can move forward or backward through the process by clicking the **Next** and **Back** buttons. When you are finished defining the Shapefile, click the **Finished** button to create it.

To export a specific network element type as a Shapefile, choose **File\Export\Shapefile** from the pull-down menu. This opens the **Shapefile Export Wizard.**

Export Shapefile Link Editor

The **Export Shapefile link editor** is similar to the **Database Table link editor**, with the following differences:

Shapefile – The name and location for the file that is being exported. The Shapefile name is limited to eight characters.

The Field Links group is used to specify the attributes and Shapefile column headings that you wish to export, as follows:

- **Model** – Each item in this column is an attribute in the model that is being exported to the Shapefile. The list of available attributes depends on the type of table.

- **Shapefile** – Each item in this column is a column heading in the Shapefile being created, which correlates to the item in the model being mapped. By default, the heading are set to an all-capitals abbreviation of the attribute name, with spaces and periods replaced by the underscore character. The column heading can be changed, but must be less than ten characters long and cannot contain periods.

The spatial data in the Shapefiles being created will be in the current display unit for map coordinates. For example, if the X and Y or Northing and Easting values in the model are displayed in meters at the time of the export, then the spatial data in the Shapefiles created will also be in meters.

The values for the exported attributes will be in the current display units for that attribute. For example, if a junction elevation attribute is displayed in feet at the time of the export, the Shapefile will contain that value in feet.

Shapefile Export Example

Follow these steps to export one or more Shapefiles from the model:

1. From the **File** menu, select the **Export\Shapefile** menu item to access the **Export Shapefile Wizard**.
2. Select the element types that you wish to export by clicking one or more of the check boxes in the list, then click the **Next** button.
3. You will be presented with an **Export Shapefile Link Editor** for each element type you chose to export. Perform the following steps for each **Export Shapefile Link Editor**:
4. Enter the name of the Shapefile you wish to create for the specified element type. Click the **ellipsis (...)** button to interactively browse for a directory in which to store the Shapefile.
5. Define as many field links as necessary by selecting the model attribute and providing a name for the associated Shapefile column. Use the **Select** button for making the selection process more efficient. Click the **Next** button.
6. Click the **Add Shapefile Connection** check box if you wish to create a persistent link between the Shapefile(s) you are exporting and the model. If you choose to create a Shapefile Connection, enter an alphanumeric label to identify the connection. Click the **Finished** button to export Shapefiles.

13.2.6 Sharing Shapefile Connections between Projects

When WaterCAD works with shapefile connections, it is using a file with a .HSC extension, which stores the information regarding the shapefiles and field mapping for each element type.

When you open a WaterCAD project file (.WCD), WaterCAD first looks for a file in the same directory and with the same filename but with the .HSC extension. If it finds this file, it uses the shapefile connectivity information contained therein. If it does not find this file, it defaults to a file in the installed WaterCAD directory called WTRC.HSC.

Sharing Shapefile Connections between Projects

If you are working on a local drive, and you have several project files that all reference common Connection information, let your project files automatically default to the WTRC.HSC file. Any connectivity changes that you work on in one project will be automatically reflected when you open any other project.

If there are several people working on different projects on different computers, but they still wish to have common connectivity information, the appropriate .HSC file can be copied (and renamed if necessary) to the individual local drives.

Preventing Shapefile Connectivity Sharing between Projects

There are times when shared connectivity can be more cumbersome than helpful such as when there are many projects, each with different database connectivity. At these times, it is more useful to have the connectivity associated with one specific project, rather than with all projects. To do this, simply copy the WTRC.HSC file from the installed WaterCAD directory to the same location as your project file, and rename it to the same name as your .WCD file.

For example, if your WaterCAD project file is PROJECT1.WCD, rename WTRC.HSC to PROJECT1.HSC. The connections in PROJECT1.WCD can then be modified without the effects being reflected in any other projects.

13.2.7 Shapefile Format

An ESRI Shapefile actually consists of three separate files that combine to define the spatial and non-spatial attributes of a map feature. The three required files are as follows:

- **Main File** – A binary file with an extension of .SHP. It contains the spatial attributes associated with the map features. For example, a polyline record contains a series of points, and a point record contains X- and Y-coordinates.
- **Index File** – A binary file with an extension of .SHX. It contains the byte position of each record in the main file.
- **Database File** – A dBase III file with an extension of .DBF. It contains the non-spatial data associated with the map features.

All three files must have the same file name with the exception of the extension, and be located in the same directory.

13.2.8 Shapefile Connection Example

Follow these steps to connect one or more Shapefiles to the model:

1. From the **File** menu, select **Synchronize\Shapefile Connections**.
2. If you do not have any connections currently defined, you will be asked if you want to create a new one now. Select **Yes**. If you already have one or more connections defined, you will go to the **Shapefile Connection Manager**. Click **Add** to access the **Shapefile Connection Wizard**.
3. Provide an alphanumeric label to uniquely identify this new connection. Click the **Next** button.
4. Choose the element types that you wish to import by clicking one or more of the check boxes in the list and click the **Next** button.
5. Configure the options for this connection. First select the unit for the spatial data of the Shapefile. Then, if appropriate for your situation, click the **Establish by Spatial Data** check box in the **When Missing Connectivity Data** group, and enter a value in the **Tolerance** field. For more information regarding these options, refer to the section on Shapefile Synchronization Options. Click the **Next** button to proceed to the **Shapefile Link Editors**.
6. You will be presented with an **Import Shapefile Link Editor** for each element type you chose to import. Perform the following steps for each **Import Shapefile Link Editor**:
7. Enter the name of the Shapefile to which you wish to connect for the specified element type. Click the **ellipsis (…)**button to interactively browse for and select your Shapefile.
8. Choose the **Key / Label Field** to define the column in the Shapefile that maps to the element labels in the model.
9. Define as many field links as you want by selecting the model attribute and the associated Shapefile column and unit if appropriate. Use the **Select** button for making the selection process more efficient. Click the **Next** button.

10. Check the **Synchronize Shapefile Connection** box if you wish to synchronize the connection immediately upon clicking the **Finished** button.

11. If the **Synchronize Shapefile Connection** box is checked, choose whether you want to Synchronize In to the model from a Shapefile, or Synchronize Out to the Shapefile from the model.

12. Click the **Finished** button to synchronize the connection (if the **Synchronize Shapefile Connection** box is checked), or return to the **Shapefile Connection Manager**.

Notes

Chapter 14

Exchanging Data with CAD Software

14.1 AutoCAD Polyline to Pipe Conversion

14.1.1 Polyline to Pipe Conversion Overview

This feature allows you to quickly construct a network based on the entities contained in an AutoCAD drawing. Although this feature is called Polyline to Pipe, *Line* and *Block* entities can be converted as well. *Polylines* and *Lines* can be converted to pipes and *Blocks* can be converted to any available node type.

Building a model based on graphical elements can be an error-prone process. This is due to the fact that a drawing can appear to be correct visually, but may contain problems that are not readily apparent. For example, what appears to be a single line in a drawing could in fact be made up of many line segments, or it could be made up of 2 lines, one directly on top of another.

To help alleviate some of the problems that you may encounter during the import process, a comprehensive drawing review is also performed. During the conversion process, the network is analyzed and potential problems are flagged for review. After performing the conversion, the **Drawing Review** window will allow you to navigate to and fix any problems that are encountered.

The Polyline to Pipe conversion cannot be undone. Be sure to save your project before you begin.

You can import entities into an existing project. Polylines will automatically be connected to nodes within the specified Tolerance. You can add nodes to your project prior to performing the import.

Stand-Alone mode issues – You should take some time to clean up your AutoCAD drawing prior to performing the conversion. Look for entities that should not be converted, such as leader lines, and move them to their own layer. Turn off layers that you do not wish to convert. Do a quick review of your drawing and correct any potential conversion problems that you may find.

After performing the conversion, we recommend that you use the converted file as a DXF Background. This will greatly enhance your review process. If you change the entities in your background drawing to a gray color from within AutoCAD, it will make it easier to distinguish between foreground elements and background entities.

AutoCAD mode issue – You can interactively convert individual entities to pipes by using the Layout Tool.

14.1.2 Converting your Drawing in Multiple Passes

Depending on how your drawing layers are set up, you may be able to save yourself a considerable amount of data entry time by converting your drawing in multiple passes.

For example, if your 12" pipes are located on a "12InchPipes" layer, 18" pipes are on a "18InchPipes" layer, etc., you can import layers one at a time. Just set up your prototypes prior to importing that layer.

To assist you in this process, your conversion settings will be retained between imports. Therefore, on subsequent passes you will simply need to revise your prototypes and specify the next layer to be imported.

This same technique can be used when importing blocks.

14.1.3 Polyline to Pipe Wizard

The **Polyline to Pipe** wizard will guide you step-by-step through the process of converting your entities to elements.

- **Step 1 –** The import behavior depends on the mode in which you are working:

Stand-Alone – Specify the DXF file that you would like to import.

AutoCAD – This step is skipped -- you will be asked to select the entities to convert before accessing the Wizard.

- **Step 2** – Specify the polyline to pipe conversion options.
- **Step 3** – Specify how T-intersections are to be handled.
- **Step 4** – Specify how blocks should be converted (optional).
- **Step 5** – Configure prototypes.
- **Step 6** – Specify the layers to be imported.

To access the Polyline to Pipe Wizard:

Stand-Alone mode – Select **File\Import\Polyline to Pipe** from the main menu.

AutoCAD mode – Select **Edit\Change Entities to Pipes** from the main menu.

Polyline to Pipe Wizard – Step 1 (Stand-Alone mode only)

This step allows you to specify the DXF file to be imported.

If you are running in AutoCAD mode, this step will be skipped. AutoCAD mode users will be asked to select the entities to be converted before accessing the Polyline to Pipe wizard.

- **DXF Filename** – Specify the name of the DXF file you would like to import. Use the **Browse** button to select the file interactively.

- **DXF Unit** – Specify the DXF conversion unit (the unit that your DXF file is in). For example, if your drawing is in metric, specify meters (m). If your drawing is in architectural units, specify inches (in).

Polyline to Pipe Wizard – Step 2

This step allows you to specify the following Polyline to Pipe conversion options:

- **Connectivity tolerance** – Polylines whose endpoints fall within the specified tolerance will be connected to the same node. A default tolerance is supplied based on the current scale. This is generally a good starting point, but you may wish to increase or decrease this default tolerance depending on your particular drawing. If you complete the conversion process and find that the tolerance was not correct (pipes that should be connected were not, or vise versa), you may wish to repeat the conversion process using a new tolerance.

- **Specifying which entities to convert** – You can optionally convert Polylines, Lines, or both. You generally want to convert both Polylines and Lines. However, if your drawing is set up so that Polylines are always used to represent pipes and Lines are used for annotation purposes, you may wish to convert only Polylines.

- **Handling missing nodes at polyline endpoints** – A pipe can only be created if there is a node at both endpoints. If a node cannot be found at a polyline endpoint, a node must be added. Otherwise, the pipe cannot be converted. This option allows you to specify whether or not a node is created, and, if so, the default type of element to create.

In general, you will want to create a default node at polyline endpoints. However, if your network already contains nodes at polyline endpoints, or if your drawing contains blocks at polyline endpoints that are to be converted to nodes, you may wish to specify that the polyline not be converted. Polylines that can not be converted, because one or both end nodes are missing, will be flagged for review at the end of the conversion process.

If the conversion does not yield the desired results, you can repeat the conversion process using different settings. Be sure to save your project before performing the conversion.

Polyline to Pipe Wizard – Step 3

This step allows you to specify how T-intersections (pipe split candidates) should be handled.

Nodes that fall within the specified tolerance of a pipe are referred to as *pipe-split candidates.* There are two ways to handle these:

- **Join the pipes at the intersection** – The *pipe-split candidate* will be used to split the intersecting pipe.

- **Do not join the intersecting pipes** – *Pipe-split candidates* will be flagged for later review using the **Drawing Review** window.

The tolerance that you specify in Step 2 will also be used for T-intersection processing.

Polyline to Pipe Wizard – Step 4 (Optional)

This step allows you to optionally convert AutoCAD blocks to nodes.

If you would like to convert blocks to nodes, activate the *Yes* toggle. A table with two columns will appear, allowing you to map the AutoCAD blocks you would like to convert to any of the available node element types. The *AutoCAD block* column provides you with a list of available blocks to convert. The *Element* column provides you with a list of available node element types.

For each AutoCAD block you would like to convert, specify the type of node element you would like to create.

When you select an AutoCAD block, the preview pane will display the graphical representation of that block. This step will be skipped if there are no AutoCAD Blocks in your drawing.

Polyline to Pipe Wizard – Step 5

Before performing the conversion, you may wish to configure your prototypes with default data. During the conversion process, elements will be created using the specified defaults.

Click a button to configure the defaults for the associated element.

Polyline to Pipe Wizard – Step 6

Specify the layers that contain the entities you would like to convert. Use the Preview button to preview the elements on the selected layers. This step can be used in conjunction with the Prototype step to allow you to convert your drawing in multiple passes.

It is recommended that you process your drawing prior to performing the import. If your drawing contains layers that you do not wish to import, turn them off from within AutoCAD and they will be ignored during the import process.

Polyline Conversion Problem Dialog

This feature is present in stand-alone mode only. This dialog displays the reason that a polyline was not converted after running the **Polyline to Pipe Wizard**.

Drawing Preview

Use the **Preview** button to view the elements in your DFX file that will be converted.

Next to the **Preview** button is a checkbox labeled **Only include elements that will be converted**.

Turn the toggle on to preview the entities that will be converted. The entities to be converted are based on the settings you specified in the **Polyline to Pipe Wizard**, such as type of line entities, blocks, and layers to be converted.

Turn the toggle off to preview all entities.

14.2 Import/Export of DXF Files

14.2.1 Import a DXF from AutoCAD or MicroStation

To import background graphics in Stand-Alone mode from another drafting program, you must first export a DXF file from your CAD program. This step is usually as simple as selecting an item from a pull-down menu in that program, such as **File\Export\As DXF**, or similar command. Once the DXF file has been created, it can be imported into this program as follows:

1. Select the **File\Import\DXF Background** command from the pull-down menu to access the **Import DXF File** dialog.
2. Select the DXF file you wish to import, and click the **Open** button.

14.2.2 Exporting a DXF file

A project file can be saved in a format that can be used by AutoCAD and other common CAD-based applications. When you use the Export command, a window appears so that you can enter the file name, drive, and directory of the DXF file you are saving. A status bar appears at the bottom of the screen as the file is being exported.

To export the drawing plan view, select the **File\Export** pull-down menu option.

> **You will be able to redefine all elements, except pipes, as blocks in AutoCAD. Pipes will be exported as polylines, so you will be able to set their line weight in AutoCAD.**

14.2.3 Redefining WaterCAD Blocks in AutoCAD

If you would like to change the appearance of these blocks in your AutoCAD drawing, you can redefine them as follows:

To begin, start AutoCAD and create thirteen separate drawing files named HMI_CKV.DWG, HMI_CSRC.DWG, HMI_FARW.DWG, HMI_PS.DWG, JUNCTION.DWG, TANK.DWG, RESERVOIR.DWG, PUMP.DWG, PRV.DWG, PBV.DWG, PSV.DWG, FCV.DWG and TCV.DWG. Save these drawings in your AutoCAD directory.

Open the existing drawing that contains the network blocks.

1. At the AutoCAD "Command:" prompt, type INSERT and press enter.
2. At the "Block Name:" prompt, type JUNCTION=C:JUNCTION.DWG and press enter.
3. At this point, the block has been redefined and you can cancel this command.
4. Repeat these steps for the other named blocks.

Refer to your AutoCAD documentation for more information on Redefining Blocks.

14.2.4 Advanced DXF Import Techniques

To import the network DXF file into an existing AutoCAD drawing file, you will have to perform a couple of preliminary steps:

1. In your existing drawing, at the AutoCAD "Command:" prompt, type (regapp "WTRC") and press Enter. This will register the program application ID. Be sure to include the parenthesis.
2. Define blocks named HMI_CKV, HMI_CSRC, HMI_FARW, HMI_PS, JUNCTION, TANK, RESERVOIR, PUMP, PRV, PBV, PSV, FCV and TCV.

You are now ready to import a WaterCAD DXF file into your existing AutoCAD drawing.

> **To save time, you can perform the above steps in a new AutoCAD drawing file and save it with the name WaterCAD.DWG. Now, instead of performing the above steps, simply insert this new drawing into your existing drawing file immediately before importing a network DXF file.**

> **Refer to your AutoCAD documentation for more information on Importing DXF files.**

Notes

Chapter 15

Additional Features of the AutoCAD Version

WaterCAD features optional support for AutoCAD R14 and AutoCAD 2000 integration. You can determine if you have purchased AutoCAD functionality for your WaterCAD 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 WaterCAD application in both AutoCAD and Stand-Alone mode.

The AutoCAD functionality has been implemented so as to be essentially identical to that offered with the Stand-Alone base product. Once you obtain familiarity with the Stand-Alone mode, you will not have any difficulty utilizing the product in AutoCAD mode.

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 WaterCAD client layer that allows you to create, view, and edit the native WaterCAD network model while in AutoCAD.

Some of the advantages of working in AutoCAD mode include:

- Layout network pipes 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 WaterCAD elements with respect to other entities in the AutoCAD drawing.
- Use native AutoCAD commands such as ERASE, MOVE, and ROTATE on WaterCAD model entities with automatic update and synchronization with the model database.
- Output contours to your AutoCAD drawing.
- Control destination layers for model elements and associated label text and annotation, giving you control over styles, linetypes, and visibility of model elements.

15.1 AutoCAD Environment

15.1.1 AutoCAD Mode Graphical Layout

In AutoCAD mode, Haestad products provide a set of extended options and functionality over that available in Stand-Alone mode. This additional functionality is provided to complement the advanced power available in the AutoCAD environment. Generally, it provides enhanced user control over general application settings and options and extends the command set, allowing user modification of the presentation of Haestad Methods software model elements within AutoCAD.

Key differences between AutoCAD and Stand-Alone mode include:

- Element Editing functionality has been extended by adding the **Scale Elements** and **Rotate Labels** commands, accessible under the **Edit\Modify Elements** pull-down menu, and the **Change Widths** under the **Edit\Modify Pipes** pull-down menu.
- You can control the appearance and destination of all model elements using the **Element Properties** command under the **Tools** pull-down menu.
- Though right-click context menus are now standard with AutoCAD 2000, a **Right-Click Context Menu Option** has been added to provide optional conformity with the Stand-Alone mode of operation in AutoCAD R14.

15.1.2 Toolbars

In AutoCAD mode, this command toggles the display of the program toolbars. The following toolbars are available:

- **Command Tools –** Enables the Commands Toolbar for quick access to the main commands, including computations, table and graphic reports, Quick View, and direct access to the Haestad Methods Web Site.
- **Layout Tools –** Enables the Layout Toolbar for access to the Tool Palette.
- **Analysis Toolbar** – Enables the Analysis Toolbar in order to display the current scenario and provide quick access to the Scenario Manager and Cost Manager. It also displays the current time step and time step viewing controls (WaterCAD Only).

To toggle the display of the Haestad toolbars, select **Toolbars** from the View menu.

15.1.3 Drawing Setup

When working in the AutoCAD mode, you may work with Haestad Methods' products in many different AutoCAD scales and settings. However, Haestad Methods' product elements can only be created and edited in model space.

15.1.4 Symbol Visibility

In AutoCAD mode, you can control display of element labels using the checkbox in the **Drawing Options** dialog.

The following commands allow you to customize the drawing by turning the visibility of flow arrows and labels on or off:

- To turn on the element labels, type: **WTRCLABELSON**
- To turn them off, type: **WTRCLABELSOFF**

> **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.**
>
> **See Rebuild Figure Labels for more information on restoring labels that have been erased using the native AutoCAD command.**

15.1.5 Rebuild Figure Labels

When running WaterCAD in the AutoCAD mode, it is possible to delete associated element label text entities. Element labels which have been erased can be selectively undeleted using the command **WTRCREBUILDLABELS**.

15.2 AutoCAD Project Files

When using WaterCAD in AutoCAD mode, there are two files that fundamentally define a WaterCAD 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 (.WCD)** – The native WaterCAD model database file that contains all the element properties, along with other important model data. WaterCAD .WCD files can be loaded and run using the Stand-alone mode. These files may be copied and sent to other WaterCAD users who are interested in running your project. This is the most important file for the WaterCAD model.

The two files will have the same base name. It is important to understand that simply *archiving the drawing file is not sufficient to reproduce the model.* You must also preserve the associated .WCD file.

Since the .WCD file can be run and modified separately from the .DWG file using Stand-Alone mode, it is quite possible for the two files to get out of sync. Should you ever modify the model in Stand-Alone mode and then later load the AutoCAD .DWG file, the WaterCAD program will compare file dates, and automatically invoke its built-in AutoCAD Synchronization routine.

15.2.1 Drawing Synchronization

Whenever you open a WaterCAD-based drawing file in AutoCAD, the WaterCAD model server will start. The first thing that the application will do is load the associated WaterCAD database (.WCD) file. If the time stamps of the drawing and database file are different, WaterCAD will automatically invoke its synchronization check. This protects against corruption that might otherwise occur from separately editing the WaterCAD database file in Stand-Alone mode, or editing proxy elements at an AutoCAD station where the WaterCAD application is not loaded.

The synchronization check will occur in two stages:

- First, WaterCAD will iterate across all the drawing model elements and compare their state with that held in the server model. Any differences it discovers will be listed. WaterCAD enforces network topological consistency between the server and the drawing state. If model elements have been deleted or added in the .WCD file during a stand-alone session, or if proxy elements have been deleted, the application 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, the application will compare other model and drawing states such as location, labels, and flow directions. Again, it will list any differences between the drawing client and server data, and a message box will pop up giving you an opportunity to indicate which state, drawing or model server, should be adopted during the second stage of synchronization.

You can run the Synchronization check at any time using the command **WTRCSYNCSERVER**.

15.2.2 Saving the Drawing as Drawing*.dwg

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 or not one exists in the default directory. Since Haestad Methods' modeling products create model databases associated with the AutoCAD drawing, the use of Drawing*.dwg as the saved name puts you at risk of getting the AutoCAD drawing and Haestad Methods modeling files out of sync.

If this situation is forced to occur (save on quit for example), simply 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.

15.3 WaterCAD Element Properties

When working in the AutoCAD mode, this feature will display a tabbed dialog with grids containing different model element types and their associated properties, along with the properties of the element's label and annotation. To modify an attribute, double-click each associated grid cell. Setting changes made in this dialog will be used for any newly created elements. If the **Apply to Existing Figures** box is checked, modifications made in this dialog are performed on a global basis. Property changes will be performed on all elements of the given type. To restrict global changes to a certain layer for a particular element type, use the "*current*" option setting for the attribute of interest.

To change the layer of an element, select **Tools\Element Properties** from the pull-down menu.

15.3.1 Select Layer

When running in AutoCAD mode, this dialog appears when you double-click the layer name ("*current*" by default) in the **Layer** column of the **Element Properties** dialog. This is accessed by selecting **Tools\Element Properties** from the pull-down menu. It displays a list of the available layers and their properties from the current AutoCAD drawing. Click the appropriate field to select a layer. The "*current*" option will use whatever layer is set to current in your AutoCAD drawing.

15.3.2 Select Text Style

When running in AutoCAD mode, this dialog appears when you double-click the text style name ("*current*" by default) in the **Text Style** column of the **Labels** and **Annotation** tabs of the **Element Properties** dialog. This is accessed by selecting **Tools\Element Properties** from the pull-down menu. It displays a list of the available text styles and their properties from the current AutoCAD drawing. Click the appropriate field to select a text style. The "*current*" option will use whatever text style is set to current in your AutoCAD drawing.

15.4 Working with Elements

15.4.1 Edit Element

In AutoCAD mode, this menu selection will open an element editor for any specific element. Select **Edit\Edit Element**, then select an element, or type "L" at the command line for a list of elements to select from. This command is also available by choosing the Select tool, then right-clicking the drawing pane.

The Edit Element command works with the current selection to allow you to generate filtered reports. Refer to Selecting Elements (AutoCAD Mode) for more information on working with selections.

15.4.2 Deleting Elements

In AutoCAD mode, this command removes all elements in the current selection. Refer to Selecting Elements (AutoCAD Mode) for more information on working with selections.

15.4.3 Modifying Elements

In AutoCAD mode, these commands are selected from the **Edit** pull-down menu. They are used for scaling and rotating model entities.

Scale Elements

In AutoCAD mode, this menu selection resizes an element based upon a scale factor. After choosing this command, select an element or group of elements, and enter the scale factor to be applied.

To access the scale elements command, select **Modify Elements** from the **Edit** pull-down menu.

Rotate Labels

In AutoCAD mode, this menu selection rotates the figure label. After choosing this command, select an element or group of elements, and enter the desired rotation in degrees.

Modify Pressure Pipes

Pipes may follow a non-linear alignment, since in pressure systems minor losses can be safely lumped with friction losses without significantly affecting model accuracy. WaterCAD uses the following specialized commands for editing pipes in AutoCAD:

- **Insert Bend** – Use this command to add a bend to a pipe. In AutoCAD, you will be prompted to select a pipe to bend. Select the pipe and the location you want the bend to appear. The pipe alignment will automatically conform to this location.
- **Remove Bend** – Use this command to remove a specific bend from a pipe. In AutoCAD, you will be prompted to select a pipe and the specific bend to remove.
- **Remove All Bends** – Use this command to completely straighten a pipe that contains bends. In AutoCAD, you will be prompted to select a pipe, and all bends will disappear.
- **Change Widths –** Use this command to change pipe widths. After choosing this command, select a pipe or group of pipes and enter the desired width. Note that the width entered is equivalent to the AutoCAD polyline width.

15.5 Working with Elements Using AutoCAD Commands

15.5.1 WaterCAD Custom AutoCAD Entities

The primary AutoCAD-based WaterCAD element entities – pipes, tanks, reservoirs, pumps and valves – 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 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 enables the implementation of highly specialized editing actions that are not available with standard AutoCAD entities. Two examples of this specialized behavior are element morphs and pipe splits. Again, these modifications will trigger an automatic update of the model network topology and associated element properties.

Using ObjectARX technology ensures the database will be adjusted and maintained during Undo and Redo transactions.

A custom model element has certain native text entities associated with it for displaying label and annotated property values. These associated label and annotation entities may be edited separately from the model element itself. However, most drawing edits made directly to a model element will be applied in the appropriate fashion against its associated label and annotation entities. Thus, if you drag an element to a new location, the annotation and label locations will update as well.

15.5.2 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 normally. For example, Haestad Methods' elements and annotation can be manipulated using common AutoCAD commands.

15.5.3 Explode Elements

When using WaterCAD in AutoCAD mode, running the AutoCAD explode command will transform all Haestad custom entities into equivalent AutoCAD native entities. When a Haestad 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 Haestad Product license, since a fully exploded drawing will not be comprised of any ObjectARX proxy objects. See Working with Proxies for more information on this topic.

15.5.4 Moving Elements

When using WaterCAD in AutoCAD mode, the AutoCAD commands Move, Scale, Rotate, Mirror and Array can be used to move elements. Refer to Selecting Elements for more details on this topic.

To move a node, execute the AutoCAD command either by typing it at the command prompt or selecting it from the pull-down menu. 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. This is the only way to move pipes.

15.5.5 Moving Element Labels

When using WaterCAD in AutoCAD mode, the AutoCAD commands Move, Scale, Rotate, Mirror and Array can be used to move element text labels. Refer to the help topics Selecting Elements and Working with Selections in AutoCAD.

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 or selecting it off the tool palette. Follow the AutoCAD prompts, and the label will be moved without the element.

15.5.6 Snap Menu

When using WaterCAD in AutoCAD mode, the **Snap** menu is a standard AutoCAD menu that provides options for picking an exact location on an object. Refer to the standard AutoCAD help system for more information.

15.6 Undo / Redo

In AutoCAD mode, you have two types of Undo/Redo available to you. From the **Edit** menu, you have access to WaterCAD 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 WaterCAD 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.

> **If you use the native AutoCAD undo, you are limited to a single redo level. The WaterCAD undo/redo is also faster than the native undo/redo. If you are rolling back WaterCAD model edits, it is recommended that you use the menu-based undo/redo implementation.**

Whenever you invoke a native AutoCAD undo, the server model will be notified when any WaterCAD entities are affected by the operation. WaterCAD will then synchronize the model to the drawing state. Wherever possible, the model will seek to map the undo/redo invocation 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, WaterCAD will flush the command history. In this case, the model will synchronize the drawing and server models.

> **If you undo using the AutoCAD command and you end up restoring WaterCAD 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 WaterCAD command history has been flushed. In such cases, you will be warned to check your data carefully.**

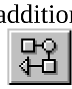
15.7 Converting Native AutoCAD Entities to WaterCAD Elements

15.7.1 Converting Native AutoCAD Entities

WaterCAD features powerful tools dedicated to assisting you in building WaterCAD models from existing AutoCAD drawing information. In addition to the standard GIS shapefile conversion options, there are two specific commands available in the AutoCAD platform that will be especially useful to the AutoCAD modeler:

- Layout Pipe Using Entity
- Change Entities to Pipe

15.7.2 Layout Pipe Using Entity

In addition to the standard options available under the Pipe layout command (accessed by clicking the  button in the WaterCAD Tools toolbar, or by selecting the **Tools\Layout\Pipe** menu option), you may elect to use an existing AutoCAD line, polyline, or arc as a template to define an equivalent WaterCAD pipe or series of pipes.

While you are in the **Pipe Layout** command, you may invoke the Entity conversion option by using the **'Entity'** keyword, or by selecting **'Entity'** from the right-mouse button context menu. Once selected, you will be prompted to choose an entity to use as a basis for a new pipe, and conditionally specify the type of nodal WaterCAD element(s) to use at each end of the pipe.

This command is extremely useful for constructing pipes that follow a curved alignment. In these cases, use an arc as the defining template entity for the pipe creation.

15.7.3 Change AutoCAD Entities to Pipes

When running WaterCAD in AutoCAD mode, this special AutoCAD command allows you to use a selection of AutoCAD entities – arcs, lines, polylines, and blocks – as a defining template set for the creation of equivalent WaterCAD elements. This command performs the element generation in batch fashion. You are prompted for the selection of entities to convert, and the selection is followed by the Polyline to Pipe Conversion Wizard that leads you through a sequence of steps defining the basis of the batch conversion. The actual steps to be followed in the Wizard are fully described in the AutoCAD Polyline to Pipe Conversion topic.

This is an automated batch process that requires some care and attention with respect to the selection set that is going to be used as a basis for generating actual WaterCAD model elements. For instance, it may be desirable to select like-sized pipe elements during each pass. This way, you can use the prototyping capabilities to their greatest advantage. A little time spent in planning and strategizing a series of individual conversion steps will go a long way toward preventing confusion, which could necessitate later re-conversions.

The **Change Entities to Pipes** command is accessed from the **Edit\Change Entities to Pipes** pull-down menu.

15.8 Special Considerations

15.8.1 Import WaterCAD

When running WaterCAD in AutoCAD mode, this command imports a selected WaterCAD data (.WCD) file for use in the current drawing. The new project file will now correspond to the drawing name, i.e. CurrentDrawingName.WCD. Whenever you save changes to the network model through WaterCAD, the associated .WCD data file is updated and can be loaded into WaterCAD 4.0 or higher.

To import a WaterCAD model into AutoCAD, select **File\Import\WaterCAD**.

15.8.2 Working with Proxies

If you open a WaterCAD drawing file on an AutoCAD workstation that does not have the WaterCAD 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 WaterCAD modeling elements are not associated with the WaterCAD native database.

WaterCAD 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 WaterCAD application, the application will automatically attempt to reconcile any differences it finds by automatically loading its Database Synchronization routine.

Notes

Appendix A

Frequently Asked Questions

Tips are available in the following categories:

- Importing/Exporting
- Modeling
- Display
- Editing

Extensive, up-to-date tips are available at your finger tips by clicking the button. This will take you to Haestad Methods web site, where you will be able to look up Frequently Asked Questions (FAQs) in our KnowledgeBase and do a search on any keyword(s). This is available if you are participating in the ClientCare program.

If the information you need is not available in this section, click the Search tab at the top of the Help window for an index. To make your work easier, WaterCAD and the Help system are designed to be used together. If you have a high resolution display monitor, you will probably find it helpful to size the frames of both the program and the Help windows so that they fit side by side. Then, while using the program, you can use the right mouse button or the Help button in any dialog box to update the Help window with context-sensitive Help.

A.1 Import/Export Tips

The following tips will be covered in this section:

- Importing Data from Previous WaterCAD/Cybernet Versions
- Transitioning from Cybernet v2
- Importing EPANET Data
- Importing KYPIPE Data
- Importing Spot Elevations
- Exporting Spot Elevations

You can import data from virtually any model using our intuitive and powerful Database and Shapefile Connections feature.

A.1.1 Importing Data from Previous WaterCAD/Cybernet Versions

Cybernet v1 drawing

No support is available for importing a Cybernet v1 drawing directly into WaterCAD v4. If you want to import a Cybernet v1 drawing, load it into Cybernet v2 (for DOS) and re-save the data as a Cybernet v2 drawing. The data can then be imported into WaterCAD v4 following the procedure described below.

Cybernet v2 drawing

If you are running WaterCAD v4 in AutoCAD mode, simply open the drawing that contains the Cybernet v2 data. The import Wizard of WaterCAD v4 will automatically begin importing the data.

If you are running WaterCAD v4 in Stand Alone mode, you must first convert the Cybernet v2 drawing into a special C2W file format that can then be imported into WaterCAD v4. This is accomplished by using one of the C2W utilities inside any of the following AutoCAD versions: AutoCAD 12, 13 Dos, 13 Windows, 14, or 2000. Consult the C2W utilities Help for specifics on its use.

Because the color coding legend is comprised of native entities, the built-in conversion is unable to automatically remove legend elements. In WaterCAD v3.0 and up in AutoCAD mode, color coding legends are block inserts. If your Cybernet v2 drawing contains legends, you must manually edit these out of the drawing.

When using the C2W utility in AutoCAD 12 or 13, there are certain limitations to the amount of data imported:

- **Only Cybernet v2 base data is imported. No change records in Cybernet scenarios are supported.**
- **Cybernet multi-point pump curves are not supported. The C2W utility converts the pump to a standard three-point curve pump. This may not be the optimum solution for the multi-point curve. Validate the points chosen, and consider manually entering the multi-point data.**
- **Cybernet demand patterns are not imported. However, junction node demand type data is imported. Use the Pattern Manager to enter the Cybernet Extended Period Simulation GDF curves.**

WaterCAD/Cybernet v3 Files

Simply open the WaterCAD v3 file, or import the .WCD file as you normally would in WaterCAD/Cybernet v3. The .WCD file will be automatically converted to the WaterCAD v4 format. Once you save this file in WaterCAD v4, it can no longer be opened in WaterCAD/Cybernet v3.

A.1.2 Transitioning from Cybernet v2

This section is intended to offer you some insight about the tools in this version of WaterCAD that are different from Cybernet v2 methods.

We have seen the questions that arise in technical support. We designed WaterCAD v3 and v4 specifically to help our users avoid many of these problems, while offering even more flexibility and a much friendlier interface.

Working with the Graphical Editor

One of the first differences is the interface, aside from just the difference of it being a Windows application. WaterCAD v4 actually has two interfaces, one for AutoCAD (called WaterCAD's AutoCAD mode), and our own CAD-style stand-alone interface (called WaterCAD's Stand-Alone Mode).

This offers an amazing amount of flexibility, especially since both interfaces can be used with the exact same hydraulic model. Organizations now have the flexibility to allow AutoCAD users and non-AutoCAD users to work with the same model, without struggling through any type of intermediate conversion. Even the style of the two interfaces is similar with identical toolbars and menus.

Where is the Modeling Control Center (MCC)?

The Modeling Control Center in Cybernet v2 served two purposes: tabular reporting and scenario management (including calculation). In WaterCAD v4, these same purposes are served by two separate objects, which are each orders of magnitude better than anything that has come before: FlexTables and Scenario Management.

Report Tables (Flex Tables)

In Cybernet v2, you were provided with a tabular view of the network that was a bit inflexible and tedious at times. WaterCAD v4's FlexTables provide tabular reporting tools that are so powerful and flexible that you can perform your typical tasks in less time than it used to take you just to enter the MCC.

With features like sorting, filtering, and global editing, you can review and adjust your data in a fraction of the time it used to take, and with none of the hassle. Even the variables and sequences that are presented in tabular form are totally customizable to fit your needs.

Scenario Management

Cybernet v2 had two levels of definition: the base data as entered in the AutoCAD interface or through the Edit menu in the MCC, and scenario changes as entered through the Setup Analysis menu. Although this was far better than any competing scenario management, it was very limited and certainly not oriented toward data-reuse.

WaterCAD v4 has a whole new outlook on scenarios, stemming from the basic principle that every system has unknowns. If there is only one unknown, such as junction demand, for example, the options are quite simple. With the addition of more unknowns, the options quickly become difficult to manage. How can a modeler keep track of so many unknowns? Through alternatives and inheritance.

Alternatives are collections of very specific data, such as junction demands, or pump and valve operational settings. A scenario references a certain combination of these alternatives, similar to a slot machine rolling different symbols in and out of each position. Rather than several similar scenarios each holding onto individually adjusted data, they can instead each reference some of the same alternatives (just as several slot machine combinations can show the same symbol). This not only allows for far more flexibility, it also greatly reduces the amount of data that is handled, and improves accuracy.

Inheritance is another attribute of WaterCAD v4's scenario management that adds a level of functionality that has never existed before. In Cybernet v2, each scenario essentially "inherited" data from the base model, unless a specific change was made. As mentioned above, this is fine for individual changes, but it falls apart for subsequent changes. Consider, for example, a system that is to have pipes replaced in phases. Phase I changes inherit from the base model, but then what happens for Phase II? Repeat all of the changes from Phase I, and add the changes for Phase II? And then again for Phase III? It is too easy to overlook a change and make simple typographical errors.

In WaterCAD v4, there is no limit to the extent of inheritance that could exist. To follow the example given above, Phase II simply inherits its data from Phase I, and then has whatever changes are specific to Phase II rehabilitation. Likewise, Phase III inherits from Phase II, and so on. Best of all, if something changes in Phase I, that change is inherited through the hierarchy such that all of the children (Phase II, Phase III, etc.) reflect the new data.

There is a scenario tutorial for the stand-alone editor, and there is a scenario Wizard to help you through your first few scenario creations. Once you have seen what alternatives and inheritance can do for your model, you will be glad that you spent a few minutes getting familiar with them.

Using the Scenario Manager

Scenario functionality is extended even further by the presence of comparative tools, such as graphing. Scenario results can be directly compared graphically, plotting all of the scenarios on the same axes. This means that determining the effects of things such as system expansion, future demand increases, and pipe deterioration can all be seen within seconds of running the models.

Combined with batch runs, which is running several scenarios in sequence, scenario management has reached a level that most modelers have only dreamed of until now.

Demand Alternatives

With the new scenario management, demand loading also has all new flexibility. Rather than being limited to Avg. Day, Max Day, Peak Hr., User 1 and User 2, there are an unlimited number of demand alternatives available in WaterCAD v4. You can still have global demand and global roughness factors for your WaterCAD v4 model, so you can make minor adjustments during calibration without having to generate new alternatives.

Composite Demands

As well as having an unlimited number of demand alternatives, there is also no limit to the number of different demand types you can have. Cybernet v2 was limited to five demand types (1, 2, 3, 4, and Fixed), but WaterCAD v4 allows as many different types necessary. Also, demands can have alpha-numeric labels, such as "Residential", "Commercial", and "Industry XYZ." This enables you to model service connections with much more detail, because you can specify diurnal demand patterns for any number of special individual customers.

In addition to demands, WaterCAD v4 also provides inflow, or flows that are introduced into the system.

Perhaps the best improvement for demands, however, is the ability to attribute more than one type of demand to a given junction node. An unlimited number of different customer types can all contribute to the total demand at a single junction, so there is no need to estimate an equivalent demand type or add another demand type to a "fake" junction immediately adjacent to another node.

Links to Graphical Information Systems (GIS) and Databases

One of the most exciting developments of the software industry in the 1990's has been the push for data reuse. Constantly improving database capabilities, published file formats (such as the Shapefile format), and even tools as simple as the Windows clipboard are all contributing to a level of data sharing that has never been seen before. WaterCAD v4 is right there on the leading edge.

With WaterCAD v4's new database connectivity, your hydraulic model can easily be linked to virtually any major database, spreadsheet, or GIS product currently in use today. Shapefile wizards and flexible database linking tools make the process simple and straightforward, without anchoring you to a specific database layout or units system.

Control Valves

Cybernet v2 provided you with three types of controlling valves: Pressure Reducing Valves (PRV's), Pressure Sustaining Valves (PSV's), and Flow Control Valves (FCV's).

WaterCAD v4 expands the options to include two additional types: Pressure Breaker Valves (PBV's) that create a constant headloss across the valve, and Throttle Control Valves (TCV's) that allow you to adjust minor loss coefficients based on system pressures, HGL's, or time.

Cybernet v2 also allowed a valve setting of "Maintain Always" for PRV's and FCV's. These settings were primarily used to simulate a pump for preliminary design work. Through numerous support calls and dozens of inaccurate models, however, we found that this feature was often misused, resulting in frustration. These "Maintain Always" settings are no longer supported in WaterCAD v4. Instead, we offer a wider variety of pump options to encourage modelers to make better educated guesses and better preliminary design decisions.

Pumps and Pump Curves

Cybernet v2 pumps can be categorized as one of three types: constant horsepower, three-point, and multi-point (up to eleven points).

In WaterCAD v4, three-point pump curves are still fully supported, as are multi-point pump curves. In fact, there is no limit to the number of points you can enter to approximate the pump's exponential curve.

Although we continue to discourage the use of constant horsepower pumps (for many of the same reasons we discouraged the use of "Maintain Always" setting for valves), this type of pump is still available in WaterCAD v4. However, do not use one unless you have actually looked at a constant horsepower curve. It only resembles the shape of a typical pump curve over a very short range near the best efficiency point, and diverges from this curve rapidly as the curve becomes asymptotic to both the head and discharge axes.

If you are performing a preliminary design or if you have another purpose that requires you to estimate pump characteristics based on insufficient data, consider using a one-point pump curve. This allows you to enter the design point and approximate a curve based on a typical pump curve. Of course, nothing beats having actual pump test data so you can generate a truly accurate representation of your pump, and, subsequently, an accurate representation of the remainder of your system.

Conclusion

WaterCAD v4 has many more features and enhancements, found in every dialog and button. The following are a two very important points that we would like to emphasize as you prepare to use WaterCAD v4 for the first time:

- Tutorials are available in the Stand-Alone editor for a deeper introduction to nearly every topic, and there is context-sensitive on-line Help available from anywhere in the program by pressing the **F1** key, or by clicking a **Help** button.
- Don't be afraid to explore. Some of the best features can be easy to overlook. Remember, whenever you see an **ellipsis (…)** button it means that a special feature is available. Play with the model, and most importantly, start to enjoy modeling again.

A.1.3 Importing EPANET Files

From the pull-down menu, select **File\Import\Network** and choose EPANet (inp). Then, from the **File\Open** dialog, select the file you would like to import. During the import procedure, you will be prompted for a map scale factor. You may also be asked to specify the Unit of Concentration.

In EPANET, pumps and valves are modeled as links. In this program, they are graphically modeled as node elements. Hence, during an import, each EPANET valve and pump link is replaced by two pipes and one pump or valve element. This will not affect the behavior of these elements in your system.

In EPANET, tanks can have an optional inactive volume parameter. If this parameter is omitted from the input file or a zero is entered for this parameter, the EPANET numerical engine will compute an inactive volume based on the tank's diameter and the minimum level. To mimic this behavior, this software will calculate the inactive volume and display it in the tank data upon import of the file. When using this software, it is important to remember that zero inactive volume means zero inactive volume, and not some internally computed value.

A.1.4 Importing KYpipe Data

This program supports the import of KYpipe 1.0, 2.0 and 3.0 data sets. If the data set does not include geometry data, all nodes will be assigned a coordinate of (0,0). This has no effect on the hydraulic state of the model. Pipe lengths will not be computed based on the coordinates of the end nodes, but will be taken directly from the KYpipe data set.

This program only supports the import of the pipes and nodes of a KYpipe model. You must insert pumps, valves, and tanks into the current project.

A.1.5 Importing Spot Elevations

A series of spot elevations can be imported from an ASCII text file, which might be generated from a survey data recorder or another software program. These ASCII files can contain a combination of the information that is required for spot elevations, such as the label, coordinates, and elevation. The fields in the text file are usually separated by either blank spaces or commas.

A.1.6 Exporting Spot Elevations

All of the spot elevations in the current project can be exported to an ASCII text file, from which they can be brought into a spreadsheet, word processor, or other program. These ASCII files can contain a combination of the information that is required for spot elevations, such as the label, coordinates, and elevation. The fields in the text file are usually separated by either blank spaces or commas.

A.1.7 Importing Database and Shapefile data created with WaterCAD v3

As a result of overwhelming user feedback, as well as through a review of common technical support questions, Haestad Methods has decided to make a fundamental change in the way pump/valve connectivity is modeled. These elements are now handled as nodes, whereas they were previously represented as links in database and GIS connections prior to the release of WaterCAD v4. Unfortunately, this change will impact existing users who have built database connections using WaterCAD Version 3.5 and earlier. However, a survey of our customers has shown that nodes are the natural and preferred way to represent a pump or valve in a database. The change is driven by the desire to improve and enhance the mapping of these elements onto GIS and enterprise data. Haestad Methods has invested significant effort to separate the model representation of pumps and valves from the user view of these elements.

Specific changes are as follows:

- Pumps and valves were formerly to be represented in the model as links, with To and From Node attributes. Now pumps and valves are represented as nodes, with To and From Pipe attributes.
- Pumps and valves exported to the Links Table will not be restored to the model. If the user tries to import a Link Table containing pumps and valves, the items in the table will be created as new pipes of zero length and at coordinates of (0,0).
- The import of pump and valve tables will work as expected, with the exception of the items listed above.

A.2 Modeling Tips

This section presents some FAQ's related to modeling water distribution networks with WaterCAD. Also, please keep in mind that Haestad Methods offers workshops in North America and abroad throughout the year. These workshops cover these modeling topics and others in depth. The following modeling tips are presented in this chapter:

- Modeling a Hydropneumatic Tank
- Modeling a Pumped Groundwater Well
- Parallel Pipes
- Modeling Pumps in Parallel and Series
- Modeling Hydraulically Close Tanks
- Modeling Fire Hydrants
- Modeling a Connection to an Existing Water Main
- Creating a System Head Curve
- Top Feed/Bottom Gravity Discharge Tank
- Variable Speed Pump Maintaining a Constant Downstream Pressure

A.2.1 Modeling a Hydropneumatic Tank

Hydropneumatic tanks can be modeled using a regular tank element and converting the tank pressures into equivalent water surface elevations. Based on the elevation differences, the tank's cross-sectional area can then be determined.

For example, consider a hydropneumatic tank that operates between 50 psig and 60 psig. The tank's storage volume is approximately 50 cubic feet.

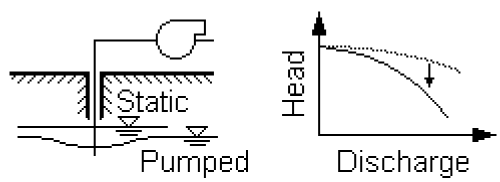
The tank base elevation is chosen to be equal to the ground elevation, and the pressures are converted into feet of water (1 psi = 2.31 feet). It is apparent that the tank operates between levels of 115.5 feet and 138.6 feet. The difference between the levels is 23.1 feet, which brings us to a needed cross-section of 2.16 square feet.

A.2.2 Modeling a Pumped Groundwater Well

A groundwater well is modeled using a combination of a reservoir and a pump. Set the hydraulic grade line of the reservoir at the static groundwater elevation. The hydraulic grade line can be entered on the reservoir tab of the reservoir editor dialog box, or under the Reservoir Surface Elevation column heading in the **Reservoir Report**.

Pump curve data can be entered on the **Pump** tab of the **Pump** editor. The following example will demonstrate how to adjust the manufacturer's pump curve to account for drawdown at higher pumping rates. Drawdown occurs when the well is not able to recharge quickly enough to maintain the static groundwater elevation at high pumping rates.

Pump Curve Accounting for Drawdown



Example:

The pump manufacturer provides the following data in a pump catalog:

Head (ft)	Discharge (gpm)
1260	0
1180	8300
1030	12400

Based on field conditions and test results, the following drawdown data is known:

Drawdown (ft)	Discharge (gpm)
40	8300
72	12400

To account for the drawdown, the pump curves should be offset by the difference between the static and pumped groundwater elevations. Subtract the drawdown amount from the pump head, and use these new values for your pump curve head data.

The following adjusted pump curve data is based on the drawdown and the manufacturers pump data.

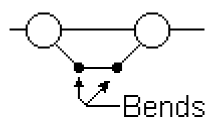
Head (ft)	Discharge (gpm)
1260	0
1140	8300
958	12400

A.2.3 Modeling Parallel Pipes

With some water distribution models, parallel pipes are not allowed. This forces you to create an equivalent pipe with the same characteristics.

With this program, however, you can create parallel pipes simply by drawing the pipes with the same end nodes. To avoid having pipes drawn exactly on top of one another, it is recommended that the pipes have at least one vertex, or bend, inserted into them.

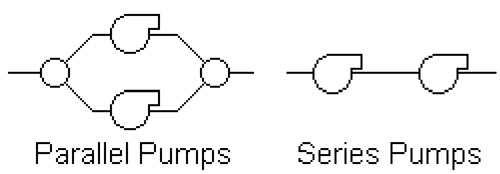
Pipe Bends



A.2.4 Modeling Pumps in Parallel and Series

Parallel pumps can be modeled by inserting a pump on different pipes that have the same From and To Nodes. Pumps in series (one pump discharges directly into another pump's intake) can be modeled by having the pumps located on the same pipe. The following figure illustrates this concept:

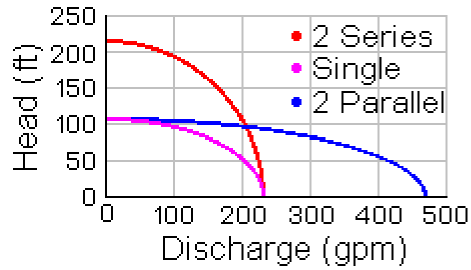
Pumps in Parallel and Series



If the pumps are identical, the system may also be modeled as a single, composite pump that has a characteristic curve equivalent to the two individual pumps. For pumps in parallel, the discharge is multiplied by the number of pumps, and used against the same head value. Two pumps in series result in an effective pump with twice the head at the same discharge.

For example, two pumps that can individually operate at 150 gpm at a head of 80 feet connected in parallel will have a combined discharge of 2•150 = 300 gpm at 80 feet. The same two pumps in series would pump 150 gpm at 2•80 = 160 feet of head. This is illustrated as follows:

Pump Curves of Pumps in Series and Parallel



With pumps in series, it is actually more desirable to use a composite pump than to use multiple pumps in the network. When pumps shut off, it is easier to control one pump. Several pumps in series can even cause disconnections by checking if upstream grades are greater than the downstream grade plus the pump heads.

A.2.5 Modeling Hydraulically Close Tanks

If tanks are hydraulically close, as in the case of several tanks adjacent to each other, it is better to model these tanks as one composite tank with the equivalent total surface area of the individual tanks.

This process can help to avoid fluctuation that may occur in cases where the tanks are modeled individually. This fluctuation is caused by small differences in flow rates to or from the adjacent tanks, which offset the water surface elevations enough over time to become a significant fluctuation. This results in inaccurate hydraulic grades.

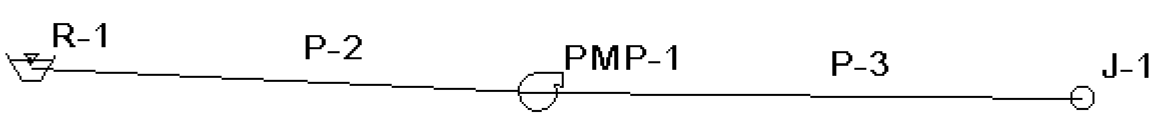
A.2.6 Modeling Fire Hydrants

Fire Hydrant flow can be modeled by using a short, small diameter pipe with a large minor loss, in accordance with the hydrant's manufacturer.

A.2.7 Modeling a Connection to an Existing Water Main

If you are unable to model an existing system back to the source, but would still like to model a connection to this system, a reservoir and a pump with a three-point pump curve may be used instead. This is shown below:

Approximating a Connection to a Water Main with a Pump and a Reservoir



The reservoir simulates the supply of water from the system. The elevation of the reservoir should be equal to the elevation at the connection point.

The pump and the pump curve will simulate the pressure drops and the available flow from the existing water system. The points for the pump curve are generated using a mathematical formula (given below), and data from a fire flow test. The pipe should be smooth, short and wide. For example, a roughness of 140, length of 1 foot, and diameter of 48 inches are appropriate numbers.

Please note that it is ALWAYS best to model the entire system back to the source. This method is only an approximation, and may not represent the water system under all flow conditions.

Qr = Qf * [(Hr/Hf)^.54]

Where:			
	Qr	=	Flow available at the desired fire flow residual pressure.
	Qf	=	Flow during test.
	Hr	=	Pressure drop to desired residual pressure (Static Pressure minus Residual Pressure)
	Hf	=	Pressure drop during test (Static Pressure minus Design Pressure)

Example: Determining the Three-Point Pump Curve

First point:

This point is generated by measuring the static pressure at the hydrant when the flow (Q) is equal to zero.

Q = 0 gpm
H = 90 psi or 207.9 feet of head (found from 90 * 2.31)

(2.31 is the conversion factor used to convert psi to feet of head).

Second point:

The engineer chooses a pressure for this point, and the flow is calculated using the formula below. The value for Q should lie somewhere between the data collected from the test.

Q = ?
H = 55 psi or 127.05 feet (found from 55 * 2.31) (chosen value)

Formula:

Qr = Qf * (Hr/Hf)^.54
Qr = 800 * [((90 – 55) / (90 – 22))^.54]
Qr = 800 * [(35 / 68)^.54]
Qr = 800 * [.514^.54]
Qr = 800 * .69
Qr = 558

Therefore,

Q = 558 gpm

Third point:

This point is generated by measuring the flow (Q) at the residual pressure of the hydrant.

Q = 800 gpm
H = 22 psi or 50.82 ft. of head (from 22 * 2.31)

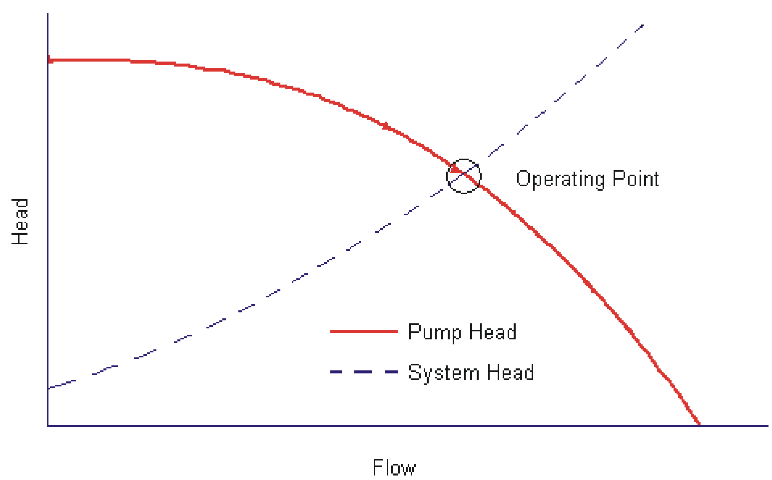
Pump curve values for this example:

Head (ft)	Discharge (gpm)
207.90	0
127.05	558
50.82	800

A.2.8 Creating a System Head Curve

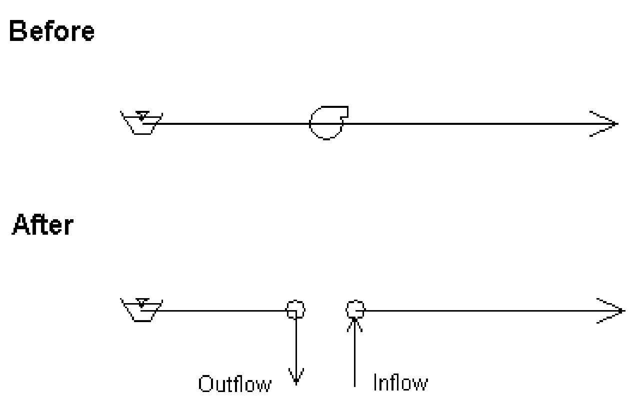
A system head curve is a graph of the increase in head across a system as flow increases. The curve can be used to find an appropriate sized pump for a system, or to find a pump's operating point. The operating point is at the intersection of the system head and pump head curves.

Pump and System Head Curve



To create a system head curve, you must first replace your pump with two nodes with a gap between them, as shown below.

Model Modification for Creating a System Curve



Next, apply a demand to the node that would be the suction side of the pump and an inflow to the node that would be the discharge side of the pump. An inflow can be applied to a node by changing the demand in the **Type** column of the Demand table to inflow. Now run the model.

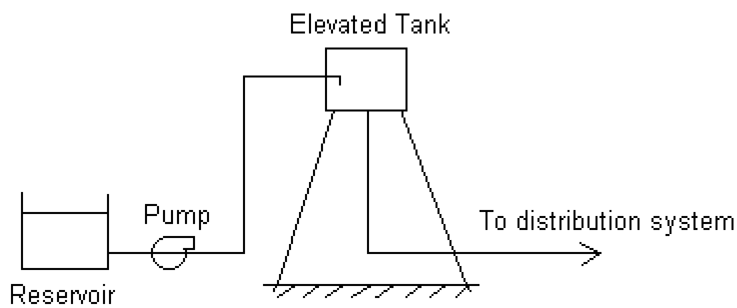
After running the model, click the suction side node and record the Hydraulic Grade. Then, click the discharge side node and record the Hydraulic Grade. The difference between the two is the change in head at that flow.

Repeat the above steps to find the change in head for several different flows. Then, graph all of the points to create the system head curve.

A.2.9 Top Feed/Bottom Gravity Discharge Tank

A tank element in WaterCAD is modeled as a bottom feed tank. Some tanks, however, are fed from the top, which is different hydraulically and should be modeled as such.

Top Feed/Bottom Gravity Tank



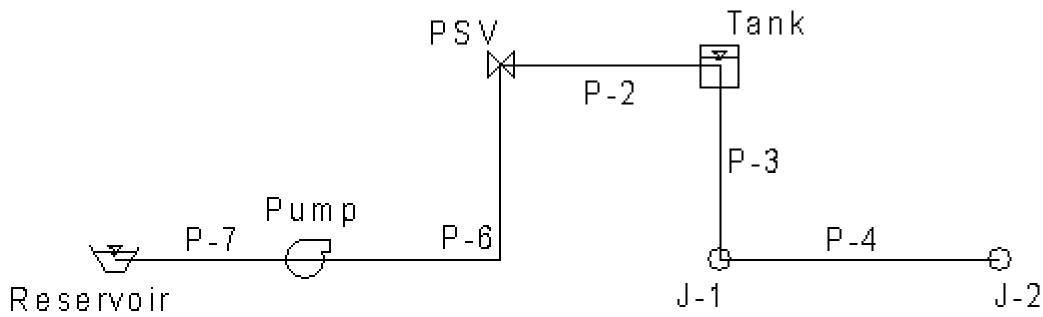
To model a top feed tank, start by placing a pressure sustaining valve (PSV) at the end of the tank inlet pipe. Set the elevation of the PSV to the elevation of the tank inlet. The pressure setting of the PSV should be set to zero to simulate the pressure at the outfall of the pipe.

Next, connect the downstream end of the PSV to the tank with a short, smooth, large diameter pipe. The pipe must have these properties so that the headloss through it will be minimal.

The tank attributes can be entered normally using the actual diameter and water elevations.

The outlet of the tank can then proceed to the distribution system.

Example Layout



A.2.10 Pump Maintaining a Constant Downstream Pressure

To model a variable speed pump that is set to maintain a constant downstream pressure you will want to use a multiple point pump curve. Enter the points so that the maximum pump head is the downstream pressure to be maintained and the curve remains relatively flat until the pumps maximum discharge. Then the curve should taper down to its maximum flow rate. With this curve the pump will discharge flow at the same pressure up until its maximum speed, when its discharge head will decrease with any increase in discharge.

Alternatively, to model the effect of a variable speed pump you can model a pump with artificially high design, shutoff, and max operating heads. Then let a PRV downstream of the pump regulate the head down to the setting of the variable speed pump.

Another method is to regulate the discharge head of the pump by varying the pumps relative speed factors by using pump controls in concordance with the Affinity laws.

A.3 Display Tips

The following display tips will be discussed in this section:

- Changing Units in a Column
- Controlling Element and Label Sizing
- Color Coding Elements
- Reusing Deleted Element Labels

A.3.1 Changing Units in a Column

In a Table you may change the units of all the data within any column. To change the units:

3. Select **Use Local Units** from the **Options** menu in the **Tabular Report** dialog.
4. Right-click the column heading, or any data item within a column.
5. Select the **Properties** option from the pop-up menu.

6. Change the units and select **OK**. All data items in that column will change to the selected units.

> **The change of units affects only the data in the Table. It DOES NOT change the units within your network design.**

A.3.2 Controlling Element and Label Sizing

To change the size of element symbols and labels:

7. Select **Tools\Options** from the pull-down menu and select the **Drawing** tab.

8. In the **Annotation Multipliers** group, change the **Symbol Size Multiplier** to modify the element size, and the **Text Height Multiplier** to modify the label size. Smaller numbers will make the element symbols and text decrease in size.

These changes will affect all symbols and text, including color coding legends, but will not have any effect on pipe lengths.

A.3.3 Color Coding Elements

To color code the elements:

1. Select **Tools\Color Coding...** from the pull-down menu, or click the **Color Coding** button on the toolbar.

2. In the **Color Coding** dialog, select the attribute you would like to color code.

3. Click the **Initialize** button to automatically build a range of colors. You may decide to modify these default ranges.

4. Click **OK** to color code the drawing.

All link or node elements and their labels will be colored based on the specified ranges. You can also use the **Color Code** button to quickly set up and modify Color Coding Options. A Color Coding Legend may be inserted into the drawing by using the **Legend** tool located on the Tool Palette.

A.3.4 Reusing Deleted Element Labels

To make the program reuse the label for a deleted element:

1. Select **Tools\Element Labeling** from the pull-down menu.

2. Enter the ID number for the deleted element in the **Next** field for the appropriate type of element.

3. Click **OK**.

4. Add a new element in the drawing editor.

A.4 Editing Tips

The following tips will be discussed in this section:

- Special Mouse Tips
- Lay out a Pipe as a Multi-segmented Polyline
- Make a Pipe into a Multi-segmented Polyline

A.4.1 Mouse Tips

The **right** mouse button can be used to:

- Select units and precision for displaying data.
- Get context-sensitive Help for dialog boxes and data entry fields.
- Open a pop-up context menu of command options for an element.

A.4.2 Laying out a Pipe as a Multi-segmented Polyline

When laying out pipes using the **Pipe Layout** tool, this program will allow you to draw pipes with multiple bends by using the **Control** key on your keyboard.

To draw a pipe with bends:

1. Click the **Pipe Layout** tool to begin laying out your network.
2. Move the mouse to the desired location, and click to insert the first element.
3. The layout tool will rubber-band, indicating that a pipe will be inserted when the next element is added.

In Stand-Alone mode:

1. At this point, hold down the **Control** key. The cursor appearance will change to crosshairs to indicate that pipe bends will be added.
2. While holding the **Control** key down, click to insert any number of pipe bends.
3. When you are through adding pipe bends, release the **Control** key.
4. The appearance of the cursor will change to reflect the next element to be added.
5. Click with the mouse to terminate the pipe and add the next element.

In AutoCAD mode:

1. Select **Bend** from the right-click menu.
2. Draw the pipe as you would draw a polyline.
3. Right-click and select the end node of the pipe.

A.4.3 Changing a Pipe into a Multi-segmented Polyline

To make a straight pipe into a multi-segmented polyline:

In Stand-Alone Mode:

1. Right-click the pipe to which you would like to add a vertex.

2. From the context menu, select the **Bend\Add Bend** menu item.

3. A vertex will be added to the pipe. You may then move the vertex by dragging with the mouse.

In AutoCAD Mode:

1. Select **Edit** from the Main Menu.

2. Select **Modify Pipe**.

3. Select **Insert Bend**.

4. Click the location in the pipe where you want the bend.

5. Use the AutoCAD **Move** command to move the bend in the pipe.

There is no limit on the number of vertices that a pipe may have.

In Stand-Alone mode, you can remove vertices by selecting the pipe. Right-click the vertex you wish to remove, and select the Bend\Remove Bend menu item.

In AutoCAD mode, you can remove vertices by selecting Edit\ Modify Pipes\Remove Bends from the pull-down menu. Select the pipe and the location of the bend that you would like removed.

Notes

Appendix B

WaterCAD Theory

WaterCAD is a state-of-the-art software tool primarily for use in the modeling and analysis of water distribution systems. However, the methodology is applicable to any fluid system that has the following characteristics:

- Steady or gradually-varying turbulent flow.
- Incompressible, Newtonian, single phase fluids.
- Full, closed conduits (pressure systems).

Examples of systems with these characteristics include potable water systems, fire protection systems, well pumps, and raw water pumping.

The WaterCAD algorithms are anticipated to grow and evolve to keep pace with the state of the practice in water distribution and water quality modeling. Because the mathematical solution methods are being continually extended, this manual deals primarily with the fundamental principles underlying these algorithms, and focuses less on the details of the implementation of the algorithms.

Acknowledgements

WaterCAD was designed, developed and programmed by Haestad Methods' staff of Software Engineers and Civil Engineers. This program is intended to represent the latest technology in Windows-based Water Distribution Analysis and Design.

WaterCAD's numerical computations are based on research conducted by the U.S. Environmental Protection Agency (EPA) Drinking Water Research Division, Risk Reduction Engineering Laboratory, its employees and consultants. As a result, WaterCAD will generate results consistent with the EPA computer program "EPANET 1."

B.1 Pressure Network Hydraulics

B.1.1 Network Hydraulics Theory

In practice, pipe networks consist not only of pipes, but of miscellaneous fittings, storage tanks and reservoirs, meters, regulating valves, pumps, and electronic and mechanical controls. For modeling purposes, these system elements are organized into the following categories:

- **Pipes** – Transport water from one location (or node) to another.

- **Junctions Nodes –** Specific points, or nodes, in the system at which an event of interest is occurring. This includes points where pipes intersect, where there are major demands on the system such as a large industry, a cluster of houses, or a fire hydrant, or critical points in the system where pressures are important for analysis purposes.
- **Reservoirs and Tanks –** Boundary nodes with a known hydraulic grade that define the initial hydraulic grades for any computational cycle. They form the baseline hydraulic constraints used to determine the condition of all other nodes during system operation. Boundary nodes are elements such as tanks, reservoirs, and pressure sources.
- **Pumps –** Represented as nodes. Their purpose is to provide energy to the system and raise the water pressure.
- **Valves –** Mechanical devices used to stop or control the flow through a pipe, or to control the pressure in the pipe upstream or downstream of the valve. They result in a loss of energy in the system.

An event or condition at one point in the system can affect all other parts of the system. While this complicates the approach that the engineer must take to find a solution, there are some governing principles that drive the behavior of the network, including the Conservation of Mass and Energy Principle, and the Energy Principle.

The two modes of analysis are Steady-State Network Hydraulics and Extended Period Simulation. Regardless of the mode used, this program solves for the distributions of flows and hydraulic grades using the Gradient Algorithm.

B.1.2 The Energy Principle

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

The energy referred to in this principle represents the total energy of the system minus the sum of the potential, kinetic, and internal (molecular) forms of energy, such as electrical and chemical energy. The internal energy changes are commonly disregarded in water distribution analysis because of their relatively small magnitude.

In hydraulic applications, energy is often represented with units of energy per unit weight, resulting in units of length. Using these length equivalents gives engineers a better feel for the resulting behavior of the system. When using these length equivalents, the state of the system is expressed in terms of head. The energy at any point within a hydraulic system is often represented in three parts:

- Pressure Head: p/γ
- Elevation Head: z
- Velocity Head: $V^2/2g$

Where:

Where:	p	=	Pressure (N/m^2, lb/ft^2)
	γ	=	Specific weight (N/m^3, lb/ft^3)
	z	=	Elevation (m, ft)
	V	=	Velocity (m/s, ft/s)
	g	=	Gravitational acceleration constant (m/s^2, ft/s^2)

These quantities can be used to express the headloss or head gain between two locations using the energy equation.

B.1.3 The Energy Equation

In addition to pressure head, elevation head, and velocity head, there may also be head added to the system, by a pump for instance, and head removed from the system due to friction. These changes in head are referred to as head gains and head losses, respectively. Balancing the energy across two points in the system, we then obtain the energy equation:

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

Where:

Where:	p	=	Pressure (N/m^2, lb/ft^2)
	γ	=	Specific weight of the fluid (N/m^3, lb/ft^3)
	z	=	Elevation at the centroid (m, ft)
	V	=	Fluid velocity (m/s, ft/s)
	g	=	Gravitational acceleration (m/s^2, ft/s^2)
	h_p	=	Head gain from a pump (m, ft)
	h_L	=	Combined headloss (m, ft)

The components of the energy equation can be combined to express two useful quantities, which are the hydraulic grade and the energy grade.

B.1.4 Hydraulic and Energy Grades

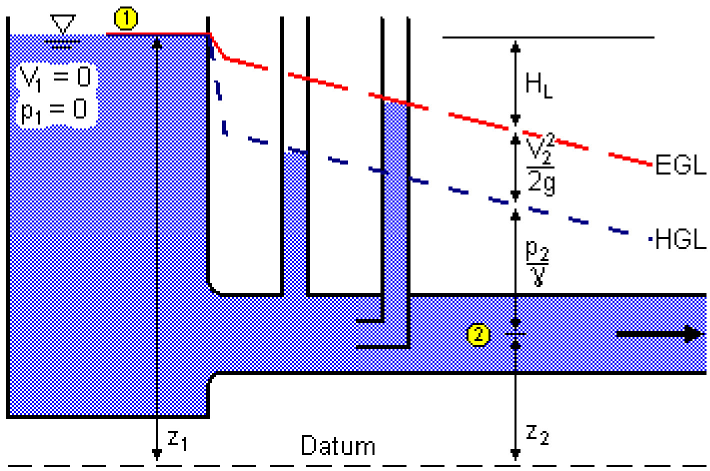
Hydraulic Grade

The hydraulic grade is the sum of the pressure head (p/γ) and elevation head (z). The hydraulic head represents the height to which a water column would rise in a piezometer. The plot of the hydraulic grade in a profile is often referred to as the hydraulic grade line, or HGL.

Energy Grade

The energy grade is the sum of the hydraulic grade and the velocity head ($V^2/2g$). This is the height to which a column of water would rise in a pitot tube. The plot of the hydraulic grade in a profile is often referred to as the energy grade line, or EGL. At a lake or reservoir, where the velocity is essentially zero, the EGL is equal to the HGL, as can be seen in the figure below.

EGL and HGL



B.1.5 Conservation of Mass and Energy

Conservation of Mass

At any node in a system containing incompressible fluid, the total volumetric or mass flows in must equal the flows out, less the change in storage. Separating these into flows from connecting pipes, demands, and storage, we obtain:

$$\sum Q_{IN}\Delta t = \sum Q_{OUT}\Delta t + \Delta V_S$$

Where: Q_{IN} = Total flow into the node (m^3/s, cfs)

Q_{OUT} = Total demand at the node (m^3/s, cfs)

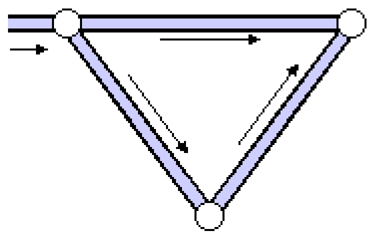
ΔVS = Change in storage volume (m^3, ft^3)

Δt = Change in time (s)

Conservation of Energy

The conservation of energy principle states that the headlosses through the system must balance throughout the loop at each point. For pressure networks, this means that the total headloss between any two nodes in the system must be the same regardless of what path is taken between the two points. The headloss must be sign consistent with the assumed flow direction (i.e. gain head when proceeding opposite the flow and lose head when proceeding with the flow).

Conservation of Energy



The same basic principle can be applied to any path between two points. As shown in the figure above, the combined headloss around a loop must equal zero in order to achieve the same hydraulic grade as at the beginning.

B.1.6 The Gradient Algorithm

The gradient algorithm for the solution of pipe networks is formulated upon the full set of system equations that model both heads and flows. Since both continuity and energy are balanced and solved with each iteration, the method is theoretically guaranteed to deliver the same level of accuracy observed and expected in other well known algorithms such as the Simultaneous Path Adjustment Method (Fowler) and the Linear Theory Method (Wood).

In addition, there are a number of other advantages that this method has over other algorithms for the solution of pipe network systems:

- The method can directly solve both looped and partly branched networks. This gives it a computational advantage over some loop-based algorithms, such as Simultaneous Path, which require the reformulation of the network into equivalent looped networks or pseudo-loops.
- Using the method avoids the post-computation step of loop and path definition, which adds significantly to the overhead of system computation.
- The method is not numerically unstable when the system becomes disconnected by check valves, pressure regulating valves, or modeler’s error. The loop and path methods fail in these situations.
- The structure of the generated system of equations allows the use of extremely fast and reliable sparse matrix solvers.

The derivation of the Gradient Algorithm starts with two matrices and ends as a working system of equations.

B.1.7 Derivation of the Gradient Algorithm

Given a network defined by N unknown head nodes, P links of unknown flow, and B boundary or fixed head nodes, the network topology can be expressed in two incidence matrices:

$A_{12} = A_{21}^T$ (P x N) Unknown head nodes incidence matrix

and

$A_{10} = A_{01}^T$ (P x B) Fixed head nodes incidence matrix

The following convention is used to assign matrix values:

$A_{12}(i,j) =$ 1, 0, or -1 if flow of pipe i enters, is not connected, or leaves node j, respectively.

Assigned nodal demands are given by:

$q^T = [q_1, q_2, \ldots, q_N]$ (1 x N) nodal demand vector

Assigned boundary nodal heads are given by:

$$H_f^{\,T} = [H_{f1}, H_{f2}, ... H_{fB}]$$ (1 x B) fixed nodal head vector

The headloss or gain transform is expressed in the matrix:

$$F^T(Q) = [f_1, f_2, ..., f_P]$$ (1 x P) non-linear laws expressing headlosses in links

$$f_i = f_i(Q_i)$$

These matrix elements that define known or iterative network state can be used to compute the final steady-state network represented by the matrix quantities for unknown flow and unknown nodal head.

Unknown link flow quantities are defined by:

$$Q^T = [Q_1, Q_2, ..., Q_P]$$ (1 x P) unknown link flow rate vector

Unknown nodal heads are defined by:

$$H^T = [H_1, H_2, ..., H_N]$$ (1 x N) unknown nodal head vector

These topologic and quantity matrices can be formulated into the generalized matrix expression using the laws of energy and mass conservation:

$$A_{12}H + F(Q) = -A_{10}H_f$$

$$A_{12}Q = q$$

A second diagonal matrix that implements the vectorized head change coefficients is introduced It is generalized for Hazen-Williams friction losses in this case:

$$A_{11} = \begin{bmatrix} R_1|Q_1|^{n_1-1} & & & & \\ & R_2|Q_2|^{n_2-1} & & & \\ & & ... & & \\ & & & ... & \\ & & & & R_P|Q_P|^{n_P-1} \end{bmatrix}$$

This yields the full expression of the network response in matrix form:

$$\begin{bmatrix} A_{11} & A_{12} \\ A_{21} & 0 \end{bmatrix} \begin{bmatrix} Q \\ H \end{bmatrix} = \begin{bmatrix} -A_{10}H_f \\ q \end{bmatrix}$$

To solve the system of non-linear equations, the Newton-Raphson iterative scheme can be obtained by differentiating both sides of the equation with respect to Q and H to get:

$$\begin{bmatrix} NA_{11} & A_{12} \\ A_{21} & 0 \end{bmatrix} \begin{bmatrix} dQ \\ dH \end{bmatrix} = \begin{bmatrix} -dE \\ dq \end{bmatrix}$$

with

$$N = \begin{bmatrix} n_1 & & & \\ & n_2 & & \\ & & \cdots & \\ & & & n_P \end{bmatrix}$$

The final recursive form of the Newton-Raphson algorithm can now be derived after matrix inversion and various algebraic manipulations and substitutions (not presented here). The working system of equations for each solution iteration, k, is given by:

$$H^{k+1} = -(A_{21}N^{-1}A_{11}{}^{-1}A_{12})^{-1}\{A_{21}N^{-1}(Q^k + A_{11}{}^{-1}A_{10}H_f) + (q - A_{21}Q^k\}$$

$$Q^{k+1} = (1 - N^{-1})Q^k - N^{-1}A_{11}{}^{-1}(A_{12}H^{k+1} + A_{10}H_f)$$

The solution for each unknown nodal head for each time iteration is computationally intensive. This high speed solution utilizes a highly optimized sparse matrix solver that is specifically tailored to the structure of this matrix system of equations.

Sources:

Todini, E. and S. Pilati, "A gradient Algorithm for the Analysis of Pipe Networks", *Computer Applications in Water Supply, Vol. 1 – Systems Analysis and Simulation*, edited by Bryan Coulbeck and Chun-Hou Orr, Research Studies Press LTD, Letchworth, Hertfordshire, England.

B.1.8 The Linear System Equation Solver

The Conjugate Gradient method is one method that, in theory, converges to an exact solution in a limited number of steps. The Conjugate Gradient working equation can be expressed for the pressure network system of equations as:

$$Ax = b$$

where:

$$x = H^{k+1}$$

$$b = -\{A_{21}N^{-1}(Q^k + A_{11}{}^{-1}A_{10}H_f) + (q - A_{21}Q^k\}$$

The structure of the system matrix A at the point of solution is:

$$A = A_{21}(NA_{11})^{-1}A_{12} = A_{21}DA_{12}$$

and it can be seen that the nature of the topological matrix components yields a total working matrix A that is:

- Symmetric
- Positive definite
- Stieltjes type

Because of the symmetry, the number of non-zero elements to be retained in the matrix equals the number of nodes plus the number of links. This results in a low density, highly sparse matrix form. It follows that an iterative solution scheme would be preferred over direct matrix inversion, in order to avoid matrix fill-in which serves to increase the computational effort.

Because the system is symmetric and positive definite, a Cholesky factorization can be performed to give:

$$A = LL^T$$

where L is the lower triangular with positive diagonal elements. Making the Cholesky factorization allows the system to be solved in two steps:

$$y = L^{-1}b$$

$$x = (L^T)^{-1}y$$

The use of this approach over more general sparse matrix solvers that implement traditional Gaussian elimination methods without consideration to matrix symmetry is preferred, since performance gains are considerable. The algorithm utilized in this software solves the system of equations using a variant of Cholesky's method which has been optimized to reduce fill-in of the factorization matrix, thus minimizing storage and reducing overall computational effort.

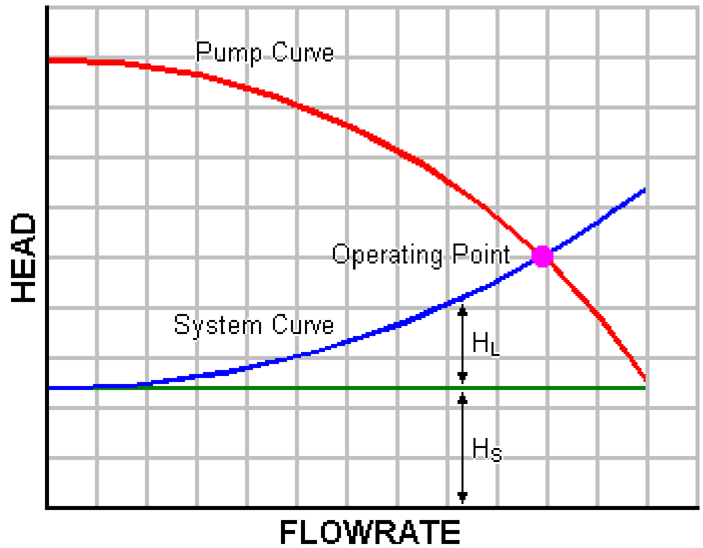
B.1.9 Pump Theory

Pumps are an integral part of many pressure systems. Pumps add energy, or head gains, to the flow to counteract headlosses and hydraulic grade differentials within the system.

A pump is defined by its characteristic curve, which relates the pump head, or the head added to the system, to the flow rate. This curve is indicative of the ability of the pump to add head at different flow rates. To model behavior of the pump system, additional information is needed to ascertain the actual point at which the pump will be operating.

The system operating point is based on the point at which the pump curve crosses the system curve representing the static lift, headlosses due to friction and minor losses. When these curves are superimposed, the operating point can easily be found. This is shown in the figure below.

System Operating Point



As water surface elevations and demands throughout the system change, the static head (Hs) and headlosses (HL) vary. This changes the location of the system curve, while the pump characteristic curve remains constant. These shifts in the system curve result in a shifting operating point over time.

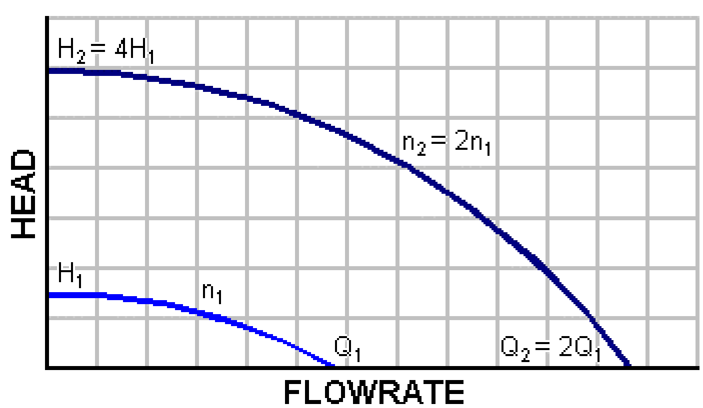
Variable Speed Pumps

A pump's characteristic curve is fixed for a given motor speed and impeller diameter, but can be determined for any speed and any diameter by applying the Affinity Laws. For variable speed pumps, these Affinity Laws are presented as:

$$\frac{Q_1}{Q_2} = \frac{n_1}{n_2} \quad \text{and} \quad \frac{h_1}{h_2} = \left(\frac{n_1}{n_2}\right)^2$$

Where: Q = Pump flowrate (m^3/s, cfs)
h = Pump head (m, ft)
n = Pump speed (rpm)

Effect of Relative Speed on Pump Curve



Constant Horsepower Pumps

During preliminary studies, the exact characteristics of the constant horsepower pump may not be known. In these cases, the assumption is often made that the pump is adding energy to the water at a constant rate. Based on power-head-flowrate relationships for pumps, the operating point of the pump can then be determined. Although this assumption is useful for some applications, a constant horsepower pump should only be used for preliminary studies.

B.1.10 Pump Type

This software currently models six different types of pumps:

- **Design Point (One-Point) –** A pump can be defined by a single design point (Hd, Qd). From this point, the curve's interception with the head and discharge axes is computed as Ho = 1.33•Hd and Qo = 2.00•Qd. This type of pump is useful for preliminary designs, but should not be used for final analysis.
- **Standard (Three-Point) –** This pump curve is defined by three points – the shutoff head (pump head at zero discharge), the design point (as with the single-point pump), and the maximum operating point (the highest discharge at which the pump performs predictably).
- **Standard Extended –** This is the same as the standard three-point pump, but with an extended point at the zero pump head point. This is automatically calculated by the program.
- **Custom Extended –** The custom extended pump is similar to the standard extended pump, but allows you to enter the discharge at zero pump head.
- **Multiple Point** – This option allows you to define a custom rating curve for a pump. The pump curve is defined by entering points for discharge rates at various heads. Since the general pump equation, shown below, is used to simulate the pump during the network computations, the user-defined pump curve points are used to solve for coefficients in the general pump equation:

$$Y = A - (B \times Q^{C})$$

Where:	Y	=	Head (m, ft)
	Q	=	Discharge (m^3/s, cfs)
	A,B,C	=	Pump curve coefficients

The Levenberg-Marquardt Method is used to solve for A, B and C based on the given multiple-point rating curve.

- **Constant Power –** These pumps may be useful for preliminary designs and estimating pump size, but should not be used for any analysis for which more accurate results are desired.

Whenever possible, avoid using constant power or design point pumps. They are often enticing because they require less work on behalf of the engineer, but they are much less accurate than a pump curve based on several representative points.

It is not necessary to place a check valve on the pipe immediately downstream of a pump, because pumps have built in check valves that prevent reverse flow.

B.1.11 Valve Theory

There are several types of valves that may be present in a pressurized system. These valves have different behaviors and different responsibilities, but all valves are used for automatically controlling parts of the system. They can be opened, closed, or throttled to achieve the desired result.

Check Valves (CV's)

Check valves are used to maintain flow in only one direction by closing when the flow begins to reverse. When the flow is in the specified direction of the check valve, it is considered to be fully open. Check valves are added to the network on a pipe element.

Flow Control Valves (FCV's)

A flow control valve limits the flow rate through the valve to a specified value in a specified direction. These valves are commonly found in areas where a water district has contracted with another district or a private developer to limit the maximum demand to a value that will not adversely affect the provider's system.

Pressure Reducing Valves (PRV's)

Pressure reducing valves are often used for separate pressure zones in water distribution networks. These valves prevent the pressure downstream from exceeding a specified level in order to avoid pressures that could have damaging effects on the system.

Pressure Sustaining Valves (PSV's)

Pressure sustaining valves maintain a specified pressure upstream from the valve. Similar to the other regulating valves, these are often used to ensure that pressures in the system (upstream, in this case) will not drop to unacceptable levels.

Pressure Breaker Valves (PBV's)

Pressure breaker valves create a specified headloss across the valve, and are often used for model components that cannot be easily modeled using standard minor loss elements.

Throttle Control Valves (TCV's)

Throttle control valves simulate minor loss elements whose headloss characteristics change over time.

B.2 Friction and Minor Losses

B.2.1 Friction Method Theory

Energy Losses in Pipes

The energy loss (h_L) in a piping system may be due to a combination of several factors. The primary cause of energy loss is due to friction between the fluid and the conduit wall. Since this friction is present throughout the length of any given pipe, the energy grade line and hydraulic grade line drop continuously in the direction of flow.

Secondary causes of energy loss are due to localized areas of increased turbulence and disruption of the streamlines. These disruptions are caused by valves, meters, or fittings, and are called minor losses. These minor losses are often negligible relative to friction losses, and may sometimes be ignored in an analysis. While the term "minor" is a reasonable generalization for most large-scale water distribution models, it may not always be the case. In piping systems that contain numerous fittings relative to the total length of pipe, the minor losses may have a significant impact on the energy losses.

Friction Losses

There are many equations that model friction losses associated with the flow of a liquid through a pressure pipe. The three most commonly used methods are:

- Hazen-Williams
- Darcy-Weisbach (Jain's)
- Manning

The Darcy-Weisbach method is applicable to a wide range of fluids, while the Hazen-Williams and Manning's equations are based on empirical development and are only applicable for water modeling. All of these methods calculate friction losses as a function of the fluid's velocity and a measure of the pipe's resistance to flow (roughness).

Typical pipe roughness values for all of these methods are shown in the Roughness Table at the end of this chapter. Of course, these values may vary depending on the manufacturer, workmanship, age, and many other factors.

B.2.2 Friction Loss Methods

Hazen-Williams Equation

The Hazen-Williams Formula is frequently used in the analysis of pressure pipe systems (such as water distribution networks and sewer force mains). The formula is as follows:

$$Q = k \cdot C \cdot A \cdot R^{0.63} \cdot S^{0.54}$$

Where:
- Q = Discharge in the section (m^3/s, cfs)
- C = Hazen-Williams roughness coefficient (unitless)
- A = Flow area (m^2, ft^2)
- R = Hydraulic radius (m, ft)
- S = Friction slope (m/m, ft/ft)
- k = Constant (0.85 for SI, 1.32 for US).

Darcy-Weisbach Equation

Because of its non-empirical origins, the Darcy-Weisbach equation is viewed by many engineers as the most accurate method for modeling friction losses. It most commonly takes the following form:

$$h_f = f \cdot \frac{L}{D} \frac{V^2}{2g}$$

Where:
- h_f = Headloss (m, ft)
- f = Darcy-Weisbach friction factor (unitless)
- D = Pipe diameter (m, ft)

L = Pipe length (m, ft)
V = Flow velocity (m/s, ft/s)
g = Gravitational acceleration constant (m/s^2, ft/s^2)

For section geometries that are not circular, this equation is adapted by relating a circular section's full-flow hydraulic radius to its diameter:

$$D=4R$$

Where: R = Hydraulic radius (m, ft)

This can then be rearranged to the form:

$$Q = A \cdot \sqrt{8g \cdot \frac{R \cdot S}{f}}$$

Where:
Q = Discharge (m^3/s, cfs)
A = Flow area (m^2, ft^2)
R = Hydraulic radius (m, ft)
S = Friction slope (m/m, ft/ft)
f = Darcy-Weisbach friction factor (unitless)
g = Gravitational acceleration constant (m/s^2, ft/s^2)

The Swamme and Jain equation can then be used to calculate the friction factor.

Swamme and Jain Equation:

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

Where:
f = Friction factor (unitless)
k = Roughness height (m, ft)
D = Pipe diameter (m, ft)
Re = Reynolds number (unitless)

The friction factor is dependent on the Reynolds number of the flow, which is dependent on the flow velocity, which is dependent on the discharge… As you can see, this process requires the iterative selection of a friction factor until the calculated discharge agrees with the chosen friction factor.

The Kinematic Viscosity is used in determining the friction coefficient in the Darcy-Weisbach Friction Method. The default units are initially set by the software.

Manning's Equation

Manning's equation is one of the most popular methods in use today for free surface flow (and, like Kutter's equation, is based on Chezy's equation). For Manning's equation, the roughness coefficient in Chezy's equation is calculated as:

$$C = k \cdot \frac{R^{1/6}}{n}$$

Where:	C	=	Chezy's roughness coefficient ($m^{1/2}/s$, $ft^{1/2}/s$)
	R	=	Hydraulic radius (m, ft)
	n	=	Manning's roughness (unitless)
	k	=	Constant (1.00 $m^{1/3}/s$, 1.49 $ft^{1/3}/s$)

Substituting this roughness C into Chezy's equation, we obtain the well-known Manning's equation:

$$Q = \frac{k}{n} \cdot A \cdot R^{2/3} \cdot S^{1/2}$$

Where:	Q	=	Discharge (m^3/s, cfs)
	k	=	Constant (1.00 $m^{1/3}/s$, 1.49 $ft^{1/3}/s$)
	n	=	Manning's roughness (unitless)
	A	=	Flow area (m^2, ft^2)
	R	=	Hydraulic radius (m, ft)
	S	=	Friction slope (m/m, ft/ft)

Chezy's equation is reviewed later in this section.

Manning's roughness coefficients are the same as the roughness coefficients used in Kutter's equation.

Colebrook-White Equation

The Colebrook-White equation is used to iteratively calculate for the Darcy-Weisbach friction factor:

Free Surface

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

Full Flow (Closed Conduit)

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{k}{12.0R} + \frac{2.51}{R_e\sqrt{f}}\right)$$

Where:	Re	=	Reynolds Number
	k	=	Darcy-Weisbach roughness height (m, ft)
	f	=	Friction factor (unitless)
	R	=	Hydraulic radius (m, ft)

Chezy's Equation

Chezy's equation is rarely used directly, but it is the basis for several other methods, including Manning's equation and Kutter's equation. Chezy's equation is:

$$Q = C \cdot A \cdot \sqrt{R \cdot S}$$

Where:	Q	=	Discharge in the section (m^3/s, cfs)
	C	=	Chezy's roughness coefficient ($m^{1/2}$/s, $ft^{1/2}$/s)
	A	=	Flow area (m^2, ft^2)
	R	=	Hydraulic radius (m, ft)
	S	=	Friction slope (m/m, ft/ft)

B.2.3 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:

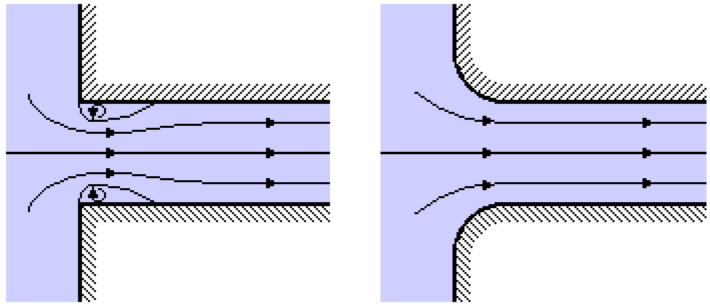
$$h_m = K\frac{V^2}{2g}$$

Where:	h_m	=	Loss due to the minor loss element (m, ft)
	V	=	Velocity (m/s, ft/s)
	g	=	Gravitational acceleration constant (m/s^2, ft/s^2)
	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



B.3 Water Quality Analysis

In the past, water distribution systems were designed and operated with little consideration of water quality, due in part to the difficulty and expense of analyzing a dynamic system. The cost of extensive sampling and the complex interaction between fluids and constituents encourages the use of numerical modeling for predicting water quality.

Water Quality Solution Algorithm

This software implements the Eulerian Discrete Volume-Element method (DVEM) to solve the numerical constituent transport problem for the system being analyzed. The method requires, as basic input, the complete solution for the distribution of flows (magnitude and direction) for the network links and nodes at each discrete time step that occurs over an extended period simulation (EPS). The EPS results generate a sequence of hydraulic steps, which are then reformulated into a distribution of hydraulic states encapsulated within discrete volume-elements by this software's DVEM algorithm. The constituent concentration is determined at each discrete volume-element in the system for every quality time step over the full duration of the simulation. The assumption is made that there is complete mixing across finite distances, such as at a junction node or a short segment of pipe.

The model for solving the constituent transport problem can be used to predict other water quality parameters including age and trace.

B.3.1 First-Order Reaction Rate Function

The first-order reaction rate expression is most commonly used for modeling constituent decay and growth processes in batch reactions. The first-order function has the form:

$$R(C_i) = kC_i$$

Where:

	R	=	Rate of reaction of constituent (mg/l.s)
	k	=	Reaction rate coefficient of concentration decay (negative) or growth (positive) (1/s)
	C_i	=	Concentration of substance in liquid (mg/l)

The reaction coefficient is zero for conservative substances.

Assuming this first-order reaction rate gives a closed form solution to the general constituent transport equation:

$$C_i(x, t+\tau) = C_i(x - u_i\tau, t).e^{k\tau}$$

Where: τ = Water quality time step (s)
u_i = Mean flow velocity of the water (m/s, ft/s)
x = Distance in the pipe (positive in the flow direction) (m, ft)

This software implements a first-order reaction rate model that uses a composite reaction rate coefficient. This effectively simulates reactions occurring in the bulk fluid as well as at the fluid/pipe wall interface.

Sources:

Rossman, Lewis A., et.al., "Discrete Volume-Element Method for Network Water-Quality Models", *Journal of Water Resources Planning and Management*, Vol. 119, No 5, Sep/Oct, 1993.

Rossman, Lewis A. and Paul F. Boulos, "Numerical Methods for Modeling Water Quality in Distribution Systems: a Comparison", *Journal of Water Resources Planning and Management*, Vol. 122, No. 2, Mar/Apr, 1996.

B.3.2 Reaction Rate Model

This software's reaction model recognizes that the growth or decay of a substance is driven by reactions occurring both within the bulk flow component and with the material along the pipe wall. The model uses first-order kinetics to simulate the wall and bulk reactions.

The expression for the general substance decay rate model for each link, *i* , is given below:

$$R(C_i) = -k_b C_i - \left(\frac{k_f}{R_{H_i}}\right)(C_i - C_{w_i})$$

Where: k_b = First-order bulk reaction rate constant (1/s)
C_i = Substance concentration in bulk flow (mg/l)
k_f = Mass transfer coefficient between bulk flow and pipe wall (m/s, ft/s)
R_{Hi} = Hydraulic radius of pipe (Diameter/4) (m, ft)
C_{wi} = Substance concentration at the wall (mg/l)

Assuming the rate of reaction at the wall is first order and no net accumulation of material occurs over the time step, the mass balance at the wall is given by:

$$k_f(C_i - C_{w_i}) = k_w C_{w_i}$$

Where: k_w = Wall reaction rate constant (m/s, ft/s)

Solving for the wall concentration and substituting into the general decay rate equation above results in the general first-order reaction rate expression for substance decay:

$$R(C_i) = -KC_i$$

Where: K = Overall rate constant equal (1/s)

K can be expressed as:

$$K = k_b + \frac{k_w k_f}{R_{H_i}(k_w + k_f)}$$

It follows that dropping the negative sign ahead of *K* in the equation above will model the growth of a substance with mass transfer from the pipe wall to the bulk flow.

In this program, the bulk coefficient and wall reaction rate coefficient are supplied by the user as part of the constituent properties. These are determined empirically using observed reaction rate data. A typical scenario might have the bulk coefficient being derived from jar tests, and the wall coefficient estimated over the course of model calibration by adjusting its value to best match actual observed field concentrations to simulated results.

The mass transfer coefficient is a function of internal pipe hydraulics, and is computed automatically by the constituent transport algorithm.

B.3.3 Mass Transfer Coefficient

The mass transfer coefficient, which is used to model the constituent transfer between the bulk fluid and pipe wall, is calculated internally by the modeling engine using the unitless Sherwood Number as follows:

$$k_f = \frac{S_h D}{d}$$

Where:
- k_f = Mass transfer coefficient (m/s, ft/s)
- D = Molecular diffusivity of substance in fluid (m^2/s, ft^2/s)
- S_h = Sherwood number (unitless)
- d = Pipe diameter (m, ft)

For $R_e \geq 2300$ (turbulent flow):

$$S_h = 0.023 R_e^{0.83} S_c^{0.333}$$

For $R_e < 2300$ (laminar flow):

$$S_h = 3.65 + \frac{0.0668 (d/L) R_e S_c}{1 + 0.04 [(d/L) R_e S_c]^{0.67}}$$

Where:
- Re = Reynolds number (unitless)
- Sc = Schmidt number (ν/d) (unitless)
- L = Pipe length (m, ft)
- q = Flow rate (m^3/s, cfs)
- A = Cross-section flow area of the pipe (m^2, ft^2)

ν = Kinematic viscosity of fluid (m^2/s, ft^2/s)

The molecular diffusivity and the kinematic viscosity are supplied by the user.

Sources:

Edwards, D.K., et.al., *Transfer Processes*, McGraw-Hill, New Your, NY, 1976

Rossman, Lewis A., *EPANET Users Manual*, *Version 1.1*, Drinking Water Research Division, Risk Reduction Engineering Laboratory, USEPA, Cincinnati, Ohio, Jan. 1994.

B.3.4 Accuracy Issues in Water Quality Simulation with this Software

The accuracy of the Discrete Volume-Element method (DVEM) is primarily driven by the size of the water quality time step, τ. Unless the modeler provides a user override of this calculation parameter, this software will automatically compute the time step based upon the minimum pipe travel time in the system. Note that transport across pumps and valves is assumed to be instantaneous.

During each quality step there should be no dispersion, because the contents of the volume-elements are maintained as separate and discrete. However, there may be a blending between hydraulic time steps should the system re-segment to a lower number of volume-elements. This will occur when the system demands are peaking and link velocities are increasing. This artificial dispersion is expected and is a consequence of the methodology.

This program provides several water quality parameter options that the modeler may use to control the segmentation scheme in response to extreme cases in the system. Key situations are described below:

- Short, high-velocity pipes in system – The presence of these link elements might cause the model to assume a very small value for τ, which could lead to excessively long computation times in large systems. This effect can be controlled via the Minimum Pipe Travel Time calculation option. Pipes with hydraulic times shorter than this will experience an artificial delay in constituent mass transport, resulting in some accuracy loss.

- Long, slow-velocity pipes in system – The presence of these elements can result in a large number of volume-elements to be generated during the segmentation phase of the algorithm. This can increase calculation times and place severe demands on the computer system memory. This situation can be controlled by setting an upper limit on the number of volume-elements that are generated using the Maximum Pipe Segments option. Mass transport through links limited by this parameter will occur at a faster rate than in reality, resulting in some loss of accuracy.

This software will generate a Water Quality Status Log that will provide a table of Quality Time Steps and Maximum Number of Pipe Segments to give you some guidance in interpreting the simulation results.

You should remember that because of their approximate nature, computer simulations are inherently inaccurate in some respects. Consequently, a significant aspect of system modeling is in determining just how much modeling inaccuracy is acceptable within the context of the problem being studied. This is frequently governed by calibration input data availability, accuracy, and reliability, over such issues as inherent limitations in models or methodologies. Ultimately, the resolution of accuracy concerns resides solely with the modeler.

B.3.5 Age and Source Trace Analyses

This software uses the built-in Water Quality Constituent transport solver to model the changes in the age of water and to track source fractions reaching any node over time. To accomplish this, the model makes the following assumptions:

- **Age –** The program sets the concentration value, *Ci* , in the basic transport equation to the age of water, and sets the reaction term, *R(Ci),* to zero. Any new water entering the system at any source is assigned an age of zero.
- **Source Trace –** The program tracks the percentage of total flow quantity arriving at any node in the network from a specified contributing source node. In this analysis, the concentration value is set to the percent of flow from the node in question, and the reaction rate is set to zero.

> **Water age gives a general indication of the overall water quality at any given point in the system. Age is typically measured from the time the water enters the system from a reservoir until it reaches a junction.**

The water age is computed as:

$$A_j = A_{j-1} + \frac{x_j}{u_j}$$

Where: A_j = Age of water at *j*-th node (s)
x_j = Distance from node *j*-1 to node *j* (m, ft)
u_j = Velocity from node *j*-1 to node *j* (m/s, ft/s)

If there are several paths for water to travel to the j-th node, the water age is computed as a weighted average as shown (i designates the pipes with an incoming flow to node j):

$$A_j = \frac{\sum_i Q_i \left(A_i + \frac{x_i}{u_i} \right)}{\sum_i Q_i}$$

Where: Q_i = Flow rate in pipe *i* (m^3/s, ft^3/s)
A_i = Age of water at the start node of pipe *i* (s)

> **Identifying the origin of flow at a point in the system is referred to as flow tracking, or trace modeling. In systems that receive water from more than one source, trace studies can be used to determine the origin percentage breakdown of flow at each point. These studies can be very useful for determining the area influenced by an individual source, observing the degree of mixing of water from several sources, and viewing changes in origin over time.**

Source:

Rossman, Lewis A., *EPANET Users Manual*, *Version 1.1*, Drinking Water Research Division, Risk Reduction Engineering Laboratory, USEPA, Cincinnati, Ohio, Jan. 1994.

B.3.6 Constituent Transport Equations

Assuming constant flow and velocity over a time step, the concentration within link *i*, at any point x (in the positive flow direction), and time t, is given by the mass-conservation differential equation:

$$\frac{\partial C_i(x,t)}{\partial t} + u_i \frac{\partial C_i(x,t)}{\partial x} - R[C_i(x,t)] = 0$$

Where: Ci(x,t)	=	Concentration of substance in liquid, at the distance x from the start node of the link *i* (mg/l/)
x	=	Distance in the link measure from the start node of the link (positive in the flow direction) (m, ft)
ui	=	Mean flow velocity of the water (m/s, ft/s)
$R[C_i(x,t)]$	=	Reaction rate function (1/s)

The reaction rate function is typically a first-order rate function.

To obtain a general solution of the basic transport equation, it is necessary to introduce an analytical time step, τ, which serves as the interval of analysis over which substance concentrations are advected within link *i*. This advection distance is expressed as $u_i\tau$. This time step is referred to as the water quality time step.

At nodes, the concentrations are determined by mass-conservation, assuming complete nodal mixing. The concentration at node *k* for each outgoing link *i* (with *j* representing each incoming link at node *k*) is given by:

$$C_i(0,t) = C_k(t) = \frac{\sum_j \left(Q_j C_j(L_j,t)\right) + Q_e C_e}{\sum_j Q_j}$$

Where: $C_k(t)$	=	Concentration at node *k* (mg/l)
L_j	=	Length of link *j* (m, ft)
Q_j	=	Flow rate in link *j* (m^3/s, ft^3/s)
Q_e	=	External source flow into node *k* (m^3/s, ft^3/s)
C_e	=	External source concentration into node *k* (mg/l)

The boundary condition at a variable level tank with an incoming link *i* can be expressed as:

$$C_T(t+\tau) = \frac{1}{V_T(t) + Q_i\tau}\left[C_i(L_i,t)Q_i\tau + V_T(t)C_T(t)\right]$$

Where: $C_T(t)$	=	Concentration of the fully mixed tank (mg/l/)
$V_T(t)$	=	Volume of the tank (mg/l/)

Q_j = Flow rate in link *j* (m^3/s, ft^3/s)

An outgoing link *i* of a tank is assumed to receive the full mixed tank concentration:

$$C_j(0, t + \tau) = C_T(t)$$

These transport equations are solved over the distributed range of volume states that are generated automatically using the water quality solver's DVEM algorithm implementation.

Once the hydraulic model has been solved for the network, the velocities are known and the mixing at nodes is known. Using this information, the water quality behavior can be derived using a numerical method.

B.3.7 Water Quality Modeling Approaches

There are several theoretical approaches available for the solution of water quality models. These methods can generally be grouped as either Eulerian or Lagrangian in nature, depending on the volumetric control approach that is taken. Eulerian, which is the method used by this program, divides the system into fixed pipe segments and then tracks the changes that occur as water flows through these segments. Langrangian models also break the system into control volumes, but then track these water volumes as they travel through the system.

B.3.8 Discrete Volume Element Method (DVEM) Algorithm

The discrete volume-element method is based on a plug-flow reactor assumption. The plug-flow model accounts for advective transport and the kinetics of the constituent reactions within the plug. According to the physical plug-flow reactor model, each reactor plug will be advected through the system and composited with incoming plugs at flow-receiving nodes. In the Eulerian DVEM numerical modeling scheme, this plug movement is simulated by transferring substance concentration state from one discrete volume-element (a plug) to the next adjacent volume-element along the direction of flow.

The DVEM proceeds by subdividing each link in the network into a number of equal-sized elements at every hydraulic event In the simulation, for example, this is a single EPS time step during which flow and velocity patterns are assumed to remain constant. To preserve accuracy, the volumetric element for each link over the duration of every hydraulic event must be correctly computed. To this end, the methodology is parameterized by the water quality time step, τ. To ensure that fluid is not transported beyond the confines of any link within a single analytical step, the total link volume must be less than the product of link discharge and time step. It follows that τ cannot exceed the shortest travel time through any network link over the analysis interval, as expressed below (with i representing all links in the network):

$$\tau = \min_i \left(\frac{V_i}{Q_i} \right)$$

Where: V_i = Total volume of liquid in link *i* (m^3, ft^3)
Q_j = Flow rate in link *i* (m^3/s, ft^3/s)

Once τ has been established, the number of volume elements and the volume of each element in a pipe *i* can be calculated as follows:

$$n_i = \frac{V_i}{Q_i \tau}$$

$$v_i = \frac{V_i}{n_i}$$

Where: n_i = Number of volume elements in link *i*

v_i = Volume of each volume element in link *i* (m^3, ft^3)

To develop a water quality simulation based on a series of EPS hydraulic intervals, the network links are individually partitioned into component volume-elements for each hydraulic time step. Ideally, this partitioning should honor the numerical constraints defined above. Under extreme conditions this may not be possible, and the modeler may wish to modify the segmentation scheme by setting practical limits on quality time step, maximum number of segments, or a limiting hydraulic time interval.

The constituent mass is propagated through the system network over each water quality time step in four phases:

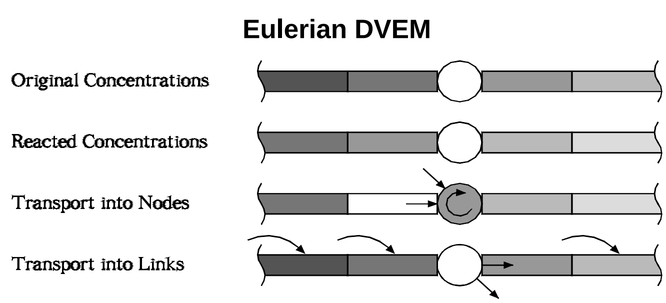
6. **Kinetic reaction** – The mass concentration undergoes a kinetic concentration change by applying the kinetic reaction function.

7. **Nodal mixing –** The constituent mass and incoming volumes are mixed at nodes.

8. **Advection –** The constituent mass is transferred between volume-elements.

9. **Allocation –** The nodal mass is assigned to the first volume-element of all outgoing links.

These steps are graphically summarized below:



B.4 Engineer's Reference

This chapter provides you with tables of commonly used roughness values and fitting loss coefficients

Roughness Values

- Roughness Values, Manning's Equation
- Roughness Values, Darcy-Weisbach (Colebrook-White) Equation
- Roughness Values, Hazen-Williams Equation

- Typical Roughness Values for Pressure Pipes

Coefficients

- Fitting Loss Coefficient

B.4.1 Roughness Values, Manning's Equation

Commonly used roughness values for different materials are:

Manning’s Coefficients n for Closed Metal Conduits Flowing Partly Full

Channel Type and Description	Minimum	Normal	Maximum
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

B.4.2 Roughness Values, Darcy-Weisbach Equation (Colebrook-White)

Commonly used roughness values for different materials are:

Darcy-Weisbach Roughness Heights k for Closed Conduits

Pipe Material	**k (mm)**	**k (ft)**
Glass, drawn brass, copper (new)	0.0015	0.000005
Seamless commercial steel (new)	0.004	0.000013
Commercial steel (enamel coated)	0.0048	0.000016
Commercial steel (new)	0.045	0.00015
Wrought iron (new)	0.045	0.00015
Asphalted cast iron (new)	0.12	0.0004
Galvanized iron	0.15	0.0005
Cast iron (new)	0.26	0.00085
Wood Stave (new)	0.18 ~ 0.9	0.0006 ~ 0.003
Concrete (steel forms, smooth)	0.18	0.0006
Concrete (good joints, average)	0.36	0.0012
Concrete (rough, visible, form marks)	0.60	0.002
Riveted steel (new)	0.9 ~ 9.0	0.003 – 0.03
Corrugated metal	45	0.15

B.4.3 Roughness Values, Hazen-Williams Formula

Commonly used roughness values for different materials are:

Hazen-Williams Roughness Coefficients C

Pipe Material	C
Asbestos Cement	140
Brass	130-140
Brick sewer	100
Cast-iron	
New, unlined	130
10 yr. Old	107-113
20 yr. Old	89-100
30 yr. Old	75-90
40 yr. Old	64-83
Concrete or concrete lined	
Steel forms	140
Wooden forms	120
Centrifugally spun	135
Copper	130-140
Galvanized iron	120
Glass	140
Lead	130-140
Plastic	140-150
Steel	
Coal-tar enamel, lined	145-150
New unlined	140-150
Riveted	110
Tin	130
Vitrified clay (good condition)	110-140
Wood stave (average condition)	120

B.4.4 Typical Roughness Values for Pressure Pipes

Typical pipe roughness values are shown below. These values may vary depending on the manufacturer, workmanship, age, and many other factors.

Comparative Pipe Roughness Values

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

B.4.5 Fitting Loss Coefficients

For similar fittings, the K-value is highly dependent on things such as bend radius and contraction ratios.

Typical Fitting K Coefficients

Fitting	K Value	Fitting	K Value
Pipe Entrance		90° Smooth Bend	
Bellmouth	0.03 - 0.05	Bend radius / D = 4	0.16 - 0.18
Rounded	0.12 - 0.25	Bend radius / D = 2	0.19 - 0.25
Sharp Edged	0.50	Bend radius / D = 1	0.35 - 0.40
Projecting	0.80		
		Mitered Bend	
Contraction - Sudden		$\theta = 15°$	0.05
$D_2/D_1 = 0.80$	0.18	$\theta = 30°$	0.10
$D_2/D_1 = 0.50$	0.37	$\theta = 45°$	0.20
$D_2/D_1 = 0.20$	0.49	$\theta = 60°$	0.35
Contraction - Conical		$\theta = 90°$	0.80
$D_2/D_1 = 0.80$	0.05		
$D_2/D_1 = 0.50$	0.07	Tee	
$D_2/D_1 = 0.20$	0.08	Line Flow	0.30 - 0.40
		Branch Flow	0.75 - 1.80
Expansion - Sudden			
$D_2/D_1 = 0.80$	0.16	Cross	
$D_2/D_1 = 0.50$	0.57	Line Flow	0.50
$D_2/D_1 = 0.20$	0.92	Branch Flow	0.75
Expansion - Conical			
$D_2/D_1 = 0.80$	0.03	45° Wye	
$D_2/D_1 = 0.50$	0.08	Line Flow	0.30
$D_2/D_1 = 0.20$	0.13	Branch Flow	0.50

Appendix C

Scenario Management Guide

Haestad Methods' scenario management feature can dramatically increase your productivity in the "W*hat If?"* areas of modeling, including calibration, operations analysis, and planning.

By investing a little time now to understand scenario 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.

In contrast to the old manual 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.

C.1 About this Guide

As implemented by Haestad Methods, the depth of scenario management is probably far beyond what you have ever seen before. With that in mind, this guide is intended as an introduction to the philosophy and terminology upon which scenario management is based.

This is not intended as a step-by-step guide to using the software. If you are a moderately experienced Windows software user, you should have no difficulty learning and exploring the scenario management interface.

Excellent tutorials and context-sensitive on-line help are also available within the software itself. These learning tools will prove to be of tremendous assistance to you for all aspects of the software, and should certainly not be ignored if you are having difficulty. For more information, just click the **Help** button, which is available from anywhere within the program.

In addition, call Haestad Methods for the schedule of workshops.

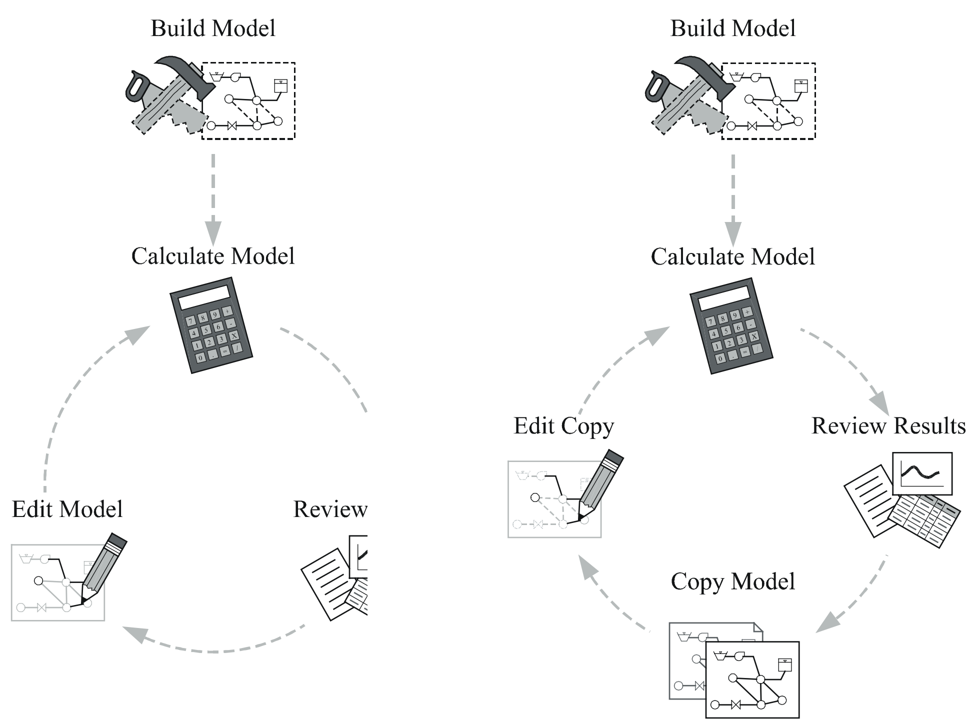
C.2 Before Haestad Methods: Manual Scenarios

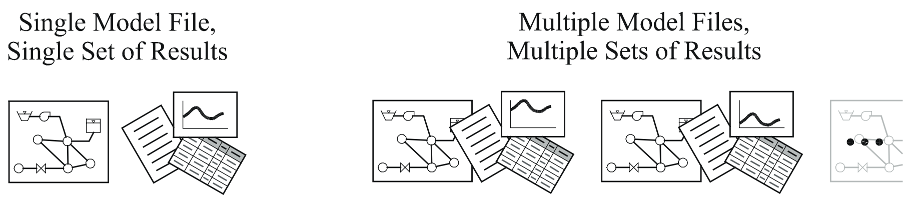
Let us begin by understanding the approaches that have historically been used to attempt "W*hat If?"* analyses. Traditionally, there have only been two possible ways of analyzing the effects of change on a software simulation:

- Change the model, recalculate, and review the results
- Create a copy of the model, edit that copy, calculate, and review the results

Although either of these methods may be adequate for a relatively small system, the data duplication, editing, and re-editing becomes very time-consuming and error-prone as the size of the system – and the number of possible conditions – increase. Additionally, comparing conditions requires manual data manipulation, because all output must be stored in physically separate data files.

Before Haestad Methods: Manual Scenarios





C.3 With Haestad Methods: Self-Contained Scenarios

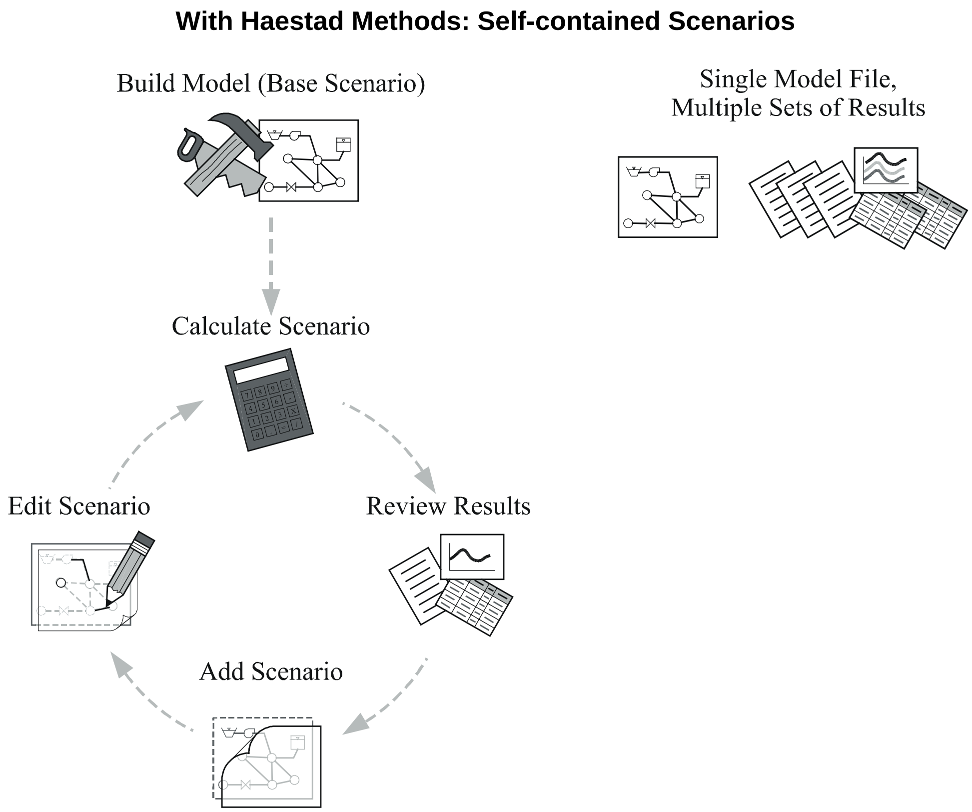
Effective scenario management tools need to meet these objectives:

- Minimize the number of project files the modeler needs to maintain (one, ideally).
- Maximize the usefulness of scenarios through easy access to resources 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 developed by Haestad Methods successfully meets all of these objectives. A single project file enables you to generate an unlimited number of "W*hat If?"* conditions, edit only the data that needs to be changed, and quickly generate direct comparisons of input and results for desired scenarios.

C.4 The Scenario Cycle

The process of working with scenarios is similar to the process of manually copying and editing data, but without the disadvantages of data duplication and troublesome file management. This process allows you to cycle through any number of changes to the model, without fear of overwriting critical data or duplicating important information. 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.



C.5 Scenario Anatomy: 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.

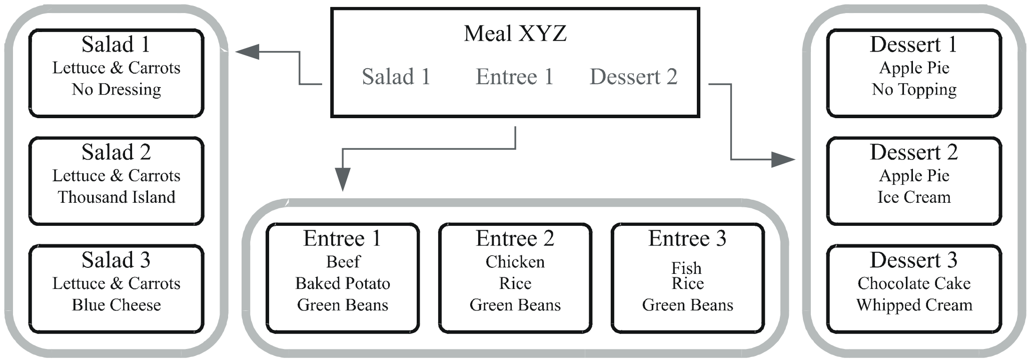
Alternative – An alternative holds a family of related attributes so pieces of data that you are most likely to change together are grouped for easy referencing and editing. For example, a physical properties alternative groups physical data for the network's elements, such as elevations, sizes, and 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.

C.6 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:

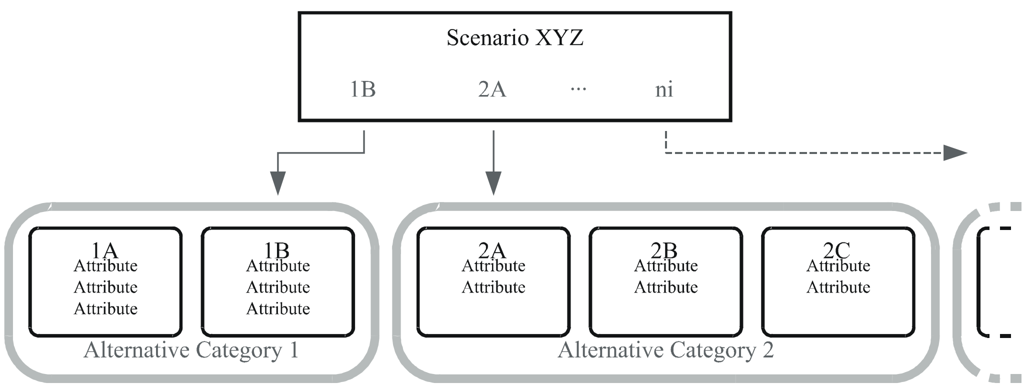
A Restaurant Meal "Scenario"



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 simply references one alternative from each category to create a "big picture" that can be analyzed. Note that different types of alternatives may have different numbers and types of attributes, and any category can have an unlimited number of alternatives to choose from.

Generic Scenario Anatomy



C.7 Scenario Behavior: Inheritance

The separation of scenarios into distinct alternatives (groups of data) meets one of the basic goals of scenario management: maximizing the number of scenarios you can develop by mixing and matching existing alternatives. Two other primary goals have also been addressed: a single project file is used, and easy access to input data and calculated results is provided in numerous formats through the intuitive graphical interface.

But what about the other objective: minimizing the amount of data that needs to be duplicated to consider conditions that have a lot of common input? Surely an entire set of pipe diameters should not be re-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
- Dynamic inheritance

C.8 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.

C.9 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.

C.10 When are values local, and when are they inherited?

Any **changes** that are made to the model belong to the currently active scenario and the alternatives that it references. If the alternatives happen to have children, those children will also inherit the changes unless they have specifically overridden that attribute. 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.

A Mid-level Hierarchy Alternative Change

Alternative Hierarchy	Value
Condition 1	100
└ Condition 2	100
└ Condition 3	100
└ Condition 4	100
└ Condition 5	150

→ *Change Condition 3 Value to 200* →

Alternative Hierarchy	Value
Condition 1	100
└ Condition 2	100
└ Condition 3	*200*
└ Condition 4	200
└ Condition 5	150

C.11 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 simply create a child from *x* and change *y* appropriately. Now we have both conditions, with no duplicated effort.

We can even apply this inheritance to our restaurant analogy as follows (inherited values are shown as gray text, local values are shown as black text):

Salad Alternative Hierarchy	Attribute: Vegetables	Attribute: Dressing
Salad 1	Lettuce & 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."

Salad 3 could inherit from Salad 2, if we prefer: "Salad 3 is just like Salad 2, except for the dressing."

Entree Alternative Hierarchy	Attribute: Meat	Attribute: Starch	Attribute: Vegetable
Entree 1	Beef	Baked Potato	Green Beans
└ Entree 2	Chicken	Rice	Green Beans
└ Entree 3	Fish	Rice	Green Beans

- "Entrée 2 is just like Entrée 1, except for the meat and the starch."
- "Entrée 3 is just like Entrée 2, except for the meat."

If the vegetable of the day changes (say from green beans to peas), only Entrée 1 needs to be updated, and the other entrées will automatically inherit the vegetable attribute of "Peas" instead of "Green Beans".

Dessert Alternative Hierarchy	Attribute: Bakery Item	Attribute: Topping
Dessert 1	Apple Pie	No Topping
└ Dessert 2	Apple Pie	Ice Cream
Dessert 3	Chocolate Cake	Whipped Cream

- "Dessert 2 is just like Dessert 1, except for the topping."

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.

C.12 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):

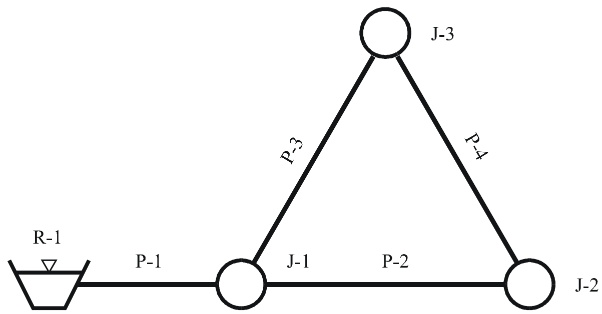
Meal Scenario Hierarchy	Salad Alternative	Entree Alternative	Dessert Alternative
Meal 1	Salad 1	Entree 2	Dessert 3
└ Meal 2	Salad 1	Entree 2	Dessert 1
Meal 3	Salad 3	Entree 3	Dessert 2
└ Meal 4	Salad 2	Entree 3	Dessert 2

- "Meal 2 is just like Meal 1, except for the dessert." The salad and entrée alternatives are inherited from Meal 1.
- "Meal 3 is nothing like Meal 1 or Meal 2." A totally new "base" or "root" is created.
- "Meal 4 is just like Meal 3, except for the salad." The entrée and dessert alternatives are inherited from Meal 3.

C.13 A Water Distribution Example

Let us consider a fairly simple water distribution system: a single reservoir supplies water by gravity to three junction nodes.

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.

C.14 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*

C.15 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.

C.16 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.

C.17 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 will ripple through to the children. After the single change is made, the demand hierarchy is as follows:

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day ├ 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.

C.18 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 └ *Larger P-1* *Larger All Pipes*	8 inches *18 inches* *12 inches*	6 inches 6 inches *12 inches*	6 inches 6 inches *12 inches*	6 inches 6 inches *12 inches*

Scenario Hierarchy	Demand Alternative	Physical Alternative

Avg. Day
├ Max. Day
└ Peak
   ├ *Peak, Big P-1*
   └ *Peak, All Big Pipes*
	Average Day
	Maximum Day
	Peak Hour
	Peak Hour
	Peak Hour
		Preliminary Pipes
		Preliminary Pipes
		Preliminary Pipes
		Larger P-1
		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 dialogs) can be used to directly determine which proposal results in the most improved pressures.

C.19 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

C.20 Conclusion

These are the fundamental concepts behind the architecture of Haestad Methods' scenario management. To learn more about actually using scenario management in Haestad Methods 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.

Haestad Methods' scenario management feature gives you a powerful tool for modeling real-world engineering scenarios when analyzing system response to different demands, reviewing the impacts of future growth, and iterating to find the least expensive design. That means you will be able to finish your projects faster, spend less money, and improve your bottom line.

Appendix D

Haestad Methods Software

Haestad Methods offers software solutions to Civil Engineers throughout the world for analyzing, modeling, and designing water distribution networks, storm and sanitary sewer networks, watersheds, culverts, hydraulic structures, open channels, and more. All software is Windows based, with point-and-click data entry, flexible units, and report quality outputs.

In addition to the ability to run in Stand-Alone mode with a CAD-like interface, three of our products – WaterCAD, StormCAD and SewerCAD – 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 **www.haestad.com** to find out about our latest software, books, training, and open houses.

D.1 WaterCAD

WaterCAD is a powerful, easy-to-use program that helps engineers design and analyze complex pressurized distribution pipe network systems. You can use WaterCAD to perform a variety of functions, including steady-state analyses of water distribution systems with pumps, tanks, and control valves.

WaterCAD is also capable of completing extended period simulations to analyze a piping system's response to varying supply and demand schedules, including automated fire flow analyses. Water quality elements, such as water source, age, and chemical constituent growth and decay can be tested utilizing WaterCAD's water quality simulation and analysis functions.

Complex networks of pipes, tanks, pumps, and more can be laid out quickly and easily, whether you are using the Stand-Alone Windows graphical interface, or running WaterCAD for AutoCAD. The Scenario Manager allows you to create multiple sets of alternatives and run any number of scenarios, then view and compare the results.

D.2 SewerCAD

SewerCAD is a powerful design and analysis tool that allows you to lay out a sanitary collection system, develop and compute sanitary loads, and simulate the hydraulic response of the entire system, including gravity collection piping and pressure force mains. The program can be run within AutoCAD, giving you all the power of AutoCAD's capabilities, or in Stand-Alone mode utilizing Haestad Methods' own Windows graphical interface.

SewerCAD allows you to construct a graphical representation of a pipe network, containing all your information such as pipe data, pump data, loading, and infiltration. Conveyance element choices include circular pipes, pipe arches, boxes, and more.

SewerCAD's flexible load model allows you to analyze dry and wet weather loads separately. Unit dry loads and peaking factor methods are entirely customizable. The gravity network is solved using the built-in numerical model, which utilizes both the direct step and standard step gradually varied flow methods. Flow calculations are valid for both pressure and varied flow situations, and will solve for hydraulic jumps, backwater, and drawdown curves. SewerCAD's flexible reporting feature allows you to customize and print the model results in any type of report, including plan and profile views for analysis, construction documents, or agency review.

D.3 StormCAD

StormCAD allows you to efficiently design and analyze storm sewer systems. Layout tools allow you to construct a graphical representation of a pipe network containing all of your information, including pipe data, inlet characteristics, watershed areas, and rainfall data.

StormCAD provides a choice of conveyance elements including circular pipes, pipe arches, boxes, and more. Inlets are designed and computed using the new FHWA HEC-22 methodology. Junction hydraulic losses may be calculated automatically using the AASHTO or HEC-22 methodology based on the geometry of the junction. Rainfall information is defined using Intensity-Duration-Frequency (IDF) in the form of rainfall tables or rainfall equations. Flows are then calculated using the rational method.

The gravity network is solved using the built-in numerical model, which utilizes both the direct step and standard step gradually varied flow profile computation methods. Flow calculations are valid for both pressure and varied flow situations, and will solve for hydraulic jumps, backwater, and drawdown curves. StormCAD's flexible reporting feature allows you to customize and print the model results in any type of report, including plan and profile views for analysis, construction documents, or agency review.

D.4 PondPack

PondPack is a powerful, comprehensive, Windows-based hydrologic modeling program that analyzes a tremendous range of situations, from simple sites to complex networked watersheds. The program 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.

This program can utilize any rainfall duration or distribution. In a single run, it computes hydrographs for multiple storm events, adds them at junctions, and routes them through reaches and ponds. PondPack graphically displays many items, including watershed diagrams, hydrographs, rainfall curves, and IDF curves. All graphics are fully AutoCAD compatible via DXF export.

PondPack builds customized reports organized by categories, and automatically creates 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.

D.5 CulvertMaster

CulvertMaster helps engineers design new culverts and analyze existing culvert hydraulics, from single barrel to complex multi-barrel culverts with roadway overtopping. CulvertMaster computations use FHWA HDS-5 methodology. This allows you to solve for most hydraulic variables, including culvert size, flow, and headwater, as well as generate output for rating curves and tables showing computed flow characteristics.

CulvertMaster provides a choice of culvert barrel shapes, including circular pipes, pipe arches, boxes and more. Flow calculations handle pressure and varied flow situations such as backwater and drawdown curves. Design discharge can be calculated using either the Rational or SCS Graphical Peak Discharge method. Rainfall information is calculated using rainfall tables, equations, or the National Weather Services' Hydro-35 data. CulvertMaster's flexible reporting feature allows you to print the design and analysis results in report format, or as a graphical plot.

D.6 FlowMaster

FlowMaster is an efficient and powerful program for the design and analysis of pipes, ditches, open channels, weirs, orifices, and inlets. FlowMaster's "Hydraulics Toolbox" can solve or create rating tables/curves for any variables using the Manning's, Hazen-Williams, Kutter's, Darcy-Weisbach, and Colebrook-White formulas. FlowMaster's new inlet computations strictly comply with the latest FHWA Hydraulic Circular Number 22 (replacing Circular 12) and AASHTO inlet computation guidelines.

FlowMaster offers you the same powerful tools that can be found in WaterCAD, SewerCAD and StormCAD, such as FlexUnits, FlexTables, and automatic generation of professional quality reports and graphs.

Notes

References

Benedict, R. P., *Fundamentals of Pipe Flow*, John Wiley and Sons, Inc., New York, 1980.

Brater, Ernest F. and Horace W. King, *Handbook of Hydraulics*, McGraw-Hill Book Company, New York, 1976.

Cesario, A. Lee, *Modeling, Analysis, and Design of Water Distribution Systems*, AWWA, 1995.

Clark, R. M., W. M. Grayman, R. M. Males, and A. F. Hess, "Modeling Contaminant propagation in Drinking Water Distribution Systems", *Journal of Environmental Engineering*, ASCE, New York, 1993.

Computer Applications in Hydraulic Engineering, Third Edition, Haestad Press, Connecticut, 1999.

CulvertMaster User's Guide, Connecticut, Haestad Methods, 1999.

Essential Hydraulics and Hydrology, Connecticut, Haestad Press, 1998.

FlowMaster PE Version 6.0 User's Guide, Connecticut, Haestad Methods, 1998.

Males R. M., W. M. Grayman and R. M. Clark, "Modeling Water Quality in Distribution System", *Journal of Water Resources Planning and Management*, ASCE, New York, 1988.

Practical Guide to Hydraulics and Hydrology, Connecticut, Haestad Press, 1997.

Roberson, John A., John J. Cassidy, and Hanif M. Chaudhry, *Hydraulic Engineering*, Houghton Mifflin Company, Massachusetts, 1988.

Roberson, John A. and Clayton T. Crowe, *Engineering Fluid Mechanics (4th Edition)*, Houghton Mifflin Company, Massachusetts, 1990.

Rossman, Lewis A., *EPANet User's Manual (AWWA Workshop Edition)*, Risk Reduction Engineering Laboratory, Office of Research and Development, USEPA, Ohio, 1993.

Rossman, Lewis A. *et al.*, "Numerical Methods for Modeling Water Quality in Distribution Systems: A Comparison", *Journal of Water Resources Planning and Management*, ASCE, New York, 1996.

Rossman, Lewis A., R. M. Clark, and W. M. Grayman, "Modeling Chlorine Residuals in Drinking-water Distribution Systems", *Journal of Environmental Engineering*, ASCE, New York, 1994.

Sanks, Robert L., *Pumping Station Design*, Butterworth-Heinemann, Inc, Massachusetts, 1989.

Streeter, Victor L. and Wylie, E. Benjamin, *Fluid Mechanics*, McGraw-Hill Book Company, New York, 1985.

Todini, E. and S. Pilati, "A Gradient Algorithm for the Analysis of Pipe Networks", *Computer Applications in Water Supply, Volume 1 – Systems Analysis and Simulation*, ed. By Bryan Coulbeck and Chun-Hou Orr, Research Studies Press LTD, Letchworth, Hertfordshire, England.

Walski, Thomas M., *Water System Modeling Using CYBERNET®*, Haestad Methods, 1993.

Zipparro, Vincent J. and Hasen Hans, *Davis' Handbook of Applied Hydraulics*, McGraw-Hill Book Company, New York, 1993.

Glossary

Age – Length of time the water has been in the system, which is used as a general water quality indicator than a measurement of anything specific.

Available Fire Flow – Amount of flow available at a node for fire protection while maintaining all fire flow pressure constraints.

Base Elevation and Level – Elevation from which all tank levels are measured. For example, a tank level of two meters represents a water surface elevation two meters above the base elevation.

Boundary Node – Node with a known hydraulic grade. It may be static (unchanging with time), such as a reservoir, or dynamic (changes with time), such as a tank. Every pipe network must contain at least one boundary node. In order to compute the hydraulic grade at the other nodes in the network, boundary nodes must be reachable from a boundary.

Bulk Reaction Coefficient – Coefficient used to define how rapidly a constituent grows or decays over time. It is expressed in units of 1 / time.

Calculated Minimum System Pressure – See minimum system pressure.

Calculated Minimum Zone Pressure – Minimum calculated pressure of all junctions in the same zone as the node where fire flow withdrawal occurs.

Calculated Residual Pressure – Calculated pressure at the junction node where the fire flow withdrawal occurs.

Calculation Unready – An element that does not have all the required information for performing an analysis.

C-Coefficient – Roughness coefficient used in the Hazen-Williams Equation.

Check Valve – Prevents water from flowing backwards through the pipe. In other words, water can only flow from the *From Node* to the *To Node*.

Closed/Inactive Status – See valve status.

Constituent – Any substance, such as chlorine or fluoride, for which the growth or decay can be adequately described through the use of a bulk reaction coefficient and a wall reaction coefficient.

Context Menu – A pop-up menu opened by right-clicking a project element or data entry field. Commands on the context menu are specific to the current state of the selected item.

Control Status – A pressure pipe can be either opened or closed. Opened means that flow occurs in the pipe, and closed means that no flow occurs in the pipe.

Conveyance Element – A pipe or channel used to transport water.

Coordinates – Distances perpendicular to a set of reference axes. Some areas may have predefined coordinate systems, while other coordinate systems may be arbitrary. Coordinates may be presented as X- and Y-values or may be defined as Northing and Easting values, depending on individual preferences.

Cross Section Type – Tanks can have either a constant area cross section or a variable area cross section. The cross section of a tank with a constant area is the same throughout the depth. The cross section of a tank with a variable area varies throughout the depth.

Crosshairs – The cursor consisting of two lines that look like a plus sign (+).

Current Storage Volume – The volume of water currently stored in a tank. It includes both the hydraulically active volume and the hydraulically inactive volume.

Database – See external file.

Database Connection – A connection between WaterCAD and a data source which is represented by a group of database links. There may be a single linked external file within a connection, or there may be several external file links within a single connection.

DBMS – An acronym that stands for Database Management System.

Demand – The total demand from an individual junction for the current time period. It is based on the information from the **Demand** tab of the **Junction Editor**.

Design Point – Point at which a pump was originally intended to operate, and is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.

Diameter – The distance between two internal points directly opposite each other in a pipe or a valve.

Discharge – Volumetric rate of flow given in units of $length^3$/time.

Double-Click – To click the left mouse button twice in rapid succession.

Drag – To hold down one of the mouse buttons while you move the mouse.

Element – An object such as a tank, junction node, or pipe in a drawing.

Elevation – The distance from a datum plane to the center of the element. Elevations are often referenced with mean sea level as the datum elevation.

Energy Grade Line (EGL) – Sum of datum (base elevation), elevation, velocity head, and pressure head at a section.

Extended Edit Button – A small button with an ellipsis (...) as the label. Extended edit buttons are located next to drop-down choice lists, and provide further editing for the associated choice list items.

External File – Any file outside of this program that can be linked. These include database files (such as FoxPro, Dbase or Paradox) and spreadsheets (such as Excel or Lotus). Throughout the documentation, all of these file types will be referred to as "databases" or "external files" interchangeably.

Extrapolate – To estimate the value of an unknown point based on other known values, with the desired point lying outside the known range. Often based upon extending the slope of the line connecting the previous known values to the desired point. See also: interpolate.

Field Links – The actual mapping between model element attributes and columns within each database table.

File Extension – The period and three characters, typically, at the end of a filename. A file extension usually identifies the kind of information the file contains. For example, files you create in AutoCAD have the extension *.DWG.

Fire Flow Upper Limit – The maximum allowable fire flow that can occur at a withdrawal location. This is a user-specified practical limit that will prevent this program from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures. Remember that a system's ability to deliver fire flows is ultimately limited by the size of the hydrant opening and service line, as well as the number of hydrants available to combat a fire at a specific location.

Flow – The calculated value of the pipe, valve, or pump discharge at the given time.

From Node – The pipe's starting node. Positive flow rates are in the direction of "from" towards "to". Negative flow rates are in the opposite direction.

From Pipe – The pipe that connects to the upstream side of a valve or pump.

Headloss – The energy lost due to friction and minor losses. The headloss field displays the pipe, valve, or pump's total headloss at the given time.

Headloss Gradient – The headloss in the pipe as a slope, or gradient. This allows you to more accurately compare headlosses for pipes of different lengths.

Hydraulic Grade – Elevation to which water would rise under zero pressure. For open surfaces, such as reservoirs and tanks, this is equal to the water surface elevation. The hydraulic grade field presents the hydraulic grade for the element at the current time period as calculated based on the system flow rates and head changes.

Hydraulic Grade Setting – The constraint to which a valve regulates, expressed in units of head (Length). Depending on the type of valve, it may refer to either the upstream or downstream hydraulic grade or the headloss across the valve.

Inactive Volume – The volume of water below the minimum elevation of the tank. This volume of water is always present, even when the tank reaches its minimum elevation and closes itself off from the system. Therefore, it is hydraulically inactive. It is primarily used for water quality calculations.

Inflow and Outflow – An inflow is a flow into a node from the system, while an outflow is a flow from the node into the system. A negative outflow is the same as a positive inflow, and a negative inflow is the same as a positive outflow.

Inheritance – The parent-child relationship used by scenarios and alternatives. Just as in the natural world, inheritance is used to refer to the situation where an entity receives something from its parent. For example, we speak of a child inheriting blue eyes from a parent. Unlike in the natural world, inheritance in scenarios and alternatives is dynamic. If the parent's attribute changes, the child's attribute automatically changes at the same time, unless the value is explicitly changed in a child.

Initial Settings – The status of an element for a steady-state analysis or the first time step in an extended period simulation. The initial settings for a pipe, pump, or valve can be set using the elemental dialogs or a table.

Initial Water Quality – The starting conditions at a node determined by age, trace, or constituent concentration. The initial value will be slightly different depending on the analysis type.

Interpolate – Estimating a value of an unknown point *between* two known points assuming a linear relationship. See also: extrapolate.

Invert – Lowest point of a pipe opening. Sometimes referred to as the flow-line.

Label – The unique name by which an element will be referenced in reports, error messages, and tables.

Length – The distance from a pipe's *From Node* to its *To Node*, according to the scaled length of the pipe. To enter an overriding length, click the *User Defined Length* field and type in your desired length value.

Manning's Coefficient – Roughness coefficient used in Manning's Formula.

Material – The selection of a pipe's construction material. This material will be used to determine a default value for the pipe's roughness.

Maximum Elevation – The highest allowable water surface elevation in a tank. If the tank fills above this point, it will automatically shut off from the system.

Maximum Extended Operating Point – The absolute maximum discharge at which a pump can operate, with zero head being added to the system. This value may be computed by the program or entered manually.

Maximum Operating Point – The highest discharge for which a pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.

Messages – The section of an element editor that contains information generated during the calculation of the model, such as warnings, errors, and status updates.

Messages Light – A light that appears on the Tab of the Messages sheet. The light will be red if errors occurred during the analysis, yellow if there are warnings or cautions, and green if there are no warnings or cautions.

Minimum Elevation – The lowest allowable water surface elevation in a tank. If the tank drains below this point, it will automatically shut off from the system.

Minimum System Junction – The junction where the calculated minimum system pressure occurs.

Minimum System Pressure – The minimum pressure allowed at any junction in the entire system as result of fire flow withdrawal. If the pressure at a node anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied. A fire flow analysis may be configured to ignore this constraint.

Minimum Zone Junction – The junction where the calculated minimum zone pressure occurs.

Minimum Zone Pressure – The minimum pressure to occur at all junction nodes within a Zone. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone associated with it, which can be specified in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another Zone.

Minor Loss – The field that presents the total minor loss K-value for a pipe or valve. If an element has more than one minor loss, each can be entered individually by clicking the **Ellipsis (…)** button.

Mouse Buttons – The left mouse button is the primary button for selecting or activating commands. The right mouse button is used to activate pop-up context menus and help. Note that the mouse button functions can be redefined using the *Windows Control Panel*.

Needed Fire Flow – The flow rate required at a junction to satisfy fire flow demands.

Network Element – An element that forms part of the network model. Annotation elements such as polylines, borders and text, are not network elements.

Number – The number of parallel conveyance elements in a model.

Notes – The field that allows you to enter text relevant to the model. It may include a description of an element, a summary of your data sources, or any other information of interest.

ODBC – An acronym that stands for Open Database Connectivity (ODBC), a standard programming interface developed by Microsoft for accessing data in relational and non-relational database management systems (DBMSs).

On/Off Status – The status of a pump can be either on or off. On means that flow will occur in the downstream direction, and the pump will add head to the system according to its characteristic curve. Off means that no flow will occur, and no head will be added.

Open/Closed Status – The status of a pipe can be either open or closed. Open means that flow can occur in either direction. Closed means that no flow will occur through the pipe.

Outflow – See inflow and outflow.

Percent Full – The ratio of the current storage volume to the total storage volume, multiplied by 100.

Pipe Status – Indicates whether the pipe is open or closed. As input, this determines how the pipe begins the simulation. As output, it shows the calculated status of the pipe at the given time.

Polyline – A composite element that consists of a series of line segments. Each line segment begins and ends at a vertex. A vertex may be another element such as a junction, tank, or pump.

Power – The water horsepower of a pump that is transferred from the pump into the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.

Pressure – The field that displays the pressure for the current time period.

Pressure Setting – The constraint to which a valve regulates, expressed in units of pressure (Force per $Length^2$). Depending on the type of valve, it may refer to either the upstream or downstream pressure or the pressure drop.

Pull-Down Menu – A menu of available commands or actions you can perform. A pull-down menu is usually selected from the menu bar at the top of the main program window.

Pump Status – A pump can have two different status conditions: On, which is normal operation, or Off, which is no flow under any condition.

RDBMS – See DBMS.

Relative Speed Factor – The characteristics of a pump relative to the speed for which the pump curve was entered, in accordance with the affinity laws. A speed factor of 1.00 would indicate pump characteristics identical to those of the original pump curve.

Residual Pressure – The minimum residual pressure to occur at a junction node. The program determines the amount of fire flow available such that the residual pressure at a junction node does not fall below this target pressure.

Reynold's Number – Ratio of viscous forces relative to inertial forces. A high Reynold's number indicates turbulent flow, while a low number indicates laminar flow.

Roughness – A measure of a pipe's resistance to flow. Pipes of different ages, construction material, and workmanship may have different roughness values.

Roughness Coefficient - A value used to represent the resistance of a conveyance element to flow. In the Manning's equation, this value is inversely proportional to flow. The smaller the roughness coefficient, the greater the flow.

Satisfies Fire Flow Constraints – A true or false statement indicating whether or not this junction node meets the fire flow constraints. A checkmark in the box means the Fire Flow Constraints were satisfied for that node. If there is no checkmark, the Fire Flow Constraints were NOT satisfied.

Select – The process of adding one or more elements to an active selection set.

Selection Set – The active group of selected elements. A selection set allows editing or an action, such as move or delete, to be performed on a group of elements.

Shape – The cross-sectional geometric form of a conveyance element (i.e. circular, box, arch, etc).

Shutoff Point – The point at which a pump will have zero discharge. Typically the maximum head point on a pump curve.

Size – Inside diameter of a pipe section for a circular pipe. See also diameter.

Starting Elevation – The value that is used as the beginning condition for an extended period simulation.

Status Pane – The area at the bottom of the window used for displaying status information.

Storage Node – Special type of node where a free water surface exists, and the hydraulic head is simply the elevation of the water surface above sea level.

Sub Menu – A list of related options that is typically reached by selecting a pull-down menu item.

Table Links – A table link must be created for every database table or spreadsheet worksheet that is to be linked to the current model. Any number of Table Links may reference the same database file.

To Node – A pipe's ending node. Positive flow rates are in the direction of "from" towards "to". Negative flow rates are in the opposite direction.

To Pipe – The pipe that connects to the downstream side of a valve or pump.

Total Active Volume – The volume of water between minimum elevation and maximum elevation of a tank. This is an input value for variable area tanks.

Total Needed Fire Flow – If you choose to add the fire flow to the baseline demand, the Total Needed Fire Flow is equal to the Needed Fire Flow plus the baseline demand. If you choose NOT to add the fire flow to the baseline demand, the Total Needed Fire Flow is equal to the Needed Fire Flow.

Total Storage Volume – The holding capacity of a tank. It is the sum of the maximum hydraulically active storage volume and the hydraulically inactive storage volume.

Trace (Source Identification) – The percentage of water at any given point originated at a chosen tank, reservoir, or junction.

Valve Status – A valve can have several different status conditions: Closed (no flow under any condition), Active (throttling, opening, or closing dependent on system pressures and flows), and Inactive (wide open, with no regulation).

Velocity – The field that displays the calculated value for a pipe, valve, or pump velocity at a given time. It is found by dividing the element's flow rate by its cross-sectional area.

Vertex – An element in a topological network.

Wall Reaction Coefficient – The rate at which a substance reacts with the wall of a pipe, and is expressed in units of length/time.

Water Quality – The field that displays the water quality for the current time period.

Water Quality Analysis – An analysis that can be one of three types: Age, Trace, or Constituent.

Index

A

B

C

D

E

F

G

H

I

J

K

L

M

S

T

U

V

W

X

Y

Z